Appendix 2 Error Information & Solution

Here describes the information error types and its solutions.

(1) Function block error information

Error code		Error Description	Solutions
Hex	Dec		
0005	5	The current motion controller does not support the function block.	This command is not performed in the current version of the controller. Please contact customer support team of our company after checking the version in which the command can be executed.
0006	6	The axis number (Axis input) or encoder number (Encoder input) of the function block exceeded the allowable range.	Set axis and encoder numbers with a range by product.
0007	7	The axis group number (AxesGroup input) of the function block exceeded the allowable range.	Set the group axis number to be between 1 and 16.
0008	8	The NC channel of the function block exceeded the allowable range.	Check the range of the NC channel and set it again.
0009	9	The slave number (Slave input) of the function block exceeded the allowable range.	Check the range of the slave number and set it again.
000B	11	The input of the function block exceeded the allowable range.	Check the input range of the function block and set it again.
000C	12	The array input of the function block exceeded the allowable range.	Check the size of the array input of the function block and set it again.
0012	18	Error block internal execution error occurred during execution of the function block.	The problem can arise in the current controller version. Please check the support version of XG5000 and controller.
0013	19	Motion response error occurred during execution of the function block.	The problem can arise in the current controller version. Please check the support version of XG5000 and controller.
0014	20	It exceeded the allowable range of the cam ID (CamTableID input) of the function block.	Check the range of the cam ID and set it again.
0015	21	Jog operation was stopped by another command.	If re-execution of jog operation is necessary, turn Enable input off and then on.

(2) System error information

Error code		Error Description	Solutions
Hex	Dec		
000E	14	System Error	Request for A/S if it occurs repeatedly even when the power is supplied again.
0017	23	Program Error	Start the program after modifying and re-loading the program
0018	24	IO Configuration Parameter Abnormality	Check the preservation status after uploading I/O parameter. If it is broken, modify and re-download it to check the operation. If there is still abnormality, exchange it.
0019	25	Basic Parameter Abnormality	Check the preservation status after uploading basic parameter. If it is broken, modify and re-download it to check the operation. If there is still abnormality, exchange it.
001D	29	Special Module Parameter Abnormality	Check the preservation status after uploading special module parameter. If it is broken, modify and re-download it to check the operation. If there is still abnormality, exchange it.
0027	39	CPU Abnormal Termination or Failure	System has been terminated abnormally due to noise and hardware abnormalities. 1) Request for A/S if it occurs repeatedly even when the power is supplied again. 2)Implement noise countermeasures
002B	43	Built-in Parameter-Encoder Abnormality	Check the preservation status after uploading built-in parameter. If it is broken, modify and re-download it to check the operation. If there is still abnormality, exchange it.
002C	44	Axis Parameter Abnormality	Modify the parameter and re-download it
002D	45	Axis Group Parameter Abnormality	Modify the parameter and re-download it
002E	46	EtherCAT Parameter Abnormality	Modify the parameter and re-download it
002F	47	NC Parameter Abnormality	Modify the parameter and re-download it
0030	48	NC Program Inspection Error	Check the program and re-download it
0032	50	Major Failure Detection Error of External Equipment	Repair the wrong equipment and restart it by referring to major failure detection error flag of the external equipment(depending on the parameter)
0038	56	Main Task Cycle Error	Check the main task cycle flag, re-download it after modifying the main task program or download it by increasing the main task cycle of the basic parameter
0039	57	Periodic Task Cycle Error	Check the periodic task cycle flag, re-download it after modifying the periodic task program or download it by increasing the periodic task cycle of the basic parameter
003A	58	Task Program Occupancy Rate Excess Error	1) Secure the time for the system internal service to operate by reducing the amount of user program execution in the main task/periodic task. 2) Secure the time for the system internal service to operate by setting the execution cycle of the main task/periodic task of the basic parameter to be higher
003B	59	Local Ethernet Parameter Inspection Error	Modify the parameter and re-download it
01F5	501	RTC Data Abnormality	Reset it using RTC clock function. If it occurs repeatedly, place requests for A/S

(3) Data log, SD additional function error information

Error code		Error Description	Solutions
Hex	Dec		
Overall Error Code			
0001	1	SD Card Recognition Error	Format it to FAT32 and connect to SD memory
0002	2	Partition Information Error	Format it to FAT32 and connect to SD memory
0003	3	File System Error	Format it to FAT32 and connect to SD memory
0004	4	Unsupported SD Card	Connect SD card with a capacity of 2GB to 32GB
0005	5	SD Card Capacity Check Error	The SD memory capacity test failed, and thus DS cannot be used. Replace SD memory or re-connect it after formatting
0006	6	SD Card Capacity Excess	SD memory capacity is used up, and data storage is not possible. Replace SD memory or re-connect it after formatting. If the available capacity is less than 20%
0007	7	Folder Creation Failed	Failed to create data log folder in SD. Replace SD memory or re-connect it after formatting
Error Code by Data Log Group			
1000	4096	Group x Folder Creation Error	Format it to FAT32 and connect to SD memory
2000	8192	Group x File Open Error	Format it to FAT32 and connect to SD memory
4000	16384	Group x File Write Error	Format it to FAT32 and connect to SD memory
SD Additional Function Error Code			
0001	1	File Error(File Open Failure, CRC Error)	Operate it after creating the file again
0002	2	Damaged File (Damages to Head and Tail, etc.)	Operate it after creating the file again
0005	5	No Password in File	The password is set in the PLC, but there is no password in the file stored in the SD card. Set the password and create a file.
0006	6	Password Mismatch	The password set in the PLC does not match the password of the file stored in the SD card. Confirm it again after checking the password.
0007	7	MAC Address Mismatch	The set MAC address does not match the MAC address of the PLC. Check the MAC address and reset it.
000A	10	No Saved File	There is no file saved in the SD card. Operate it after creating a file.
000B	11	PLC Mode Is RUN State	Check it after switching PLC mode to STOP.

(4) Analog error information

Error code		Error Description	Solutions
Hex	Dec		
0064	0100	Range setting error of input channel 0	Set the range that can be set
0065	0101	Range setting error of input channel 0	Set the range that can be set
00C8	0200	Filter value setting error of input channel 0	Set the filter value that can be set
00C9	0201	Filter value setting error of input channel 0	Set the filter value that can be set
012C	0300	Average value setting error of input channel 0	Set the average value that can be set
012D	0301	Average value setting error of input channel 0	Set the average value that can be set
0190	0400	Range setting error of output channel 0	Set the output range that can be set
0191	0401	Range setting error of output channel 0	Set the output range that can be set
01F4	0500	Input value setting error of output channel 0	Set the input value that can be set
01F5	0501	Input value setting error of output channel 0	Set the input value that can be set
0258	0600	Interpolation method range setting error of output channel 0	Set the interpolation method range that can be set
0259	0601	Interpolation method range setting error of output channel 0	Set the interpolation method range that can be set

(5) Cnet error information

1) XGT server error information

Error code		Error Description	Actions
Hex	Dec		
0003	0003	Block number excess error	Set the block number as 16 or less.
0004	0004	Variable length error	Set the variable length as 16 or less.
0007	0007	Data type error	Set a data type as X, B, W, D and L.
0011	0017	Data error	Set the data type, area, and length that can be set.
0090	0144	Monitor execution error	Register a monitor for execution.
0190	0400	Monitor execution error	When executing a monitor, set the range of the registration number requested by a client to the settable input range.
0290	0656	Monitor registration error	When registering a monitor, set the range of the registration number requested by a client to the settable input range.
1132	4402	Device memory error	Set it as an available device.
1232	4658	Data size error	Set the data size to 60 words or less.
1234	4660	Spare frame error	Remove unnecessary contents of the spare frame.
1332	4914	Data type mismatch error	Set all blocks to the same data type.
1432	5170	Data value error	Check if the data value can be changed to Hex.
7132	28978	Variable request area excess error	Set a device to a usable area.

2) Modbus server error information

Error code		Error Description	Actions
Hex	Dec		
0001	0001	Function code error	The function code that is not supported by a server device. Check whether the Modbus server supports the function code.
0002	0002	Address error	Set the address range to the one supported by a server device.
0003	0003	Data setting error	Set the address type to the one supported by a server device.
0004	0004	Server station abnormality error	Check the error status of the server (slave) station.
0005	0005	Request a server station to resend	There is a lot of processing on the server, which cannot be processed. Request to resend at the processable time on the client side.
0006	0006	Server station processing time delay	It takes times for the server station to process. The client side must make the request again.

3) P2P client error information

Error code		Error Description	Actions
Hex	Dec		
0005	0005	P2P block time out error	Check the state of the server connection and media setting.

4) PLC CPU error information

Error code		Error Description	Actions
Hex	Dec		
0015	0021	P2P Client timeout error	PLC does not respond within 5 seconds after making a P2P request to a communication module. Check the state of a communication module.
0016	0022	P2P client device error	A wrong device area was used. Reset the device.

(5) Motion error information

Error code		Error Description	Solutions
Hex	Dec		
0E00	3584	Command data range transmitted from XG5000 was out of the allowed value.	The problem can arise in the current controller version. Please check the support version of XG5000 and controller.
0E01	3585	The XG5000 test operation function cannot be executed if the controller is in the RUN state.	Execute the test operation of XG5000 after changing controller to STOP state.
0E02	3586	Cam data cannot be written if there is an axis in operation.	Write the cam data while all axes are not in operation.
0E03	3587	Encoder parameters cannot be written if there is an axis in operation.	Write the encoder parameters while all axes are not in operation.
0E04	3588	EtherCAT parameters cannot be written while EtherCAT communication is being connected.	Rewrite the EtherCAT parameters after disconnecting the EtherCAT communication.
0E10	3600	Encoder parameter data is abnormal.	Download the data again from XG5000 and place requests for A/S if the error occurs repeatedly after re-execution.
0E11	3601	Encoder 1 pulse input type of encoder parameter exceeded the setting range.	Set the encoder 1 pulse input of the encoder parameter to be between 0 and 5.
0E12	3602	Encoder 1 maximum value of encoder parameter was out of the range of pulse unit expression value.	Set the encoder 1 maximum value of the encoder parameter in the range of -2,147,483,648 to 2,147,483,647 when converted in pulse unit.
0E13	3603	Encoder 1 minimum value of encoder parameter was out of the range of pulse unit expression value.	Set the encoder 1 minimum value of the encoder parameter in the range of -2,147,483,648 to 2,147,483,647 when converted in pulse unit.
0E14	3604	Encoder 1 maximum value and minimum value of encoder parameters exceeded the range.	Set the encoder 1 minimum value of the encoder parameter to be smaller than the maximum value.
0E15	3605	Encoder 2 pulse input type of encoder parameter exceeded the range.	Set the encoder 2 pulse input of the encoder parameter to be between 0 and 5.
0E16	3606	Encoder 2 maximum value of encoder parameter was out of the range of pulse unit expression value.	Set the encoder 2 maximum value of the encoder parameter in the range of -2,147,483,648 to 2,147,483,647 when converted in pulse unit.
0E17	3607	Encoder 2 minimum value of encoder parameter was out of the range of pulse unit value.	Set the encoder 2 minimum value of the encoder parameter in the range of -2,147,483,648 to 2,147,483,647 when converted in pulse unit.
0E18	3608	Encoder 2 maximum value and minimum value of encoder parameters exceeded the range.	Set the encoder 2 minimum value of the encoder parameter to be smaller than the maximum value.
0E19	3609	Encoder input settings cannot be made above encoder settings of encoder parameter.	Confirm the encoder-related items of the encoder parameter and set values within the range.
0E1A	3610	The number of pulses per rotation of encoder 1 of encoder parameter exceeded the setting range.	Set the number of pulses per rotation of encoder 1 of encoder parameter to be greater than 0 and less than or equal to 4294967295.
0E1B	3611	The transfer distance per rotation of encoder 1 of encoder parameter exceeded the setting range.	Set the transfer distance per rotation of encoder 1 of encoder parameter to be more than 0.000000001 and less than or equal to 4294967295.
0E1C	3612	The number of pulses per rotation of encoder 2 of encoder parameter exceeded the setting range.	Set the number of pulses per rotation of encoder 2 of encoder parameter to be greater than 0 and less than or equal to 4294967295.
0E1D	3613	The transfer distance per rotation of encoder 2 of encoder parameter exceeded the setting range.	Set the transfer distance per rotation of encoder 2 of encoder parameter to be more than 0.000000001 and less than or equal to 4294967295.

Error code		Error Description	Solutions
Hex	**Dec**		
0E1E	3614	Encoder 1 input filter value of encoder parameter exceeded the setting range.	Set the encoder 1 input filter value of encoder parameter to a value between 0 and 6.
0E1F	3615	Encoder 2 input filter value of encoder parameter exceeded the setting range.	Set the encoder 2 input filter value of encoder parameter to a value between 0 and 6.
0E20	3616	Encoder 1 maximum value and minimum value of encoder parameters are set not to include the current position of encoder 1.	Set the range of encoder 1 minimum and maximum values of encoder parameters to include the current position of encoder 1. Or, in order to operate with the set parameters, change the current position of encoder to the value within the parameter by using the encoder preset command.
0E21	3617	Encoder 2 maximum value and minimum value of encoder parameters are set not to include the current position of encoder 2.	Set the range of encoder 2 minimum and maximum values of encoder parameters to include the current position of encoder 2. Or, in order to operate with the set parameters, change the current position of encoder to the value within the parameter by using the encoder preset command.
0E22	3618	Encoder 1 position latch value of encoder parameter exceeded the setting range.	Set the encoder 1 position latch value of encoder parameter to a value between 0 and 1.
0E23	3619	Encoder 2 position latch value of encoder parameter exceeded the setting range.	Set the encoder 2 position latch value of encoder parameter to a value between 0 and 1.
0E30	3632	EtherCAT parameter data is abnormal.	Download the data from XG5000 again and place requests for A/S if the error occurs repeatedly after re-execution.
0E31	3633	The periodic communication timeout count of the EtherCAT parameter exceeded the range.	Set the periodic communication timeout count of the EtherCAT parameter to be between 1 and 8.
0E32	3634	Error occurred during the EtherCAT parameter parsing	Check the EtherCAT parameters and set them again.
0E40	3648	The connection command cannot be executed beyond the EtherCAT parameter.	Check the EtherCAT parameter and set it again.
0E41	3649	The EtherCAT slave connect command is running.	Check if the EtherCAT slave connect command has been entered again while the EtherCAT slave connect command is running.
0E42	3650	The EtherCAT slave disconnect command is running.	Check whether the EtherCAT disconnect command has been entered again while the EtherCAT slave disconnect command is running.
0E43	3651	The connection/disconnection command cannot be executed due to mode switch.	Check whether the mode switch has been executed during the EtherCAT slave connection/disconnection command operation.
0E44	3652	The connection/disconnection command cannot be executed with the ESTOP command.	Check whether the ESTOP command was executed during the EtherCAT slave connection/disconnection command operation.
0E50	3664	The encoder preset command cannot be executed due to abnormal encoder parameter.	Confirm the encoder-related items of the encoder parameter, heck if they are set to values within the range and set the encoder parameter to normal values by using XG5000.
0E51	3665	The preset command cannot be executed because there is an axis that operates with the encoder as the main axis.	Check if the encoder preset command has been entered when there is an axis that operates with the encoder as the main axis.
0E52	3666	The encoder preset position was out of the range of maximum value or minimum value of the encoder.	Set the encoder preset position to a range that is greater than or equal to the minimum value, and smaller than or equal to the maximum value of the encoder.
0E53	3667	The encoder selection of encoder preset	Set the encoder selection to be between 0 and 1 (0: Encoder 1,

Error code		Error Description	Solutions
Hex	Dec		
		command exceeded the range.	1: Encoder 2).
0E54	3668	As the built-in ENC1 is set to the encoder of the spindle axis that is automatically controlled by the NC function module, the encoder's current position setting command cannot be executed.	Check if the built-in ENC1 is set to the encoder connected to the spindle axis in the item 'Spindle Encoder Selection' of the axis parameter.
0E55	3669	As the built-in ENC2 is set to the encoder of the spindle axis that is automatically controlled by the NC function module, the encoder's current position setting command cannot be executed.	Check if the built-in ENC2 is set to the encoder connected to the spindle axis in the item 'Spindle Encoder Selection' of the axis parameter.
0E60	3680	The command cannot be executed due to the basic parameter data abnormality.	Download the basic parameter again from XG5000 and place requests for A/S if 'the basic parameter error' occurs repeatedly after re-execution.
0E61	3681	Cam data is abnormal.	Download the data again from XG5000 and place requests for A/S it is occurs repeatedly after re-execution.
0E90	3728	An error occurred while running the full axis command MC_PowerAll.	Check if the relevant axis is in a state where the command can be executed, and execute the command again.
0E91	3729	An error occurred while executing the entire axis command MC_HomeAll.	Check if the relevant axis is in a state where the command can be executed, and execute the command again.
0E92	3730	An error occurred while running the full axis command LS_HomeAll.	Check if the relevant axis is in a state where the command can be executed, and execute the command again.
0E93	3731	An error occurred while executing the entire axis command MC_StopAll.	Check if the relevant axis is in a state where the command can be executed, and execute the command again.
0E94	3732	An error occurred while running the full axis command MC_HaltAll.	Check if the relevant axis is in a state where the command can be executed, and execute the command again.
0E95	3733	An error occurred while executing the full axis command MC_Reset2All.	Check if the relevant axis is in a state where the command can be executed, and execute the command again.
0E96	3734	An error occurred while executing the entire axis command MC_SetPositionAll.	Check if the relevant axis is in a state where the command can be executed, and execute the command again.
0F00	3840	It failed to change to EtherCAT INIT state.	Check the communication cable status and slave operation status (power-on and error occurrence). And check whether the communication cable is exposed to noise.
0F06	3846	It is EtherCAT INIT state initialization (DC_INIT) error.	Check the communication cable connection status and slave operation status(power-on and error occurrence). And check whether the communication cable is exposed to noise.
0F09	3849	There is no EtherCAT slave connected to the controller.	Check whether the communication cable between the controller and the EtherCAT slave is properly installed, the power is normally applied to the EtherCAT slave, or the communication cable is exposed to noise if there is a slave connected to the controller.
0F0A	3850	It exceeded the maximum number of connected slaves.	Make sure that there are not more than 64 EtherCAT slaves connected to the controller.
0F0E	3854	There is a difference in node ID and EtherCAT parameter settings of the EtherCAT slave.	Check whether the order of the network cable connection between the controller and the EtherCAT slaves matches the EtherCAT parameter settings.
0F0F	3855	There is an error in setting node ID of the EtherCAT slave.	Check whether there are duplicate node IDs or errors in settings.

Error code		Error Description	Solutions
Hex	Dec		
0F10	3856	It failed to change to EtherCAT PREOP stage.	Check the communication cable connection status and slave operation status (power-on and error occurrence). And check whether the communication cable is exposed to noise.
0F1E	3870	There is no slave setting data of EtherCAT parameter.	Set the slave of the EtherCAT parameter using XG5000, and then send the EtherCAT parameter to the controller.
0F1F	3871	The slave setting data of the EtherCAT parameter and the connected slave are different.	Set the slave setting of the EtherCAT parameter to match the actually connected slave information. The slave of the EtherCAT parameter can be set automatically using the 'EtherCAT Slave Auto Connection' function in XG5000.
0F20	3872	It failed to change to EtherCAT SAFEOP state.	Check the communication cable connection status and slave operation status (power-on and error occurrence). And check whether the communication cable is exposed to noise.
0F30	3888	It failed to change to EtherCAT OP state.	Check the communication cable connection status and slave operation status (power-on and error occurrence). And check whether the communication cable is exposed to noise.
0F40	3904	It failed to change from EtherCAT OP state to INIT state.	Check the communication cable connection status and slave operation status (power-on and error occurrence). And check whether the communication cable is exposed to noise.
0F50	3920	There is no response in a communication connection state.	Check the communication cable connection status and slave operation status (power-on and error occurrence). And check whether the communication cable is exposed to noise.
0F51	3921	Periodic communication error occurred. (Communication error that exceeds the periodic communication timeout count of master parameter occurs)	Check whether servo power is off during communication, communication cable is properly installed, or communication cable is exposed to noise.
0F52	3922	A periodic communication error occurred. (The AlStatus error occurred on the slave device)	Check the AL Status Code of the slave device.
0F60	3936	The slave device address (Adp) setting value of ESC read command exceeded the range.	Check the slave device address (Adp) range according to the EtherCAT command code(EcatCmd) setting value to set it.
0F61	3937	The data size setting value of the ESC read command exceeded the range.	Set the data size setting value of the ESC read command to 1 ~ 4 (BYTE).
0F62	3938	EtherCAT command code(EcatCmd) setting value of the ESC read command was incorrect.	Set the EtherCAT command code to one among 1(APRD), 4(FPRD) and 7(BRD).
0F63	3939	There was no response from the slave device for the ESC read command.	Check whether the salve device designated as Adp is installed properly, or the Ado address value is in the read-permission area.
0F64	3940	ESC read command cannot be executed when the slave is in Init state	Change the status of the slave to PreOP, SafeOP, or OP, then execute again.
0F70	3952	The slave device address (Adp) setting value of ESC write command exceeded the range.	Check the slave device address (Adp) range according to the EtherCAT command code(EcatCmd) setting value to set it.
0F71	3953	The data size setting value of the ESC write command exceeded the range.	Set the data size setting value of the ESC read command to 1 ~ 4 (BYTE).
0F72	3954	EtherCAT command code(EcatCmd) setting value of the ESC write command was incorrect.	Set the EtherCAT command code to one among 2(APWR), 5(FPWR) and 8(BWR).
0F73	3955	There was no response from the slave device for the ESC write command.	Check whether the salve device designated as Adp is installed properly, or the Ado address value is in the read-permission area.

Error code		Error Description	Solutions
Hex	Dec		
0F74	3956	You cannot write the specified area to the ESC address (Ado) during execution of the communication connection/disconnection command or in the communication connection state.	Set the ESC address (Ado) that can be written during execution of the communication connection/disconnection command or in the communication connection state.
0F75	3957	ESC write command cannot be executed when the slave is in Init state.	Change the status of the slave to PreOP, SafeOP, or OP status and then execute again.
0FF2	4082	The normal operation in relation to encoder input cannot be executed due to controller H/W abnormality.	Place requests for A/S if it occurs repeatedly when the power is supplied again.
1000	4096	The axis is not ready for driving. (It is not connected to EtherCAT network.)	Execute the command when the axis is ready for operation.
1001	4097	It cannot be executed in "Disabled" state.	Check the operable axis status of the command and execute the command when the command can be run.
1002	4098	It cannot be executed in "Standstill" state.	Check the operable axis status of the command and execute the command when the command can be run.
1003	4099	It cannot be executed in "Discrete" state.	Check the operable axis status of the command and execute the command when the command can be run.
1004	4100	It cannot be executed in "Continuous" state.	Check the operable axis status of the command and execute the command when the command can be run.
1005	4101	It cannot be executed in "Synchronized" state.	Check the operable axis status of the command and execute the command when the command can be run.
1006	4102	It cannot be executed in "Homing" state.	Check the operable axis status of the command and execute the command when the command can be run.
1007	4103	It cannot be executed in "Stopping" state.	Check the operable axis status of the command and execute the command when the command can be run.
1008	4104	It cannot be executed in "Errorstop" state.	Check the operable axis status of the command and execute the command when the command can be run.
100A	4106	Motion command cannot be executed when the belonging axis group is active.	Execute the command after changing the axis group to GroupDisabled state with the axis group disable command.
100B	4107	This command cannot be given to a virtual axis.	The command cannot be executed on a virtual axis. Check whether the command is executed on the virtual axis.
100C	4108	The command cannot be executed if it is registered as an NC channel/axis and is in NC control operation.	Check the operable axis status of the command and execute the command when the command can be run.
100D	4109	The command cannot be executed because the axis is not enabled.	Check whether the setting axis of the command is registered in axis parameter. The axis can be registered in the axis parameter among the motion data items of XG5000.
100E	4110	It is changed to 'run' state during execution of motion test run command, and thus the operation cannot continue.	Check whether the controller was changed to the 'run' state while the axis is running.
100F	4111	The axis operation cannot continue because the controller is stopped by the ESTOP command.	Check whether the controller was stopped by the ESTOP command during the axis operation.
1010	4112	The controller was changed to 'Stop' or 'Error' state, and thus operation cannot continue.	Check whether the controller has been changed to 'Stop' or 'Error' state while the axis is running.
1011	4113	The EtherCAT network connection was lost, and thus operation cannot continue.	Check whether the EtherCAT network connection has been disconnected due to slave power supply error, network cable

Error code		Error Description	Solutions
Hex	Dec		
			error and noise inflow on network cable while the axis is running.
1012	4114	The position setting value of the command was out of the range of pulse unit expression value.	It exceeded a 32-bit area when the command position value was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
1013	4115	The operation speed value was less than 0, or it exceeded the maximum speed value.	Set the operation speed value to be larger than 0 and less than the maximum speed value set in the axis.
1014	4116	The acceleration was set as the negative number.	Set the acceleration value to a value greater than or equal to 0.
1015	4117	The deceleration was set as the negative number.	Set the deceleration value to a value greater than or equal to 0.
1016	4118	The jerk was set as the negative number.	Set the jerk value to a value greater than or equal to 0.
1017	4119	The direction specification exceeded the range.	Check the range of the direction setting value of the command and set the value within the range. (Refer to Chapter 6 Commands and Functions)
1018	4120	The torque setting value exceeded the range.	Set the torque setting value within 1000%.
1019	4121	The torque ramp setting value exceeded the range.	Set the torque ramp setting value to a value greater than or equal to 0.
101A	4122	Buffer Mode setting value exceeded the input range.	Set value (0 ~ 5) that can be set in the Buffer Mode.
101B	4123	Execution Mode setting value exceeded the input range.	Set value (0 ~ 1) that can be set in the Execution Mode.
101C	4124	Tracking error-over range occurred, and thus operation cannot continue.	Deviation between command position and current position exceeded 'Tracking error-over range value'. To prevent an alarm from occurring, tune the servo drive or set the 'Tracking error-over range value' to a larger value.
101D	4125	Tracking error-over range occurred.	Deviation between command position and current position exceeded 'Tracking error-over range value'. To prevent an alarm from occurring, tune the servo drive or set the 'Tracking error-over range value' to a larger value.
101F	4127	The command position value transmitted to the servo driver was out of the range of pulse unit expression value.	It exceeded a 32-bit area when the command position value was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
1020	4128	It is the undefined axis command.	This command is not performed in the current version of the controller. Please contact customer support team of our company after checking the version in which the command can be executed.
1021	4129	The same command was executed, and thus the previously executed command was canceled.	The motion command can be executed only once per scan. Change the operation condition of the program so that one motion command can be executed in one scan.
1022	4130	It exceeded the number of commands that can execute a Buffered command.	The command cannot be executed because the command buffer of the axis is full. The number of commands that can be executed with the Buffered command is 100. Adjust the timing of the command execution.

Error code		Error Description	Solutions
Hex	Dec		
1023	4131	The value of the parameter exceeded the input range	Check the parameter input range and execute the command.
1030	4144	Axis parameters cannot be written when the axis is in operation.	Perform parameter writing when the axis is not in operation.
1040	4160	Axis parameter data is abnormal.	Download the data again from XG5000 and place requests for A/S if the error occurs repeatedly after re-execution.
1041	4161	It is not possible to execute operation due to axis parameter error.	Check the axis parameter and set it again.
1042	4162	The speed limit of the axis parameter cannot be set to the value less than 0.	Set the speed limit of the basic parameter to a value greater than 0.
1043	4163	The soft upper/lower limit value of axis parameter exceeded the range.	Set the soft upper limit value of the axis parameter to be greater than or equal to the soft lower limit value.
1044	4164	The current speed filter time constant value of axis parameter exceeded the range.	Set the parameter setting value to 0 ~ 100.
1045	4165	The error reset monitoring time of axis parameter exceeded the range.	Set the parameter setting value to 1 ~ 1000.
1046	4166	The setting value of transfer distance per rotation exceeded the range.	Set the parameter setting value to more than 0.000000001 and less than 4294967295.
1047	4167	The setting value of infinite length repeat position exceeded the range.	Set the parameter setting value to more than 0 and less than 2,147,483,647 in pulse unit.
1048	4168	The setting value of in-position width exceeded the range.	Set the parameter setting value to more than 0 and less than 2,147,483,647 in pulse unit.
1049	4169	The setting value of tracking error-over range exceeded the range.	Set the parameter setting value to more than 0 and less than 2,147,483,647 in pulse unit.
104A	4170	The setting value of current position display compensation amount exceeded the range.	Set the parameter setting value to more than 0 and less than 2,147,483,647 in pulse unit.
104B	4171	The setting value of jog high speed exceeded the range.	Set the parameter setting value to more than 0, larger than the jog low speed value and less than the speed limit.
104C	4172	The setting value of jog low speed exceeded the range.	Set the parameter setting value to more than 0, smaller than the jog high speed value and less than the speed limit.
104D	4173	The setting value of jog acceleration exceeded the range.	Set the parameter setting value to more than 0.
104E	4174	The setting value of jog deceleration exceeded the range.	Set the parameter setting value to more than 0.
104F	4175	The setting value of jog jerk exceeded the range.	Set the parameter setting value to more than 0.
1050	4176	Motor-side gear ratio setting value exceeded the range.	Set the parameter setting value to 1 ~ 65535.
1051	4177	Machine-side gear ratio setting value exceeded the range.	Set the parameter setting value to 1 ~ 65535.
1052	4178	The setting value for the number of pulses per rotation exceeded the range.	Set the parameter setting value to be greater than 0 and less than or equal to 4294967295.
1053	4179	Connection device setting value exceeded the range.	Set the device number of the slave that can be supported. The node setting range is 0(no connection device), and 1 ~ 64.
1054	4180	Axis type setting value exceeded the range.	Set the parameter setting value to '0: actual axis' or '1: virtual axis'.
1055	4181	Speed command unit setting value exceeded the range.	Set the parameter setting value from '0: unit/sec', '1: unit/min', '2: rpm'.

Error code		Error Description	Solutions
Hex	Dec		
1056	4182	The backlash compensation amount setting value exceeded the range.	Set the parameter setting value to 0 or more and the pulse unit to 65,535 or less.
1060	4192	Servo On cannot be executed due to occurrence of servo drive errors.	Execute Servo On after checking the error factor of the servo drive and removing the servo drive error.
1061	4193	Servo On command was executed again in the middle of processing Servo On.	Check whether the Servo On command was performed again in the middle of processing Servo On in program or XG5000.
1062	4194	It is not possible to complete Servo On because the servo drive cannot be changed to "ReadyToSwitchON" state.	Check the status of the servo drive. The Servo On command may not be executed in certain circumstances.
1063	4195	It is not possible to complete Servo On because the servo drive cannot be changed to "Switched on" stage.	Check the status of the servo drive. The Servo On command may not be executed in certain circumstances.
1064	4196	It is not possible to complete Servo On because the servo drive cannot be changed to "Operation enabled" state.	Check the status of the servo drive. The Servo On command may not be executed in certain circumstances.
1065	4197	It is not possible to complete Servo On because "Quick Stop" of servo drive is enabled.	Check the status of the servo drive. The Servo On command may not be executed in certain circumstances.
1066	4198	Servo Off command was executed again in the middle of processing Servo Off.	Check whether the Servo Off command was performed again in the middle of processing Servo Off in program or XG5000.
1067	4199	The execution of Servo Off command was not completed.	Check the status of the servo drive.
1070	4208	It exceeded the servo error reset monitoring time.	The servo drive error has not been cleared even after the error reset monitoring time set in the axis parameter went by. Execute the error reset command again after removing the servo drive error factor.
1080	4224	Commands that use absolute coordinates cannot be executed at absolute coordinates of the state of undetermined origin.	Execute the absolute coordinate operation command after making the determined origin state with homing command or current position setting command.
1081	4225	In infinite length repeat enable state, the target position was beyond the range of infinite length repeat position from the direction specification.	Set the target position within the infinite length repeat position from the direction specification.
1082	4226	SuperImposed command cannot be executed during operation with speed control or torque control.	Execute the SuperImposed command when it is not in operation with speed control or torque control.
1083	4227	SuperImposed operation stop command cannot be executed when the SuperImposed operation is not working.	Execute the SuperImposed operation stop command when the SuperImposed operation is in progress.
1090	4240	The position value of the current position change command exceeded the range.	Execute the current position preset command after setting the position setting value to more than the soft lower limit value of the extended parameter and less than the soft upper limit value.
1091	4241	The current position change command cannot be executed in case of operation with homing, speed synchronization, cam and torque control.	Execute the current position change when the axis is not in operation of among one of homing, speed synchronization, cam and torque control.
1092	4242	If the spindle axis is automatically controlled by the NC function module, if the item 'Spindle Encoder Selection' of the axis parameter is '0: Disable', the axis' current position setting	After correctly setting the connection method of the encoder connected to the spindle axis in the item 'Spindle Encoder Selection' of the axis parameter, execute the axis' current position setting command.

Error code		Error Description	Solutions
Hex	**Dec**		
		command cannot be executed.	
1093	4243	If the item 'Spindle Encoder Selection' of the axis parameter is '1: motor ENC', as there is not a 'Position actual value (0x6064)' object in the TxPDO setting of the EtherCAT slave connected to the spindle axis, the axis' current position setting command cannot be executed.	If the item 'Spindle Encoder Selection' of the axis parameter is '1: motor ENC', after reconnecting the EtherCAT by adding the 'Position actual value (0x6064)' object to the TxPDO setting of the EtherCAT slave connected to the spindle axis, execute the axis' current position setting command.
1094	4244	If the item 'Spindle Encoder Selection' of the axis parameter is '2: Built-in ENC1', as the encoder1 parameter setting is wrong, the axis' current position setting command cannot be executed.	If the item 'Spindle Encoder Selection' of the axis parameter is '2: Built-in ENC1', after setting the encoder1 unit of the encoder parameter = pulse, the maximum value of the encoder1 = 2147483647 and the minimum value of the encoder1 = -2147483648, execute the axis' current position setting command.
1095	4245	If the item 'Spindle Encoder Selection' of the axis parameter is '3: Built-in ENC2', as the encoder2 parameter setting is wrong, the axis' current position setting command cannot be executed.	If the item 'Spindle Encoder Selection' of the axis parameter is '3: Built-in ENC2', after setting the encoder2 unit of the encoder parameter = pulse, the maximum value of the encoder2 = 2147483647 and the minimum value of the encoder2 = -2147483648, execute the axis' current position setting command.
10A0	4256	Servo drive does not support torque control mode.	Perform torque control by using serve drive that supports CST mode of EtherCAT CoE.
10A1	4257	There is no target torque object (0x6071) setting that can execute the torque control on RxPDO entry setting in slave data of the EtherCAT parameter.	Set the target torque object (0x6071) that supports torque control to the RxPDO entry of the EtherCAT parameter slave data in XG5000, and then send it to the controller.
10B0	4272	Servo drive does not support homing mode.	Perform homing by using servo drive that supports homing mode of EtherCAT CoE.
10B1	4273	An error occurred during the execution of the homing in the servo drive.	Check the error factor of the servo drive and perform homing after removing the servo drive error.
10B2	4274	The homing command cannot be executed when axis is running.	Perform the homing command again in the standstill state after the axis stops.
10B3	4275	The switch search speed value of the extended homing command exceeded the range.	When converting the switch search speed value to pulse unit, set it to a value between 0 and 1073741824.
10B4	4276	The zero search speed value of the extended homing command exceeded the range.	When converting the zero search speed value into pulse units, set it to a value between 0 and 1073741824.
10B5	4277	Homing acceleration value of extended homing command exceeded the range.	When converting the homing acceleration value into pulse units, set it to a value between 0 and 1073741824.
10B6	4278	The home offset value of the extended homing command exceeded the range.	When converting the home offset value into pulse units, set it to a value between -2147483648 and 2147483647.
10B7	4279	Failed to write extended homing command parameters to the drive.	Among the parameters of the extended homing command, there is an item that is not supported by the drive. (Refer to the user manual of the drive) Use the basic homing command (MC_Home).
10C0	4288	The override command cannot be executed when the position /speed control is not in operation.	Execute override command during operation with position control or speed control.
10C1	4289	The override factor of the override command	Set the VelFactor, AccFactor and JerkFactor values of the

Error code		Error Description	Solutions
Hex	**Dec**		
		exceeded the range.	override command to be 0 or more and then execute the override command.
10C2	4290	The operation speed value exceeded the maximum speed value after reflecting the factor of the override command.	Perform override within the range that does not exceed the maximum speed value of the axis.
10D0	4304	The gear ratio denominator value cannot be zero.	Execute the command after setting the gear ratio denominator to a value other than 0.
10D1	4305	Gear operation MasterValueSource setting value exceeded the range.	Execute the command after setting the MasterValueSource input value to a value between 0 and 1.
10D2	4306	Gear operation main axis setting was out of the range.	Set axis and encoder numbers with a range by product.
10D3	4307	The gear operation main axis setting is identical with the subordinate axis.	Execute the command after setting the main axis to the different axis from the subordinate axis(command axis).
10D4	4308	The gear operation main axis was not ready.	Execute the command when the main axis is ready.
10D5	4309	In case the gear operation main axis is set as an encoder, the gear operation command cannot be executed if the encoder parameter setting error of the built-in parameter occurs.	Confirm the encoder-related items of the encoder parameters, check if they are set to values within the range and set the encoder parameters to normal values by using XG5000.
10D6	4310	The MC_GearInPos command cannot be executed when the main axis is running with torque control.	Execute the MC_GearInPos command while the main axis is not operating in torque control.
10D7	4311	The speed of the gear operation subordinate axis exceeded the speed limit.	Reduce the speed of the main axis or change the gear ratio lest the speed of the subordinate axis in gear operation should exceed the speed limit set on the subordinate axis.
10D8	4312	The gear release command cannot be executed if it is not gear operation.	The gear release command can be used only when the gear is in operation.
10D9	4313	The command cannot be executed because the target speed setting value of MC_GearInPos command is smaller than the current operation speed or the gear operation speed.	Execute the command after setting the target speed setting value of MC_GearInPos command to the current operation speed or gear operation speed or more.
10DA	4314	It will not be able to reach the subordinate axis synchronization position within the time when the main axis operates to the main axis synchronization position during the MC_GearInPos operation.	Execute the command after increasing the target speed setting value MC_GearInPos command or adjusting MasterStartDistance so that the subordinate axis moves to the subordinate axis synchronization position within the time when the main axis operates to the main axis synchronization position.
10DB	4315	The synchronous operation command (gear, cam, etc.) cannot be executed if the main axis is in homing operation.	Execute the synchronous operation (gear, cam, etc.) when the main axis is not in homing operation.
10E0	4320	There are no object settings in the slave data of the EtherCAT parameter than enable the touch probe to PDO entry settings.	Set the object that supports touch probe to the PDO entry of the EtherCAT parameter slave data in XG5000 and then send it to the controller.
10E1	4321	TriggerInput input setting value was out of the range.	Set the TriggerInput setting value to 0(Touch Probe1) or 1(Touch Probe2).
10E2	4322	TriggerInput input setting value of extended touch probe command is out of range.	Set the TriggerInput setting value of the extended touch probe command to a value between 0 (rising edge of TouchProbe1) and 5 (index pulse of TouchProbe2).
10E3	4323	TriggerMode input setting value of extended	Set the TriggerMode setting value of the extended touch probe

Error code		Error Description	Solutions
Hex	**Dec**		
		touch probe command is out of range.	command to 0 (single trigger) or 1 (continuous trigger).
10F0	4336	The parameter number setting value of the parameter read/write command was out of the range.	Execute the command after setting the parameter number setting value of parameter read/write command to be between 0~28, 100~106 and 200~206.
10F1	4337	The data setting value of the set parameter of the parameter write command was out of the range.	Check the data setting range of the parameter to be set.
10F2	4338	The parameter cannot be changed because the maximum value of encoder 1 is out of the pulse unit expression value when the encoder parameter is changed.	Change the maximum value of encoder 1 in advance to prevent errors when converted in pulse unit, and then change the parameter.
10F3	4339	The parameter cannot be changed because the minimum value of encoder 1 is out of the pulse unit expression value when the encoder parameter is changed.	Change the minimum value of encoder 1 in advance to prevent errors when converted in pulse unit, and then change the parameter.
10F4	4340	The parameter cannot be changed because the maximum value of encoder 2 is out of the pulse unit expression value when the encoder parameter is changed.	Change the maximum value of encoder 2 in advance to prevent errors when converted in pulse unit, and then change the parameter.
10F5	4341	The parameter cannot be changed because the minimum value of encoder 2 is out of the pulse unit expression value when the encoder parameter is changed.	Change the minimum value of encoder 2 in advance to prevent errors when converted in pulse unit, and then change the parameter.
1100	4352	The jog operation command cannot be executed when the axis is running.	Execute the jog command when the axis is in stop state.
1101	4353	If the 'Tool Retract' command is executed on the NC channel, the jog operation command cannot be simultaneously executed on more than 2 axes.	If aborting the 'Tool Retract' command or using the 'Tool Retract' command of the NC channel, execute one jog operation command on one axis.
1110	4368	There is an error in the cam operation MasterScaling input value.	You cannot put 0 in the MasterScaling input value.
1111	4369	There is an error in the cam operation MasterStartDistance input value.	Set the MasterStartDistance input value to a value greater than 0 and execute the command.
1112	4370	There is an error in the cam operation MasterSyncPosition input value.	Set the MasterSyncPosition input value to a value greater than 0 and execute the command.
1113	4371	The cam operation StartMode input value exceeded the range.	Set the StartMode input value to a value between 0 and 1 and execute the command.
1114	4372	The cam operation MasterValueSource input value exceeded the range.	Set the MasterValueSource input value to a value between 0 and 1 and execute the command.
1115	4373	The specified cam table does not exist.	Adjust the cam table number to a valid cam table number and execute the command.
1116	4374	The cam operation main axis setting was out of the range.	Execute the command after setting the main axis in the area of 1~36 (axis) and 1001~1002(encoder). Check whether the VarOffset value deviates from the memory area if the main axis is a variable.
1117	4375	The cam operation main axis setting is identical with the subordinate axis.	Execute the command after setting the main axis to the different axis from the subordinate axis(command axis).
1118	4376	The cam operation main axis was not ready.	Execute the command when the main axis is ready.

Error code		Error Description	Solutions
Hex	Dec		
1119	4377	In case the cam operation main axis is set as an encoder, the command cannot be executed if the encoder parameter error of the built-in parameter occurs.	Confirm the encoder-related items of the encoder parameter check if they are set to values within the range and set the encoder parameter to normal values by using XG5000.
111A	4378	The speed of the cam operation main axis exceeded the speed limit.	Reduce the speed of the main axis or adjust the cam table lest the speed of the subordinate axis in cam operation should exceed the speed limit set on the subordinate axis.
111B	4379	The cam release command cannot be executed if it is not cam operation.	The cam release command can be used only when the cam is in operation.
111C	4380	The value for setting the number of cam data of the cam data write command exceeded the range.	Set the value for setting the number of cam data of the cam data write command to less than the number of registered cam points. Add a cam table if the cam table is not registered.
111D	4381	The specified cam table data of the cam data read command is abnormal.	Reset the cam data and place requests for A/S if it occurs again after re-execution.
111E	4382	The cam skip command cannot be executed if the cam is not in operation.	Execute the cam skip command when the cam is running.
111F	4383	The number of cam cycles to be skipped by the cam skip command was set to 0.	Set the number of cam cycles to be skipped by the cam skip command to be greater than 0.
1121	4385	The skip mode setting value of the cam skip command exceeded the range.	Execute the command after setting the skip mode setting value of the cam skip command to a value between 0 and 2.
1122	4386	The cam table is not registered.	Register the cam table or set the data again to execute the command.
1123	4387	The cam data of the cam data write command is abnormal.	Set the data of the cam data write command correctly.
1124	4388	Cam main axis value does not exist within the specified range.	Check the MasterStartPos and MasterEndPos values and run the command again.
1130	4400	The phase correction command cannot be executed if the command axis is not InSync or InGear status of synchronous control(cam, gear operation)operation.	Execute the phase correction command when the command axis of the phase correction command is in synchronous control operation and InSync or InGear state.
1131	4401	There is an error in setting the main axis of the phase correction command.	Execute the command after setting the main axis of the phase correction command to the same as the actual axis of the current synchronous operation.
1132	4402	The phase correction amount of the phase correction command was out of the position expression range.	Execute the command after setting the phase correction amount so that the phase correction amount is within the range from more than -2,147,483,648 to less than 2,147,483,647 in pulse unit.
1133	4403	The speed setting value of the phase correction command was out of the range.	Execute the command after setting the speed value of the phase correction command to be larger than 0 and less than the speed limit of the main axis.
1140	4416	The connected slave device does not support the speed control mode.	Perform the speed control by using the slave device that supports the velocity mode of the EtherCAT CoE.
1150	4432	The connected slave device does not support the position control mode.	Perform the position control by using the slave device that supports the CSP mode of the EtherCAT CoE.
1151	4433	The operation mode (0x6060, Mode Of Operation) SDO read service of the connected slave device has failed.	Check if the slave is in a state that can perform SDO service. If the error repeatedly occurs by re-executing the command, use the Disconnect EtherCAT – Connect command to reconnect

Error code		Error Description	Solutions
Hex	Dec		
			EtherCAT before using it.
1152	4434	Operation mode of the connected slave device (0x6060, Mode Of Operation) A timeout occurred for the SDO read request.	Check if the slave is in a state that can perform SDO service. If the error repeatedly occurs by re-executing the command, use the Disconnect EtherCAT - Connect command to reconnect EtherCAT before use.
1160	4448	The connected slave device does not support the Cyclic Synchronous Velocity (CSV) mode.	Control the speed by using a slave device that supports the EtherCAT CoE CSV mode.
1161	4449	There is no target speed object (0x60FF) setting that can execute the Cyclic Synchronous Velocity (CSV) on the RxPDO entry setting in the slave parameter.	Set the target speed object (0x60FF) that supports the Cyclic Synchronous Velocity (CSV) on the RxPDO entry setting of the EtherCAT parameter slave data in XG5000, and then send it to the controller.
1162	4450	The CmdPosMode setting value of the Cyclic Synchronous Velocity (CSV) operation exceeded the input range.	The CmdPosMode supports the value of 0 (the current position is applied to the command position). After setting the CmdPosMode to 0, execute the command again.
1200	4608	The hardware upper limit error occurred.	Remove the error by executing the error reset command after breaking away from the external upper limit signal range with the use of reverse jog command.
1201	4609	The hardware lower limit error occurred.	Remove the error by executing the error reset command after breaking away from the external lower limit signal range with the use of forward jog command.
1203	4611	The command cannot be executed due to servo drive error during operation.	Remove the servo error with the error reset command after eliminating the servo error factor.
1204	4612	The command cannot be executed due to servo-off during operation.	Re-execute the command after changing the command axis to servo-on state with the servo-on command.
1205	4613	The software upper limit error occurred.	Remove the error by executing the error reset command after breaking away from the software upper limit range with the use of reverse jog command.
1206	4614	The software lower limit error occurred.	Remove the error by executing the error reset command after breaking away from the software lower limit range with the use of forward jog command.
1210	4624	If the spindle axis is automatically controlled by the NC function module, motion commands related to transfer cannot be executed.	After confirming the motion commands that can be executed on the axis allocated to the NC spindle axis, execute the motion commands allowed to the NC spindle axis.
1220	4640	The parameter number setting value of the motion information read command is out of range.	Execute the command after setting the parameter number setting value of the motion information read command between 0 and 5.
1230	4656	Master position loop control cannot be executed/disabled during operation.	Execute/release master position loop control when the corresponding axis is not in operation.
1231	4657	The P gain input value of the master position loop control function block is negative.	Enter a value of 0 or more in the P gain of the master position loop control function block.
1232	4658	The I gain input value of master position loop control function block is negative.	Enter a value of 0 or higher in the I gain of the master position loop control function block.
1233	4659	The control output limit value of the master position loop control function block is negative.	Enter a value of 0 or more in the control output limit of the master position loop control function block.
1234	4660	Cross-coupled control cannot be executed/disabled during operation.	Execute/release cross-coupled control when the corresponding axis is not in operation.
1235	4661	The command axis and connection axis of the	Set different axes for the command axis and the connection axis

Error code		Error Description	Solutions
Hex	Dec		
		cross-coupled control function block must be different axes.	of the cross-coupled control function block..
1236	4662	To drive cross-coupled control, the master position loop controller must be operating.	In order to drive cross-coupled control, execute master position lure control for the relevant axis first.
1237	4663	The P gain input value of the cross-coupled control function block is negative.	Enter a value of 0 or higher for the P gain of the cross-coupled control function block.
1238	4664	The command was re-executed on an axis for which cross-coupled control is not running.	When re-executing the cross-coupled control function block, set the command axis and connection axis to the previously executed values. The value of P gain can be changed without re-executing the function block.
1F00	7936	The periodic communication error occurred. (The communication error exceeding the master parameter periodic communication timeout count occurred)	Check whether servo power is off during communication, communication cable is normally installed and communication cable is exposed to noise.
1F10	7952	SDO commands can no longer be executed due to the SDO processing failure of slave device that was performed previously	Reset the connection after checking whether the status of the slave device is normal.
1F11	7953	The SDO parameter write command cannot be executed during operation.	Execute the SDO parameter write command when the axis is not in operation.
1F12	7954	The range of data such as SDO parameter Index, subIndex and etc. was out of the allowed value.	Execute the SDO parameter write command after setting the SDO parameter Index to 0x0000~0x9FFF, SubIndex to 0x00~0xFF and data size within 4 bytes.
1F13	7955	Abort occurred during SDO parameter write command.	Stop of writing operation was done in the middle of writing SDO parameter in slave device. Check the writting data and the status of slave device.
1F14	7956	There is no response of the slave device regarding the command to write SDO parameter.	There is no response from slave device in the middle of writing SDO parameter. Check the status of slave device.
1F16	7958	Abort occurred while saving SDO parameter EEPROM.	Cancelation was done in the middle of saving SDO parameter EEPROM in slave device. Check the status of slave device.
1F17	7959	There is no response of the slave device regarding the parameter to save SDO parameter EEPROM.	There is no response from slave device in the middle of saving SDO parameter EEPROM. Check the status of slave device.
1F19	7961	Other commands cannot be executed while writing SDO parameter or saving SDO parameter EEPROM.	Execute other commands after saving SDO parameter EEPROM.
1F20	7968	Abort occurred in the middle of the command to write SDO parameter.	Stop of writing operation was done in the middle of writing SDO parameter in slave device. Check the writting data and the status of slave device.
1F21	7969	There is no response of the slave device regarding the command to read SDO parameter.	There is no response from slave device in the middle of reading SDO parameter EEPROM. Check the status of slave device.
1F22	7970	The SDO parameter read/write commands cannot be executed while the SDO parameter read/write commands are being executed.	Execute the command after the currently executed SDO parameter read/write is completed.
1F33	7987	It failed to change the operation mode of the servo drive to the position control(CSP) mode	Confirm whether the servo drive supports EtherCAT CoE CSP mode and check the status of servo drive.
1F34	7988	It failed to change the operation mode of the servo drive to the homing mode	Confirm whether the servo drive supports EtherCAT CoE Homing mode and check the status of servo drive.

Error code		Error Description	Solutions
Hex	Dec		
1F35	7989	It failed to change the operation mode of the servo drive to the torque control (CST) mode.	Confirm whether the servo drive supports EtherCAT CoE CST mode and check the status of servo drive.
1F36	7990	It failed to change the operation mode of the servo drive to the speed control (CSV) mode.	Confirm whether the servo drive supports the EtherCAT CoE CSV mode and check the status of the servo drive.
1F50	8016	The XG5000 manual tuning function cannot be executed in case the controller is in the RUN state.	Perform manual tuning of XG5000 after changing the controller to STOP state.
1F60	8032	It is the command that is not available in the operation mode of the current slave.	Execute the command after setting the slave to the Boot state.
1F61	8033	Transmission timeout occurred during file transfer.	Check the status of the transmission line or slave and execute the command.
1F62	8034	Receive timeout occurred during file transfer.	Check the status of the transmission line or slave and execute the command.
1F63	8035	Packet error occurred during file transfer.	Check the status of the transmission line or slave and execute the command.
1F64	8036	There is a memory shortage in slave.	Check the transferred file and execute the command.
1F65	8037	The device does not exist.	Check whether the FOE function is available slave and execute the command.
1F66	8038	Access to the slave is defied.	Check whether the FOE function is available slave and execute the command.
1F67	8039	The password does not match.	Check the password and execute the command.
1F68	8040	Data to be downloaded by the FoE function was not transferred to the controller.	Check the communication cable connection status and controller operation status.
1F6F	8047	There was a slave error during file transfer.	Remove the slave error and execute the command.
1170	4464	PartLength value is invalid.	The PartLength cannot be smaller than 0.
1171	4465	Circumference value is invalid.	The Circumference cannot be smaller than 0.
1172	4466	The value for the cutting start position is invalid.	The cutting start position cannot be smaller than 1/4 of the Circumference.
1173	4467	The value for the cutting end position is invalid.	The cutting end position cannot be bigger than 1/4 of the Circumference or bigger than the cutting start position.
1174	4468	The value for the synchronization speed ratio is invalid.	The synchronization speed ratio value must be between 50-200.
1175	4469	The ratio for the 0 speed region is invalid.	The ratio for the 0 speed region must be between 0-50.
1176	4470	The value for the cam profile type is invalid.	Change the cam profile type and run the command.
1177	4471	The value for the cam point number is invalid.	Change the cam point number and run the command.
1178	4472	The value for the cam curve type is invalid.	Change the cam curve type and run the command.
1179	4473	The cutting region is too wide.	Change the length of the cutting region or the speed ratio and run the command.
2000	8192	The axis group was not ready for operation.	Execute the command when the axis group is ready for operation.
2001	8193	The axis group cannot be executed in "Disabled" state.	Check the operable axis status of the command and execute the command when the command can be run.
2002	8194	The axis group cannot be executed in "Standby" state.	Check the operable axis status of the command and execute the command when the command can be run.
2003	8195	The axis group cannot be executed in "Moving" state.	Check the operable axis status of the command and execute the command when the command can be run.
2004	8196	The axis group cannot be executed in "Homing"	Check the operable axis status of the command and execute the

Error code		Error Description	Solutions
Hex	Dec		
		state.	command when the command can be run.
2005	8197	The axis group cannot be executed in "Stopping" state.	Check the operable axis status of the command and execute the command when the command can be run.
2006	8198	The axis group cannot be executed in "Errorstop" state.	Check the operable axis status of the command and execute the command when the command can be run.
2007	8199	The axis configuration of the axis group is not servo-on state.	Check the operable axis status of the command and execute the command when the command can be run.
200F	8207	The axis group operation cannot continue because the controller is stopped by the ESTOP command.	Check whether the controller was stopped by the ESTOP command during the axis group operation.
2010	8208	The controller was changed to 'Stop' or 'Error' state, and thus operation cannot continue.	Check whether the controller was changed to 'Stop' or 'Error' state during operation of the axis group.
2011	8209	The EtherCAT network connection was lost, and thus operation cannot continue.	Check whether the EtherCAT network connection has been disconnected due to slave power supply error, network cable error and noise inflow on network cable during operation of the axis group.
2012	8210	The position setting value of the command was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
2013	8211	The operation speed value was less than 0, or it exceeded the maximum speed value.	Set the operation speed value to a value that is greater than 0 and less than or equal to the maximum speed value set in the axis group.
2014	8212	The acceleration was set as the negative number.	Set the acceleration value to a value greater than or equal to 0.
2015	8213	The deceleration was set as the negative number.	Set the deceleration value to a value greater than or equal to 0.
2016	8214	The jerk was set as the negative number.	Set the jerk value to a value greater than or equal to 0.
201A	8218	Buffer Mode setting value exceeded the range.	Set the value that can be set in Buffer Mode.
201B	8219	Execution Mode setting value exceeded the input range.	Set the value that can be set (0 ~ 1) in Execution Mode.
201C	8220	Transition Mode setting value exceeded the range.	Set the value that can be set in Transition Mode in the command.
201D	8221	Transition Parameter setting value exceeded the range.	Set the value that can be set in Transition Parameter in the command.
201E	8222	The axis group operation was stopped due to the error occurrence of axis group configuration axis.	Execute the command after eliminating the error factor and removing the error with the axis or axis group reset command.
201F	8223	The command position value transmitted to the servo drive was out of the pulse unit expression value.	It exceeded the 31-bit area when the command position value was converted in pulse unit. Set the value in the range of -2,147,483,648 ~ 2,147,483,647 when converted in pulse unit. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
2020	8224	It is undefined axis group command.	This command is not performed in the current version of the controller. Please contact customer support team of our company after checking the version in which the command can be executed.

Error code		Error Description	Solutions
Hex	Dec		
2021	8225	The previously executed command was canceled because the same command was executed.	Check whether the command was executed again during execution of the same command.
2022	8226	It exceeded the number of commands that can execute Buffered command.	The command cannot be executed because the command buffer of the axis is full. The number of commands that can be executed with the Buffered command is 100. Adjust the timing of the command execution.
2030	8240	Axis group parameters cannot be written if the axis group is in operation.	Perform the axis group parameter writing when the axis group is not in operation.
2040	8256	Axis group parameter data is abnormal.	Download the data again from XG5000 and place requests for A/S if the error occurs repeatedly after re-execution.
2041	8257	Operation cannot be executed due to error of the axis group parameter.	Check the axis group parameter and set it again.
2042	8258	The speed limit of the axis group parameter cannot be set to a value less than or equal to 0.	Set the speed limit to a value greater than 0.
2043	8259	The configuration axis number setting value of the axis group parameter exceeded the range.	Set the configuration axis (axis and encoder number) of the axes group to the range by product.
2051	8273	The axis which you are going to add is already registered in the axis group.	Check whether the same axis number exists in the axis group and then set another axis.
2052	8274	The current axis group is active, and the axis you want to add is already included in the other activated axis group.	Execute the command after changing the enabled axis group that includes the axis to GroupDisabled state.
2053	8275	The IdentInGroup setting value of axis group add/remove command exceeded the range.	Set the IdentInGroup setting value in the range of 1 to 10.
2060	8288	There is no axis setting in the specified axis group of the axis group enable/disable command.	Set one or more axes in the axis group and execute the command.
2061	8289	The axis group cannot be enabled because there is an operating axis among the configuration axes of the current axis group.	Execute the command while all axes belonging to the axis group are not in operation.
2062	8290	The specified axis group cannot be enabled because the configuration axis of the current axis group is the configuration axis of another enabled axis group.	Check whether the axis belonging to the axis group is included in another enabled axis group.
2063	8291	The axis group operation cannot be executed because the configuration axes of the axis group have different units.	Set the unit of the configuration axis belonging to the axis group to the same value in order to execute the operation.
2064	8292	The axis group cannot be enabled due to the parameter error of the axis group configuration axis.	Set the parameter of the configuration axis belonging to the axis group within the normal range.
2065	8293	The axis group cannot be enabled because the speed command units of the axis group configuration axes are different from each other.	Set the same speed command unit for the configuration axes belonging to the axis group.
2066	8294	The axis group cannot be enabled because there is an axis whose speed command unit is rpm among the axis group configuration axes.	The speed command unit of the configuration axes belonging to the axis group cannot be set to rpm. Set it to a value other than rpm.
2067	8295	The coordinate system operation cannot be	Set the unit of the configuration axis belonging to the axis group

Error code		Error Description	Solutions
Hex	Dec		
		executed because the unit of the axis group configuration axes is different from the coordinate type.	to match the coordinate system type in order to execute the command.
206F	8303	The axis group cannot be activated if the axis group configuration axis is in NC control operation.	Execute the command when configuration axis belonging to the axis group is not in NC control operation.
2070	8304	The servo drive of the configuration axis does not support homing mode.	Confirm whether the servo drive supports EtherCAT CoE Homing mode and check the status of servo drive.
2071	8305	There is an axis where homing is not completed normally among configuration axes.	Execute the command again after eliminating the error factor by checking the configuration axis error code.
2072	8306	The axis group homing command cannot be executed when the axis group is in operation.	Execute the axis group homing command again in GroupStandby state after axis group operation stops.
2080	8320	There is an axis that has an error during the setting of the current position among the configuration axes.	Execute the command again after eliminating the error factor by checking the configuration axis error code.
2090	8336	The absolute coordinate linear interpolation command cannot be executed if the configuration axis is in the undetermined origin state.	Execute the command after making origin determination state with the homing command or the current position setting command.
2091	8337	It exceeded the speed limit of the linear interpolation configuration axis.	Execute the command at a lower command speed so as not to exceed the speed limit of the configuration axis.
2092	8338	In the case of a specified corner distance transition, transition operation cannot be executed because the corner distance specification value is larger than the moving distance to the target position.	Set the corner distance value specified in the transition parameter to be smaller than the moving distance to the target position.
2093	8339	In the case of a specified corner distance transition, transition operation cannot be executed because the radius of an arc to be inserted exceeds 2,147,483,647 pulse.	Execute the linear interpolation by resetting the target position or changing the transition mode so that the two straight lines cannot be located on a straight line.
2094	8340	Linear interpolation operation cannot be executed when the main axis or subordinate axis is in infinite length repeat "allowable" state	Execute the command after changing the infinite length repeat setting of the main axis or subordinate axis to "0: Disable".
20A0	8352	Absolute coordinate circular interpolation command cannot be executed when the configuration axis is in undetermined origin state.	Execute the command after making the determined origin state with homing command or current position setting command.
20A1	8353	Circular interpolation mode setting value exceeded the range.	Set the circular interpolation mode to a value between 0 and 2 (0: auxiliary point, 1: center point, 2: radius).
20A2	8354	Circular interpolation pass selection setting value exceeded the range.	Set the circular interpolation pass selection to a value between 0 and 1 (0: CW, 1: CCW).
20A3	8355	The radius setting exceeded the range in the circular interpolation radius method.	Set the radius setting value of the circular interpolation main axis operation data to be more than 80% of the half of the length from the start point to the end point.
20A4	8356	In circular interpolation, operation cannot be executed if start point = center point(midpoint) or center point(midpoint) = end point.	Execute the circular interpolation after setting the center point(or midpoint) to a different position from the start point(or end point).
20A5	8357	In circular interpolation, the start point and the	If you set the circular interpolation to midpoint (or radius), set the

Error code		Error Description	Solutions
Hex	Dec		
		end point cannot be the same in midpoint(or radius) method.	position of the start point differently from that of the end point and then execute the circular interpolation.
20A6	8358	It is radius setting error in circular interpolation.	The radius of a circle where the circular interpolation operation can be executed is greater than 0 and less than 2,147,483,647 pulse. Execute the command after setting the input value so that the radius can be calculated within the setting range. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
20A7	8359	The operation cannot be executed because linear profile appears in circular interpolation.	Execute the circular interpolation changing the midpoint so that it cannot be located on the straight line of the start point and the end point in the case of a circular interpolation midpoint method.
20A8	8360	Linear interpolation operation cannot be executed when the main axis or subordinate axis is in infinite length repeat "allowable" state.	Execute the command after changing the infinite repeat setting of the main axis or subordinate axis to "0: Disable".
20A9	8361	Circular interpolation cannot be executed if there are more than four axes constituting the axis group.	Set the axis group to 2-axis for circular interpolation and 3-axis for helical interpolation.
20AA	8362	Circular interpolation cannot be executed if the axis configuration of the axis group is not configured in regular sequence.	For circular interpolation, set the configuration axis for the axis group in regular sequence.
20AB	8363	It exceeded the speed limit of the circular interpolation configuration axis.	Execute the command with the command speed lowered so as not to exceed the speed limit of the configuration axis.
20AC	8364	In circular interpolation, the midpoint(center point) must be in the same XY plane as the start point in the midpoint(or radius) method.	Execute the circular interpolation after setting the center point (or midpoint) to the position in the same the XY plane as the start point (end point) in circular interpolation.
20C0	8384	The coordinate system operation command cannot be executed when the constituent axis is in the undetermined origin state.	Execute the command after making the determined origin state with the homing command or the current position setting command.
20C1	8385	The PCS setting parameter data of the coordinate system parameter is abnormal.	Check the PCS setting parameter and set it again.
20C2	8386	Coordinate system parameter, coordinate system-type parameter data is abnormal.	Check the coordinate system-type parameter and set it again.
20C3	8387	Coordinate system parameter, coordinate system-type parameter data is abnormal.	Check the instrument parameter and set it again.
20C4	8388	Coordinate system parameter, workspace-type data is abnormal.	Check the workspace-type parameter and set it again.
20C5	8389	Coordinate system parameter, workspace parameter data is abnormal.	Check the workspace parameter and set it again.
20C6	8390	It is the position where you cannot start the coordinate system operation.	Move to the position where you can start the coordinate system operation and execute the command.
20C7	8391	It is the target position that cannot be reached by the coordinate system operation.	Check whether there is abnormality in the target position or coordinate system parameter and set it again.
20C8	8392	It is the operation out of the workspace.	Check whether there is abnormality in the workspace parameter or target position and set it again.
20C9	8393	The axis group cannot be activated because the unit of the axis group configuration axis is different from the coordinate system type.	Make sure that the unit of the configuration axis belonging to the axis group matches the coordinate system time.

Error code		Error Description	Solutions
Hex	Dec		
20CA	8394	The coordinate system operation exceeded the maximum interpolation speed.	Execute the command by lowering the command speed so as not to exceed the maximum interpolation speed.
20CB	8395	The coordinate system operation cannot be executed when the configuration axis is in infinite length repeat "allowable" state.	Execute the command after changing the infinite length repeat setting of the configuration axis to "0: Disable".
20CC	8396	It is the CoordSystem that is not supported.	Execute the command after setting the supported CoordSystem.
20CD	8397	It is the TrajType that is not supported.	Execute the command after setting the supported TrajType.
20D0	8400	The conveyor axis setting value exceeded the range.	Set the conveyor axis (axis and encoder number) to the range by product.
20D1	8401	The axis set as the conveyor axis was set as the axis group configuration axis.	Execute the command when the conveyor axis is set to another axis.
20D2	8402	There is an error in setting the conveyor axis unit.	Set the unit of the conveyor axis to mm/inch.
20D3	8403	The conveyor axis was not ready.	Execute the command when the conveyor axis is ready for operation.
20D4	8404	The conveyor synchronization command cannot be executed if the main axis is in homing operation.	Execute the command when the conveyor axis is not in homing operation.
20D5	8405	The conveyor synchronization command cannot be executed if the main axis is operating with torque control.	Execute the command when the conveyor axis is not in operation with torque control.
20D6	8406	Conveyor synchronization function cannot be executed if the conveyor axis is in infinite length repeat "Disable" state.	Execute the command after changing the infinite length repeat setting of the conveyor axis to "1: Permit".
20E0	8416	The SETP value of the coordinate system path operation exceeded the range.	Set the STEP value of the coordinate system path operation to a value between 0 and 99 and execute the command.
20E1	8417	The CommandType value of the coordinate system path operation exceeded the range.	Set the CommandType value of the coordinate system path operation to a value between 0 and 4 and execute the command.
20E2	8418	The Mode value of the coordinate system path operation exceeded the range.	Set the Mode value of the coordinate system path operation to a value between 0 and 2 and execute the command.
20E3	8419	It exceeded the number of paths that can be performed by the coordinate system path operation.	Set the STEP value of the coordinate system path operation to a value between 0 and 99 and execute the command.
2110	8464	Axis group override command cannot be executed unless axis group interpolation operation or jog operation is in progress.	Execute axis group override command during axis group interpolation operation or jog operation.
2111	8465	The override factor of the axis group override command exceeded the range.	Execute the command after setting VelFactor, AccFactor, and JerkFactor values of the axis group override command to 0 or higher.
2112	8466	After reflecting the factor of the axis group override command, the axis group operation speed value exceeded the maximum speed value.	Execute override within the range that does not exceed the maximum speed value of the relevant axis group.
2113	8467	After reflecting the factor of the axis group override command, the operating speed value of the axis belonging to the axis group exceeded the maximum speed value.	Execute the override within the range that does not exceed the maximum speed value of the axis included in the relevant axis group.

Error code		Error Description	Solutions
Hex	**Dec**		
2120	8480	Parameter number set value of read/write command of axis group parameter is out of range	Execute the command after setting the parameter number setting value of the axis group parameter read and write command to a value between 1 and 41.
2121	8481	The maximum interpolation speed setting value of the axis group parameter write command is out of range.	Please set the maximum speed interpolation value of the axis group as a positive number.
2122	8482	Coordinate system parameter setting value of axis group parameter write command is out of range.	Among the axis group parameters, check the setting value of the parameter in the 'Coordinate system setting' item.
2123	8483	Workspace setting value of axis group parameter write command is out of range.	Among the axis group parameters, check the setting values of the parameters in the 'workspace setting' item.
2124	8484	PCS setting value of axis group parameter write command is out of range.	Among the axis group parameters, check the setting value of the parameter in the 'PCS setting' item.
2125	8485	The jog speed setting value of the axis group parameter write command is out of range.	Please set the jog speed setting value as a positive number and less than the maximum interpolation speed setting value.
2126	8486	Configuration axis setting value of axis group parameter write command is out of range.	Execute the command after setting the axis setting value of the axis group parameter to a value between 1 and 36.
2130	8496	Among the configuration axes, there is an axis that has not completed servo on/off normally.	Check the error code of the component axis, remove the cause of the error of the axis, and then execute the command again.
3000	12288	The NC channel was not ready for operation.	Check whether the NC channel is ready for operation. In order to use the NC channel, the NC channel should be registered in NC parameter in XG5000.
3001	12289	NC program data is abnormal.	Download the data again from XG5000 and place requests for A/S if it occurs repeatedly after re-execution.
3002	12290	Program cannot be written when NC channel is running automatically.	Execute the program writing when the automatic operation stops if the NC channel is in automatic operation.
3003	12291	NC program writing was not completed normally. (File processing (DELETE) failure in NC program writing)	Download the data again from XG5000 and place requests for A/S if it occurs repeatedly after re-execution.
3004	12292	NC program writing was not completed normally. (File processing(OPEN) failure in writing NC program)	Download the data again from XG5000 and place requests for A/S if it occurs repeatedly after re-execution.
3005	12293	NC program writing was not completed normally. (File processing(WRITE) failure in writing NC program)	Download the data again from XG5000 and place requests for A/S if it occurs repeatedly after re-execution.
3006	12294	NC program writing was not completed normally. (File processing(CLOSE) failure in writing NC program)	Download the data again from XG5000 and place requests for A/S if it occurs repeatedly after re-execution.
3007	12295	Parameters cannot be written when NC channel is running automatically.	Execute the program writing when the automatic operation stops if the NC channel is in automatic operation.
3008	12296	The automatic operation cannot continue because the controller's mode is changed to STOP or ERROR state during the NC channel automatic operation.	Check whether the controller's mode is changed to STOP or ERROR state during the NC channel automatic operation.
3009	12297	The automatic operation cannot continue because EtherCAT network connection is disconnected during the NC channel automatic	Check whether the EtherCAT network connection has been disconnected due to slave power supply error, network cable error and noise inflow on network cable during the NC channel

Error code		Error Description	Solutions
Hex	**Dec**		
		operation.	automatic operation.
300A	12298	The automatic operation cannot continue because the controller is stopped by the ESTOP command during the NC channel automatic operation.	Check whether the controller was stopped by the ESTOP command during the NC channel automatic operation.
3011	12305	The automatic operation cannot continue because NC X axis is not ready for operation.	Check whether the NC X axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3012	12306	The automatic operation cannot continue because NC Y axis is not ready for operation.	Check whether the NC Y axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3013	12307	The automatic operation cannot continue because NC Z axis is not ready for operation.	Check whether the NC Z axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3014	12308	The automatic operation cannot continue because NC A axis is not ready for operation.	Check whether the NC A axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3015	12309	The automatic operation cannot continue because NC B axis is not ready for operation.	Check whether the NC B axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3016	12310	The automatic operation cannot continue because NC C axis is not ready for operation.	Check whether the NC C axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3017	12311	The automatic operation cannot continue because NC U axis is not ready for operation.	Check whether the NC U axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3018	12312	The automatic operation cannot continue because NC V axis is not ready for operation.	Check whether the NC V axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3019	12313	The automatic operation cannot continue because NC W axis is not ready for operation.	Check whether the NC W axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
301A	12314	The automatic operation cannot continue because NC S axis is not ready for operation.	Check whether the NC S axis is in servo-off state or drive alarm state. The NC channel automatic operation can be executed when the configuration axis is in servo-on state, or drive alarm does not occur.
3020	12320	It is undefined NC channel command.	The NC command cannot be supported in the current controller version. Check the version in which the command can be

Error code		Error Description	Solutions
Hex	Dec		
			executed and contact the customer support team of our company.
3021	12321	The previously executed command was canceled because the same NC channel command was executed.	The NC channel command can be executed only once per scan. Change the operating condition of the program so that one NC channel command can be executed in one scan.
3030	12336	The automatic operation cannot continue because interpreter (IPR) alarm occurs during the NC channel automatic operation.	Check the interpreter (IPR) error code among the NC channel flags. You can execute the automatic operation start command (NC_CycleStart) again after resetting the NC channel with the NC reset command (NC_Reset).
3031	12337	The automatic operation cannot continue because program processor(PA) alarm occurs during the NC channel automatic operation.	Check the program processor (PA) error code among the NC channel flags. You can execute the automatic operation start command (NC_CycleStart) again after resetting the NC channel with the NC reset command (NC_Reset).
3040	12352	The command position setting value was out of the pulse unit expression value during NC channel automatic operation.	It exceeded a 32-bit area when the command position value was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3041	12353	The command position setting value of NC X axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC X axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3042	12354	The command position setting value of NC Y axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC Y axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3043	12355	The command position setting value of NC Z axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC Z axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3044	12356	The command position setting value of NC A axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC A axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3045	12357	The command position setting value of NC B axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC B axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it

Error code		Error Description	Solutions
Hex	Dec		
			is possible to set the range to 48-bit INT)
3046	12358	The command position setting value of NC C axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC C axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3047	12359	The command position setting value of NC U axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC U axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3048	12360	The command position setting value of NC V axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC V axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3049	12361	The command position setting value of NC W axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC W axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
304A	12362	The command position setting value of NC S axis was out of the pulse unit expression value.	It exceeded a 32-bit area when the command position value of NC S axis was converted in pulse unit. Set the value in the range of -2,147,483,648 to 2,147,483,647 when converting the command position value to pulse. (When using the 'Position Control Range Expansion' function, it is possible to set the range to 48-bit INT)
3050	12368	The command position of the NC channel configuration axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in the NC configuration axis where the error occurred, and then remove the error by executing the error reset command.
3051	12369	The command position of NC X axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC X axis, and then remove the error by executing the error reset command.
3052	12370	The command position of NC Y axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC Y axis, and then remove the error by executing the error reset command.
3053	12371	The command position of NC Z axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC Z axis, and then remove the error by executing the error reset command.
3054	12372	The command position of NC A axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC A axis, and then remove the error by executing the error reset command.

Error code		Error Description	Solutions
Hex	Dec		
3055	12373	The command position of NC B axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC B axis, and then remove the error by executing the error reset command.
3056	12374	The command position of NC C axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC C axis, and then remove the error by executing the error reset command.
3057	12375	The command position of NC U axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC U axis, and then remove the error by executing the error reset command.
3058	12376	The command position of NC V axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC V axis, and then remove the error by executing the error reset command.
3059	12377	The command position of NC W axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC W axis, and then remove the error by executing the error reset command.
305A	12378	The command position of NC S axis was out of the software upper limit position.	Deviate from the software upper limit range by using the reverse jog command in NC S axis, and then remove the error by executing the error reset command.
305B	12379	The command position of the NC channel/axis was out of the inner range of G22 traverse prohibited area.	Deviate from the range of G22 traverse prohibited area by using the jog command in the NC configuration axis where the error occurred, and then remove the error by executing the error reset command.
305C	12380	The command position of the NC channel/axis was out of the outer range of G22 traverse prohibited area.	Deviate from the range of G22 traverse prohibited area by using the jog command in the NC configuration axis where the error occurred, and then remove the error by executing the error reset command.
305D	12381	The command position of the NC channel/axis was outside the range of the third traverse prohibited area.	Deviate from the range of the third traverse prohibited area by using the jog command in the NC configuration axis where the error occurred, and then remove the error by executing the error reset command.
3060	12384	The command position of the NC channel configuration axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in the NC configuration axis where the error occurred, and then remove the error by executing the error reset command.
3061	12385	The command position of NC X axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC X axis, and then remove the error by executing the error reset command.
3062	12386	The command position of NC Y axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC Y axis, and then remove the error by executing the error reset command.
3063	12387	The command position of NC Z axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC Z axis, and then remove the error by executing the error reset command.
3064	12388	The command position of NC A axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC A axis, and then remove the error by executing the error reset command.
3065	12389	The command position of NC B axis was out of	Deviate from the software lower limit range by using the forward

Error code		Error Description	Solutions
Hex	**Dec**		
		the software lower limit position.	jog command in NC B axis, and then remove the error by executing the error reset command.
3066	12390	The command position of NC C axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC C axis, and then remove the error by executing the error reset command.
3067	12391	The command position of NC U axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC U axis, and then remove the error by executing the error reset command.
3068	12392	The command position of NC V axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC V axis, and then remove the error by executing the error reset command.
3069	12393	The command position of NC W axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC W axis, and then remove the error by executing the error reset command.
306A	12394	The command position of NC S axis was out of the software lower limit position.	Deviate from the software lower limit range by using the forward jog command in NC S axis, and then remove the error by executing the error reset command.
3071	12401	The automatic operation cannot continue because NC X axis is not in the origin determination complete state.	Check whether NC X axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3072	12402	The automatic operation cannot continue because NC Y axis is not in the origin determination complete state.	Check whether NC Y axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3073	12403	The automatic operation cannot continue because NC Z axis is not in the origin determination complete state.	Check whether NC Z axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3074	12404	The automatic operation cannot continue because NC A axis is not in the origin determination complete state.	Check whether NC A axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3075	12405	The automatic operation cannot continue because NC B axis is not in the origin determination complete state.	Check whether NC B axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3076	12406	The automatic operation cannot continue because NC C axis is not in the origin determination complete state.	Check whether NC C axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3077	12407	The automatic operation cannot continue because NC U axis is not in the origin determination complete state.	Check whether NC U axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3078	12408	The automatic operation cannot continue because NC V axis is not in the origin determination complete state.	Check whether NC V axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
3079	12409	The automatic operation cannot continue because NC W axis is not in the origin determination complete state.	Check whether NC W axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).
307A	12410	The automatic operation cannot continue because NC S axis is not in the origin determination complete state.	Check whether NC S axis is in the undetermined origin state. You can change the axis to the determined origin state by using the homing command (MC_Home, NC_Home).

Error code		Error Description	Solutions
Hex	Dec		
3080	12416	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of the NC configuration axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of the NC configuration axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3081	12417	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC X axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC X axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3082	12418	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC Y axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC Y axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3083	12419	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC Z axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC Z axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3084	12420	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC A axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC A axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3085	12421	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC B axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC B axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3086	12422	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC C axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC C axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3087	12423	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC U axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC U axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3088	12424	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC V axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC V axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3089	12425	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC W axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC W axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
308A	12426	The automatic operation cannot continue because drive abnormal condition (upper/lower limit, alarm, servo off) of NC S axis occurs during the NC channel automatic operation.	Remove the cause for abnormal condition after checking whether the drive status of NC S axis was changed to the upper limit/lower limit, or alarm occurrence or servo-off state during the NC channel automatic operation.
3100	12544	NC channel parameter data is abnormal.	Download the data again from XG5000 and place requests for A/S if it occurs repeatedly after re-execution.

Error code		Error Description	Solutions
Hex	**Dec**		
3101	12545	Operation cannot be executed due to the NC parameter abnormality.	Check the NC parameter and reset it if the settings such as the data range are not correct.
3102	12546	The cutting feed upper/lower limit rate setting value of NC channel parameter exceeded the range.	Set the cutting feed upper/lower limit rate value of the NC channel parameter to a value greater than 0. Set the cutting feed upper limit rate value to be larger than the cutting feed lower limit rate value.
3103	12547	The circular processing speed limit upper/lower limit cutting feed rate setting value of NC channel parameter exceeded the range.	Set the arc processing speed limit upper/lower limit cutting speed value of the NC channel parameter to a value greater than 0. Set the arc processing speed limit upper limit speed value to be larger than the arc processing speed limit lower limit speed value.
3104	12548	When controlling the constant surface speed of the NC channel parameter, the setting value of the maximum/minimum rotation number in the spindle exceeded the range.	When controlling the constant surface speed of the NC channel parameter, set the value of the maximum rotation number in the spindle larger than the value of the minimum rotation number in the spindle.
3200	12800	NC channel/axis parameter data is abnormal.	Download the data again from XG5000 and place requests for A/S if it occurs repeatedly after re-execution.
3310	13072	NC Feed Hold command was executed in a state other than automatic operation, or the currently executed program block cannot execute NC Feed Hold.	Execute NC Feed Hold command (NC_FeedHold) when the NC channel is in automatic operation. Check whether the currently executed program block is ready for Feed Hold if it is in automatic operation.
3320	13088	The override factor of the NC rapid traverse override command exceeded the range.	Execute the override command after setting the VelFactor, AccFactor and JerkFactor value of the override command to be greater than 0.
3321	13089	The operation speed value of NC X axis exceeded the speed limit value after the override factor of NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC X axis.
3322	13090	The operation speed value of NC Y axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC Y axis.
3323	13091	The operation speed value of NC Z axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC Z axis.
3324	13092	The operation speed value of NC A axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC A axis.
3325	13093	The operation speed value of NC B axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC B axis.
3326	13094	The operation speed value of NC C axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC C axis.

| Error code | | Error Description | Solutions |
Hex	Dec		
3327	13095	The operation speed value of NC U axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC U axis.
3328	13096	The operation speed value of NC V axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC V axis.
3329	13097	The operation speed value of NC W axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC W axis.
332A	13098	The operation speed value of NC S axis exceeded the speed limit value after the override factor of the NC rapid traverse override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC S axis.
332B	13099	The reset by NC_Reset or NC_Emergency command was executed, and thus the NC rapid traverse override command was canceled.	Execute the command again after NC_Reset or NC_Emergency command ends, if the NC rapid traverse override function was needed.
332C	13100	The NC_RapidTraverseOverride command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3330	13104	The override factor of NC cutting feed override command exceeded the range.	Execute the override command after setting the VelFactor, AccFactor and JerkFactor value of the override command to be greater than 0.
3331	13105	The operation speed value exceeded the cutting feed upper limit rate value after the override factor of NC cutting feed override command was reflected.	Perform the override within the range that does not exceed the cutting feed upper limit rate value after checking the cutting feed upper limit rate value of the NC channel parameter.
3332	13106	The reset by NC_Reset or NC_Emergency command was executed, and thus the NC cutting feed override command was canceled.	Execute the command again after NC_Reset or NC_Emergency command ends, if the NC cutting feed override function was needed.
3333	13107	The NC_CuttingFeedOverride command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
350A	13578	The NC_CycleStart command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3340	13120	The override factor of NC spindle override command exceeded the range.	Execute the override command after setting the VelFactor, AccFactor and JerkFactor value of the override command to be greater than 0.
3341	13121	The operation speed value of the spindle exceeded the speed limit value after the override factor of the NC spindle override command was reflected.	Perform the override within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to the spindle axis.
3342	13122	The NC_SpindleOverride command cannot be executed when it is resetting by NC_Reset or	Execute the command again after NC_Reset or NC_Emergency command ends, if the NC spindle override

Error code		Error Description	Solutions
Hex	Dec		
		NC_Emergency command.	function was needed.
3343	13123	The NC_SpindleOverride command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3344	13124	The Spindle Override command cannot be executed because the spindle axis (NC S axis) is not ready for operation.	Check whether the spindle axis (NC S axis) is in the servo-off state or drive alarm state.
3350	13136	The setting axis of the NC parameter read command was not enabled as the NC axis.	Check whether the setting axis of the NC parameter read command was registered as NC channel/axis parameter. The NC channel/axis can be registered in the NC channel parameter among the motion data items of XG5000.
3351	13137	The axis setting value of the NC parameter read command exceeded the allowable range.	Set the axis number to a value between 0 and 10. Perform channel parameter reading if the axis value is 0 and NC axis X ~ NC axis S reading if 1 ~ 10.
3352	13138	The parameter group setting value of the NC parameter read command exceeded the allowable range.	The setting range of the parameter group is 1 ~ 17 for the channel parameter, and 1 ~ 5 for the channel/axis parameter. Check the group number to which the parameter you want to read belongs, and then execute the parameter read command (NC_ReadParameter).
3353	13139	The parameter number set in the parameter group of the NC parameter read command is not supported.	Check whether the parameter number set in channel parameter or channel/axis parameter group is supported. Check the group number to which the parameter you want to read belongs and the parameter number, and then execute the parameter read command (NC_ReadParameter).
3362	13154	The NC_MirrorImage command was executed in a non-SingleBlock state	When you want to execute the automatic operation mirror image (NC_MirrorImage) command, first create a single block state with the block operation designation (NC_BlockControl) command and then execute it..
3370	13168	If the reverse operation buffer is 0, it cannot be executed.	Set the reverse operation buffer value of the NC channel parameter to a value between 1 and 50.
3380	13184	The block skip function is canceled as the reset by the NC_Reset or NC_Emergency command is executed.	If the block skip function is needed, execute the command again after the NC_Reset or NC_Emergency command ends.
3381	13185	Machine position gets out of a measurable range. (G37)	After confirming the measurable range of machine position, execute the automatic machine measurement (NC_BlockSkip) command.
3500	13568	The automatic operation start operation cannot be executed if the NC channel is in automatic operation.	Check whether NC channel is currently running automatically. The automatic operation can be restarted after the automatic operation is completed.
3501	13569	The automatic operation start operation cannot be executed because NC Feed Hold command is in Enable status.	Execute the automatic operation start command (NC_CycleStart) again after releasing the Enable input of the NC Feed Hold command (NC_FeedHold).
3502	13570	The automatic operation start operation cannot be executed because NC emergency stop command is in Enable status.	Execute the automatic operation start command (NC_CycleStart) again after releasing the Enable input of the NC emergency stop command (NC_Emergency).
3503	13571	The operation start command cannot be executed when NC interpreter(IPR) is not	Execute the automatic operation start command (NC_CycleStart) again after resetting the NC channel with the

Error code		Error Description	Solutions
Hex	**Dec**		
		terminated normally.	NC reset command(NC_Reset).
3504	13572	The automatic operation start command cannot be executed because NC interpreter(IPR) or program processor(PA) is in error state.	Check the interpreter(IPR) and program processor(PA) error code in the NC channel flags. The automatic operation start command (NC_CycleStart) can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3505	13573	The automatic operation start command cannot be executed because the program to be executed on the NC channel is not set.	Execute the automatic operation start command (NC_CycleStart) again after designating the program to be executed with the NC program designation command(NC_LoadProgram).
3506	13574	The automatic operation start command cannot be executed because the NC channel reached the target processing quantity or the target processing quantity in M99 repeat processing.	Confirm the processing quantity of the NC channel flag or the processing quantity in M99 repeat processing, and then check whether it has reached the target processing quantity. Execute the automatic operation start command (NC_CycleStart) again after resetting the processing quantity or the processing quantity flag in M99 repeat processing
3507	13575	The automatic operation start command cannot be executed if NC M/S/T-code output strobe signal is on.	Execute the automatic operation start command (NC_CycleStart) after completing the NC M/S/T-code output strobe signal.
3508	13576	The NC channel interpreter (IPR) was not executed normally.	Execute the automatic operation start command (NC_CycleStart) again after resetting the NC channel with the NC reset command (NC_Reset).
3509	13577	The automatic operation start command cannot be executed when entered into the NC traverse prohibited area.	Deviate from the range of the traverse prohibited area by using the jog command in the NC configuration axis where an error occurred, and then remove the error by executing the error reset command.
350A	13578	The NC_CycleStart command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
350B	13579	As the NC spindle device does not support csv or vl modes, the automatic operation start command cannot be executed.	Check if the slave connected to the spindle axis supports csv or vl operation modes. If the spindle device does not support csv or vl operation modes, the operation with the spindle axis cannot be performed.
350C	13580	As there is no essential object needed for the spindle operation in PDO setting of the EtherCAT slave connected to the spindle axis, the automatic operation start command cannot be executed.	After reconnecting the EtherCAT slave connection by setting the essential object needed for the spindle operation in PDO setting of the EtherCAT slave connected to the spindle axis, execute the automatic operation start command (NC_CycleStart). (See the essential PDO setting of the spindle device in the User Manual)
350D	13581	As the reverse operation buffer is all consumed during reverse operation, the automatic operation start command cannot be executed.	After releasing the Enable input of the NC_Reset or NC_RetraceMove commands, execute the automatic operation start command (NC_CycleStart) again.
350F	13583	Automatic operation start command cannot be executed while writing the downloaded NC program during run.	Execute the automatic operation start command after completing the writing of the NC program downloaded during run.
3510	13584	The automatic operation start command cannot be executed because NC channel configuration axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT

Error code		Error Description	Solutions
Hex	**Dec**		
			network or set as virtual axes.
3511	13585	The automatic operation start command cannot be executed because NC X axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3512	13586	The automatic operation start command cannot be executed because NC Y axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3513	13587	The automatic operation start command cannot be executed because NC Z axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3514	13588	The automatic operation start command cannot be executed because NC A axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3515	13589	The automatic operation start command cannot be executed because NC B axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3516	13590	The automatic operation start command cannot be executed because NC C axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3517	13591	The automatic operation start command cannot be executed because NC U axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3518	13592	The automatic operation start command cannot be executed because NC V axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3519	13593	The automatic operation start command cannot be executed because NC W axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
351A	13594	The automatic operation start command cannot be executed because NC S axis is not ready.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axes are all

Error code		Error Description	Solutions
Hex	**Dec**		
			ready. In order to start the automatic operation, the NC channel configuration axes should be connected to the EtherCAT network or set as virtual axes.
3520	13600	The automatic operation start command cannot be executed because NC channel configuration axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3521	13601	The automatic operation start command cannot be executed because NC X axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3522	13602	The automatic operation start command cannot be executed because NC Y axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3523	13603	The automatic operation start command cannot be executed because NC Z axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3524	13604	The automatic operation start command cannot be executed because NC A axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3525	13605	The automatic operation start command cannot be executed because NC B axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3526	13606	The automatic operation start command cannot be executed because NC C axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3527	13607	The automatic operation start command cannot be executed because NC U axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3528	13608	The automatic operation start command cannot be executed because NC V axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3529	13609	The automatic operation start command cannot be executed because NC W axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
352A	13610	The automatic operation start command cannot be executed because the NC S axis is in operation.	Execute the automatic operation start command (NC_CycleStart) when the NC channel configuration axis stops.
3530	13616	The automatic operation start command cannot be executed because NC channel configuration axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3531	13617	The automatic operation start command cannot be executed because NC X axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3532	13618	The automatic operation start command cannot be executed because NC Y axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3533	13619	The automatic operation start command cannot be executed because NC Z axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3534	13620	The automatic operation start command cannot be executed because NC A axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3535	13621	The automatic operation start command cannot be executed because NC B axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3536	13622	The automatic operation start command cannot	Execute the automatic operation start command

Error code		Error Description	Solutions
Hex	Dec		
		be executed because NC C axis is enabled as a motion axis group configuration axis.	(NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3537	13623	The automatic operation start command cannot be executed because NC U axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3538	13624	The automatic operation start command cannot be executed because NC V axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3539	13625	The automatic operation start command cannot be executed because NC W axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
353A	13626	The automatic operation start command cannot be executed because NC S axis is enabled as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3540	13632	The position unit or speed unit setting of NC channel configuration axis is invalid.	For NC operation, set the unit of the NC channel configuration axis (except for spindle) to mm or deg. Set the unit of speed to RPM for spindle axis(S axis) and unit/min for other axes(X, Y, Z, A, B, C, U, V, W).
3541	13633	The position unit or speed unit setting of NC X axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3542	13634	The position unit or speed unit setting of NC Y axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3543	13635	The position unit or speed unit setting of NC Z axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3544	13636	The position unit or speed unit setting of NC A axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3545	13637	The position unit or speed unit setting of NC B axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3546	13638	The position unit or speed unit setting of NC C axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3547	13639	The position unit or speed unit setting of NC U axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3548	13640	The position unit or speed unit setting of NC V axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
3549	13641	The position unit or speed unit setting of NC W axis is invalid.	For NC operation, set the unit of the NC channel configuration axis to mm or deg. Set the unit of speed to unit/min.
354A	13642	The position unit or speed unit setting of NC S axis is invalid.	For NC operation, set the speed unit of the spindle axis to RPM.
3600	13824	The program cannot be loaded because there is no program set in NC program designation command in the controller.	Execute the program designation command (NC_LoadProgram) again after writing the NC program created in XG5000 in the controller.
3601	13825	The program designation command cannot be executed if NC channel is in automatic operation.	Check whether the NC channel is currently running automatically. You can designate a new program after the automatic operation ends.
3602	13826	The program designation command cannot be executed due to the NC program data abnormality.	Download the data again from XG5000 after checking whether NC program data abnormal error (0x3001) occurs and place requests for A/S if it occurs repeatedly after re-execution.

Error code		Error Description	Solutions
Hex	Dec		
3603	13827	The LoadMode of the NC program designation command was invalid.	Execute the program designation command (NC_LoadProgram) again after entering a value of 0 in the LoadMode of the NC program designation command(NC_LoadProgram).
3604	13828	The NC_LoadProgram command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3605	13829	Program designation command cannot be executed while writing the downloaded NC program during run.	Execute the program designation command after completing the writing of the NC program downloaded during run.
3610	13840	The NC channel interpreter(IPR) was not reset normally.	Rest the NC channel with the NC reset command (NC_Reset). Place requests for A/S if it occurs repeatedly after re-execution.
3620	13856	The NC_Emergency command was executed in a state other than automatic operation.	Execute the NC emergency stop command (NC_Emergency) when the NC channel is in automatic operation.
3630	13872	It is not the range of origin that can be specified in NC homing operation.	The range of the origin(ReferenceNum) is from the first origin to the fourth origin. Specify it as a value between 1 and 4.
3631	13873	The NC homing command cannot be executed when the channel is in automatic operation.	Execute the homing command after the automatic operation is completed.
3632	13874	The homing command cannot be executed because the NC emergency stop command is in Enable state.	Execute the homing command (NC_Home) again after releasing the Enable input of the NC emergency stop command (NC_Emergency)
3633	13875	The homing command cannot be executed because the NC channel configuration axis is not ready.	Execute the homing command(NC_Home) with all the NC channel configuration axes ready. In order to perform the homing command, the NC channel configuration axis should be connected to the EtherCAT network or set as a virtual axis.
3634	13876	The homing command cannot be executed because the NC channel configuration axis is enabled as a motion axis group configuration axis.	Execute the homing command (NC_Home) with the NC channel configuration axis disabled as a motion axis group configuration axis.
3635	13877	Error occurred during homing operation in the servo drive.	Check the error factor of the servo drive and perform homing after removing the servo drive error.
3636	13878	The NC_Home command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3637	13879	If the spindle axis is automatically controlled by the NC function module, if the item 'Spindle Encoder Selection' of the axis parameter is '0: Disable', the homing operation cannot be executed.	After correctly setting the connection method of the encoder connected to the spindle axis in the item 'Spindle Encoder Selection' of the axis parameter, execute the homing operation command.
3638	13880	If the item 'Spindle Encoder Selection' of the axis parameter is '1: Motor ENC', as there is no 'Position actual value (0x6064)' object in the TxPDO setting of the EtherCAT slave connected to the spindle axis, the homing operation command cannot be executed.	If the item 'Spindle Encoder Selection' of the axis parameter is '1: Motor ENC', after reconnecting the EtherCAT by adding the 'Position actual value (0x6064)'object to the TxPDO setting of the EtherCAT slave connected to the spindle axis, execute the homing operation command.
3639	13881	If the item 'Spindle Encoder Selection' of the axis parameter is '2: Built-in ENC1', as the encoder1	If the item 'Spindle Encoder Selection' of the axis parameter is '2: Built-in ENC1', after setting the encoder1 unit of the encoder

Error code		Error Description	Solutions
Hex	**Dec**		
		parameter setting is wrong, the homing operation command cannot be executed.	parameter = pulse, the maximum value of the encoder1 = 2147483647 and the minimum value of the encoder1 = -2147483648, execute the homing operation command.
363A	13882	If the item 'Spindle Encoder Selection' of the axis parameter is '3: Built-in ENC2', as the encoder2 parameter setting is wrong, the homing operation command cannot be executed.	If the item 'Spindle Encoder Selection' of the axis parameter is '3: Built-in ENC2', after setting the encoder2 unit of the encoder parameter = pulse, the maximum value of the encoder2 = 2147483647 and the minimum value of the encoder2 = -2147483648, execute the homing operation command.
3640	13888	NC M-code operation completion command cannot be executed when M-code output Strobe signal is off.	Execute the M-code operation completion command (NC_McodeComplete) with the M-code output Strobe signal on after checking the status of the M-code output Strobe signal among the NC channel flags.
3641	13889	The NC_McodeComplete command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3650	13904	NC S-code operation completion command cannot be executed when S-code output Strobe signal is off.	Execute the S-code operation completion command (NC_McodeComplete) with the M-code output Strobe signal on after checking the status of the M-code output Strobe signal among the NC channel flags.
3651	13905	The NC_ScodeComplete command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3660	13920	NC T-code operation completion command cannot be executed when T-code output Strobe signal is off.	Execute the T-code operation completion command (NC_McodeComplete) with the T-code output Strobe signal on after checking the status of the T-code output Strobe signal among the NC channel flags.
3661	13921	The NC_TcodeComplete command cannot be executed when it is resetting by NC_Reset or NC_Emergency command.	Execute the NC command after NC_Reset or NC_Emergency command ends.
3670	13936	The NC parameter write command cannot be executed when the channel is in automatic operation.	Check whether NC channel is currently in automatic operation. Execute the NC parameter write command (NC_WriteParameter) in stop state after the automatic operation ends.
3671	13937	The setting axis of the NC parameter write command was not enabled as the NC axis.	Check whether the setting axis of the NC parameter write command was registered as NC channel/axis parameter. The NC channel/axis can be registered in the NC channel parameter among the motion data items of XG5000.
3672	13938	The axis setting value of the NC parameter write command exceeded the allowable range.	Set the axis number to a value between 0 and 10. Perform channel parameter writing if the axis value is 0 and NC axis X ~ NC axis S writing if 1 ~ 10.
3673	13939	The parameter group setting value of the NC parameter write command exceeded the allowable range.	The setting range of the parameter group is 1 ~ 17 for the channel parameter, and 1 ~ 5 for the channel/axis parameter. Check the group number to which the parameter you want to write belongs, and then execute the parameter write command(NC_WriteParameter).
3674	13940	The parameter number set in the parameter group of the NC parameter write command is not supported.	Check whether the parameter number set in channel parameter or channel/axis parameter group is supported. Check the group number to which the parameter you want to write belongs and

Error code		Error Description	Solutions
Hex	**Dec**		
			the parameter number, and then execute the parameter write command(NC_WriteParameter).
3675	13941	The data setting value of the parameter set in the NC parameter write command exceeded the range.	Check the data setting range of the parameter to be set, and then execute the parameter write command(NC_WriteParameter) with a value within the range.
3690	13968	Only the tool retract/recover mode cancellation (0), the tool retract mode (1) or the tool recover mode (2) commands can be made.	After specifying the tool retract/recover mode cancellation (0), the tool retract mode (1) or the tool recover mode (2) in the ToolMode input, execute the tool retract/recover (NC_ToolMode) commands.
3691	13969	If it is not the tool retract mode, the tool recover command cannot be made.	After completing the tool retract in the tool retract mode (1), execute the tool recover mode (2) command.
3692	13970	If it is the tool retract mode, if tool retract is not operated, the tool recover command cannot be made.	After executing the tool retract operation with JOG operation in the tool retract mode, execute the tool recover mode (2) command.
36A0	13984	The Block Optional Skip number gets out of range. (0~9)	Set the value to be set in SkipNum to one value among 0~9 and execute the command.
36B0	14000	The value to set the axis of the manual tool compensation command exceeded the tolerance range. (X, Y, Z)	Set one of X~Z (1~3) axes to NcAxis and execute the command.
36C1	14017	The maximum velocity value of the spindle axis Less than or equal to 0.	Set the MaxVelocity value of the NC_ChgSpindleGear command as a value greater than 0 and execute the command.
36C2	14018	The setting value of the gear ratio on the motor side is less than or equal to 0.	Set the GearOfMotor value of the NC_ChgSpindleGear command as a value greater than 0 and execute the command.
36C3	14019	The setting value of the gear ratio on the machine side is less than or equal to 0.	Set the GearOfMachine value of the NC_ChgSpindleGear command as a value greater than 0 and execute the command.
36C4	14020	The setting value of Backlash is less than 0.	Set the Backlash value of the NC_ChgSpindleGear command as a value greater than 0 and execute the command.
36C5	14021	The setting value of P Gain is less than 0 or greater than 500.	Set the P_Gain value of the NC_ChgSpindleGear command to a value from 0 to 500 and execute the command.
36C6	14022	The setting value of FF Gain is less than 0 or greater than 100.	Set the FF_Gain value of the NC_ChgSpindleGear command to a value from 0 to 100 and execute the command.
36C7	14023	The spindle axis is not in operation with NC automatic operation.	Operate the spindle axis with NC automatic operation and perform the NC_ChgSpindleGear operation.
36D0	14032	As the main spindle axis is not set to be automatically controlled by the NC function module, the spindle control command cannot be executed.	After setting the spindle axis (S axis) to the NC channel/axis of the NC parameter, set the number of the motor axis connected to the spindle axis (S axis) to the 'number of the main spindle axis' of the NC channel parameter.
36D1	14033	The spindle control command cannot be executed because the spindle axis is not ready for operation.	The motor axis connected to the spindle axis is not ready for operation. After making the axis ready for operation with the LS_Connect command, etc., execute the spindle control command.
36B1	14001	As the axis of the manual tool compensation command is not ready for a configuration axis, the command cannot be executed.	Set the axis set in NcAxis to a configuration axis and execute the command.
36B2	14002	The axis of the manual tool compensation command is not activated by the NC axis.	Allocate the NC axis set in NcAxis and execute the command.
36B3	14003	Manual tool compensation commands cannot be	After automatic operation is finished, execute the homing

Error code		Error Description	Solutions
Hex	Dec		
		executed when the channel is in automatic operation.	command.
36C0	14016	The setting value of the speed to change gears of the spindle axis exceeded speed limit values.	Set the velocity to be set in ChgVelocity of the NC_ChgSpindleGear command within the range of speed limit values of the relevant axis and execute the command.
36C1	14017	Spindle axis maximum speed value less than or equal to zero	Set the MaxVelocity value of the NC_ChgSpindleGear command to a value greater than 0 and execute the command.
36C2	14018	The gear ratio setting value on the motor side is less than or equal to 0.	Execute the command after setting the GearOfMotor setting value of the NC_ChgSpindleGear command to a value greater than 0.
36C3	14019	Gear ratio setting value on machine side is less than or equal to 0.	Execute the command after setting the GearOfMachine setting value of the NC_ChgSpindleGear command to a value greater than 0.
36C4	14020	The Backlash setting is less than 0.	Execute the command after setting the backlash setting value of the NC_ChgSpindleGear command to a value greater than or equal to 0.
36C5	14021	The P Gain setting is less than 0 or greater than 500	Set the P_Gain setting value of the NC_ChgSpindleGear command to a value in the range of 0 to 500 and execute the command.
36C6	14022	The FF Gain setting is less than 0 or greater than 100.	Execute the command after setting the FF_Gain value of the NC_ChgSpindleGear command to a value in the range of 0 to 100
36C7	14023	The spindle axis is not operating in NC automatic operation.	Operate the spindle axis with NC automatic operation and perform NC_ChgSpindleGear operation.
36D0	14032	The spindle control command cannot be executed because the main spindle axis is not set to be automatically controlled by the NC function module	After setting the spindle axis (S-axis) in NC channel/axis of NC parameter, set the motor axis number connected to the spindle axis (S-axis) in 'Main spindle axis number' of NC channel paramete
36D1	14033	Spindle control command cannot be executed because the spindle axis is not ready for operation.	The motor axis connected to the spindle axis is not currently ready for operation. After making the axis ready for operation with the LS_Connect command, etc., execute the spindle control command.
36E0	14048	The SD card is not properly installed.	Please check if the SD card is properly installed and is ready for use.
36E1	14049	SrcProgramName The program does not exist.	Please check if the program of SrcProgramName is saved, and input the program name of the language correctly.
36E2	14050	DstProgramName This program has been set to overwrite protection.	If you want to modify the program of DstProgramName, please set to allow overwrite when executing command.
36E3	14051	An error occurred while reading the file.	Please check if the status of the saved file is normal.
3800	14336	In the NC rapid traverse command, the operation speed value of the configuration axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to the NC configuration axis where error occurred.
3801	14337	In the NC rapid traverse command, the operation speed value of NC X axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC X axis where error occurred.
3802	14338	In the NC rapid traverse command, the operation	Set the rapid traverse rate within the range that does not exceed

Error code		Error Description	Solutions
Hex	Dec		
		speed value of NC Y axis exceeded the speed limit value.	the speed limit value after checking the speed limit value of the axis connected to NC Y axis where error occurred.
3803	14339	In the NC rapid traverse command, the operation speed value of NC Z axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC Z axis.
3804	14340	In the NC rapid traverse command, the operation speed value of NC A axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC A axis.
3805	14341	In the NC rapid traverse command, the operation speed value of NC B axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC B axis.
3806	14342	In the NC rapid traverse command, the operation speed value of NC C axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC C axis.
3807	14343	In the NC rapid traverse command, the operation speed value of NC U axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC U axis.
3808	14344	In the NC rapid traverse command, the operation speed value of NC V axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC V axis.
3809	14345	In the NC rapid traverse command, the operation speed value of NC W axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC W axis.
380A	14346	In the NC rapid traverse command, the operation speed value of NC S axis exceeded the speed limit value.	Set the rapid traverse rate within the range that does not exceed the speed limit after checking the speed limit value of the axis connected to NC S axis.
380B	14347	In-position of rapid traverse component axes was not completed within in-position completion monitoring time during the NC rapid traverse operation.	Check the command in-position width and the in-position completion monitoring time of the NC channel parameter among the parameters connected to the NC axis.
380C	14348	The automatic operation cannot continue because there is an axis that has an error among NC configuration axes during the NC rapid traverse operation.	Check the axis where the error occurred among the NC configuration axes. You can check the error that occurred on the NC axis in the axis error code number of the NC channel/axis flag.
3810	14352	In NC feed per rotation mode, the cutting feed operation rate was set to 0.	Set the rate of the cutting feed operation to a value other than 0 in NC feed per rotation mode.
3811	14353	The operation speed of NC cutting feed command exceeded the cutting feed upper limit rate value of the NC channel parameter.	Set the cutting feed rate value(F) within the range that does not exceed the parameter value after checking the cutting feed upper limit rate value of the NC channel parameter.
3812	14354	The in-position of the cutting feed configuration axes was not completed within the in-position completion monitoring time during the NC cutting feed operation.	Check the command in-position width and the in-position completion monitoring time of the NC channel parameter among the parameters connected to the NC axis.
3820	14368	In the NC cutting feed command, the operation speed value of the configuration axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to the NC configuration axis where error occurred.

Error code		Error Description	Solutions
Hex	Dec		
3821	14369	In the NC cutting feed command, the operation speed value of NC X axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC X axis.
3822	14370	In the NC cutting feed command, the operation speed value of NC Y axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC Y axis.
3823	14371	In the NC cutting feed command, the operation speed value of NC Z axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC Z axis.
3824	14372	In the NC cutting feed command, the operation speed value of NC A axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC A axis.
3825	14373	In the NC cutting feed command, the operation speed value of NC B axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC B axis.
3826	14374	In the NC cutting feed command, the operation speed value of NC C axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC C axis.
3827	14375	In the NC cutting feed command, the operation speed value of NC U axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC U axis.
3828	14376	In the NC cutting feed command, the operation speed value of NC V axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC V axis.
3829	14377	In the NC cutting feed command, the operation speed value of NC W axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC W axis.
382A	14378	In the NC cutting feed command, the operation speed value of NC S axis exceeded the speed limit value.	Set the cutting feed rate within the range that does not exceed the speed limit value after checking the speed limit value of the axis connected to NC S axis.
3840	14400	In NC circular interpolation, operation cannot be executed in case of start point = center point or center point = end point.	In NC circular interpolation, set the position of the center point to a different position from the start point(or end point).
3841	14401	The radius setting was incorrect in NC circular interpolation.	The radius value of the circle where the NC circular interpolation operation can be executed is greater than 0 and less than or equal to 2,147,483,647pulse based on the pulse unit. Set the center point or radius input value so that the radius can be calculated within the setting range. (When using the 'Position Control Range Expansion' function
3850	14416	Axis designation was incorrect in NC cylindrical interpolation.	In performing circular interpolation operation in NC cylindrical interpolation, Y-axis should be designated in the XY plane, Z-axis in the YZ plane, and Z-axis in the ZX plane.
3860	14432	The rest method is specified as the number of rotations, but the number of rotations is 0.	Run the S-axis with MC_MoveVelocity in the NC program.
3870	14448	As an error occurred in the spindle axis during automatic operation of the NC channel, the automatic operation cannot continue.	After making the spindle axis ready for operation again by confirming the error code that occurred in the spindle axis, execute the automatic operation start command

Error code		Error Description	Solutions
Hex	Dec		
			(NC_CycleStart).
3871	14449	The operation mode of the slave connected to the spindle axis cannot be changed.	Check if the slave connected to the spindle axis supports csv or vl operation modes. And confirm if the operation state of the slave is normal.
3872	14450	As the spindle axis is not in the state of homing completion, the orientation operation cannot be executed.	After making the homing state by executing NC_Home or MC_SetPosition commands on the spindle axis, execute the spindle orientation operation.
3873	14451	If the spindle axis is automatically controlled by the NC function module, if the item 'Spindle Encoder Selection' of the axis parameter is '0: Disable', the orientation operation (M19) cannot be executed.	After correctly setting the connection method of the encoder connected to the spindle axis in the item 'Spindle Encoder Selection' of the axis parameter, execute the orientation operation (M19).
3874	14452	If the item 'Spindle Encoder Selection' of the axis parameter is '1: Motor ENC', as there is no 'Position actual value (0x6064)' object in the TxPDO setting of the EtherCAT slave connected to the spindle axis, the orientation operation (M19) cannot be executed.	After reconnecting the EtherCAT by adding the 'Position actual value (0x6064)'object to the TxPDO setting of the EtherCAT slave connected to the spindle axis, execute the orientation operation (M19).
3875	14453	If the item 'Spindle Encoder Selection' of the axis parameter is '2: Built-in ENC1', as the encoder1 parameter setting is wrong, the orientation operation (M19) cannot be executed.	If the item 'Spindle Encoder Selection' of the axis parameter is '2: Built-in ENC1', after setting the encoder1 unit of the encoder parameter = pulse, the maximum value of the encoder1 = 2147483647 and the minimum value of the encoder1 = -2147483648, execute the orientation operation (M19).
3876	14454	If the item 'Spindle Encoder Selection' of the axis parameter is '3: Built-in ENC2', as the encoder2 parameter setting is wrong, the orientation operation (M19) cannot be executed.	If the item 'Spindle Encoder Selection' of the axis parameter is '3: Built-in ENC2', after setting the encoder2 unit of the encoder parameter = pulse, the maximum value of the encoder2 = 2147483647 and the minimum value of the encoder2 = -2147483648, execute the orientation operation (M19).
3880	14464	The reference axis of constant surface speed is not set. Shifting to the constant surface speed (G96) mode is not possible.	When controlling constant surface speed of the NC channel parameter, set the setting value of the reference axis to '1: X~ 9: W' range values and execute the command.
3F00	16128	Interpreter (IPR) parsing error - LEX MAIN TABLE configuration is invalid.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F01	16129	Interpreter(IPR) parsing error - Undefined character exists.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F02	16130	Interpreter (IPR) parsing error - Number exceeded the maximum buffer.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F03	16131	Interpreter(IPR) parsing error - The number of LEX tokens exceeded the maximum buffer.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).

Error code		Error Description	Solutions
Hex	Dec		
3F04	16132	Interpreter(IPR) parsing error - There are one or more decimal points.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F05	16133	Interpreter (IPR) parsing error - The number of brackets in the formula is incorrect.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F06	16134	Interpreter(IPR) parsing error - There exist characters that cannot be used in the formula.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F07	16135	Interpreter(IPR) parsing error - The syntax of the formula is incorrect.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F08	16136	Interpreter (IPR) parsing error - It is not a permitted macro variable.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F09	16137	Interpreter (IPR) parsing error - It is a TANGENT operation error.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F0A	16138	Interpreter(IPR) parsing error - It is a SQUARE ROOT operation error.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F0B	16139	Interpreter(IPR) parsing error - The denominator of division cannot be 0.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F0C	16140	Interpreter(IPR) parsing error - Syntax is incorrect.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F0D	16141	Interpreter(IPR) parsing error - YACC MAIN TABLE configuration is invalid.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F0E	16142	Interpreter(IPR) parsing error - The number of YACC tokens exceeded the maximum buffer.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F0F	16143	Interpreter (IPR) parsing error - It is not possible to open IPR semaphore.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).

| Error code | | Error Description | Solutions |
Hex	Dec		
3F10	16144	Interpreter(IPR) parsing error - It was terminated without M02 or M30.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F11	16145	Interpreter(IPR) parsing error - It can be commanded only at the head of the block.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F12	16146	Interpreter(IPR) parsing error - The same progress block exists.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F13	16147	Interpreter(IPR) parsing error - The number of statement numbers exceeded the maximum buffer.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F14	16148	Interpreter (IPR) parsing error - It is not possible to find the next block to proceed with.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F15	16149	Interpreter(IPR) parsing error - Subprogram call syntax is incorrect.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F16	16150	Interpreter(IPR) parsing error - It exceeded the maximum subprogram call.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F17	16151	Interpreter(IPR) parsing error - It is a program that was already called.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F18	16152	Interpreter (IPR) parsing error - There is no M99 in the subprogram.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F19	16153	Interpreter (IPR) parsing error - M99 syntax is incorrect.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F1A	16154	Interpreter (IPR) parsing error - There are a large number of loops.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F1B	16155	Interpreter(IPR) parsing error - There is no start of loop.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).

Error code		Error Description	Solutions
Hex	Dec		
3F1C	16156	Interpreter(IPR) parsing error - The connection of loops is invalid.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F1D	16157	Interpreter(IPR) parsing error - It exceeded M command limit in one block.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F1E	16158	Interpreter (IPR) parsing error - It is an unused G code.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F1F	16159	Interpreter (IPR) parsing error - It is not possible to be commanded simultaneously in one block.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F20	16160	Interpreter(IPR) parsing error - The center point of the arc cannot be found.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F21	16161	Interpreter(IPR) parsing error - It is not possible to create a path of cycle code.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F22	16162	Interpreter(IPR) parsing error - The taper amount of the cycle code is too large.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F23	16163	Interpreter(IPR) parsing error - It is not possible to be commanded within a cycle shape block.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F24	16164	Interpreter(IPR) parsing error - There is a problem with the cycle shape block command.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F26	16166	Interpreter(IPR) parsing error - The tool offset number is not valid.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F27	16167	Interpreter(IPR) parsing error - The center point of the arc is not correct.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F28	16168	Interpreter(IPR) parsing error - It is not possible to call the subprogram in MDI mode.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).

Error code		Error Description	Solutions
Hex	Dec		
3F29	16169	Interpreter(IPR) parsing error - Chamfering and rounding can applied only to the cutting feed command.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F2A	16170	Interpreter(IPR) parsing error - Chamfering and rounding are duplicated in instructions.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F2B	16171	Interpreter(IPR) parsing error - Only a single axis command is available in chamfering and rounding.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F2C	16172	Interpreter(IPR) parsing error - The chamfering and rounding reference value is greater than the feed amount.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F2D	16173	Interpreter(IPR) parsing error - It is not possible to obtain the next block information in chamfering and rounding.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F2E	16174	Interpreter(IPR) parsing error - An arc in the next block is not allowed in chamfering and rounding.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F2F	16175	Interpreter(IPR) parsing error - Rounding cannot be performed in the same straight line feed.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F30	16176	Interpreter(IPR) parsing error - Correction start and end can be made only in linear feed.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F31	16177	Interpreter(IPR) parsing error - There is no feed command in cycle shape end block.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F32	16178	Interpreter(IPR) parsing error - There is an axis command that is irrelevant to the plane in chamfering and rounding.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F33	16179	Interpreter(IPR) parsing error - It exceeded IJK command limit within one block in calling macro.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F34	16180	Interpreter(IPR) parsing error - Modal macro cannot be called from the subprogram.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).

Error code		Error Description	Solutions
Hex	Dec		
3F35	16181	Interpreter(IPR) parsing error - It exceeded the multiple call limits of modal macro.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F36	16182	Interpreter(IPR) parsing error - It is unused M code.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F37	16183	Interpreter(IPR) parsing error - Pitch cannot be calculated in rigid tapping.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F38	16184	Interpreter(IPR) parsing error - String exceeded the maximum buffer.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F39	16185	Interpreter(IPR) parsing error - String syntax is incorrect.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F3A	16186	Interpreter(IPR) parsing error - It reached the target processing quantity.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F3B	16187	Interpreter(IPR) parsing error - It is user stop of macro program.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F3C	16188	Interpreter(IPR) parsing error - It is not possible to create a path for compound thread cycle.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F3D	16189	Interpreter(IPR) parsing error - It cannot be commanded at polar coordinate interpolation.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F3E	16190	Interpreter (IPR) parsing error - It cannot be traversed to 0 at polar coordinate interpolation.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F3F	16191	Interpreter (IPR) parsing error - It is a syntax error in cylindrical interpolation command.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F40	16192	Interpreter(IPR) parsing error - It cannot be commanded during cylindrical interpolation.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).

Error code		Error Description	Solutions
Hex	Dec		
3F41	16193	Interpreter(IPR) parsing error - It is a constant surface speed control mode in polar coordinate and cylindrical interpolations.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F42	16194	Interpreter(IPR) parsing error - It is not the origin.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3F43	16195	Interpreter (IPR) parsing error - Tool interference occurred.	Check whether there is a program error in the block after confirming the 'error block number' among the NC channel flags. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3FE0	16352	Program processor (PA) error - There is no corresponding pointer location of the program file.	Execute the automatic operation again after resetting the NC channel with the NC reset command (NC_Reset).
3FE1	16353	Program processor (PA) error - It is not possible to read from the program file.	Execute the automatic operation again after resetting the NC channel with the NC reset command (NC_Reset).
3FE2	16354	Program processor (PA) error - The selected program file does not exist.	Check whether the specified program is stored in the controller. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).
3FE3	16355	Program processor (PA) error - It is not possible to open NcAccess semaphore.	Execute the automatic operation again after resetting the NC channel with the NC reset command (NC_Reset).
3FE4	16356	Program processor (PA) error - The number of characters per block is limited to 300.	Check whether the number of characters per block of the specified program exceeds 300. The automatic operation can be executed again after the NC channel is reset with the NC reset command (NC_Reset).

Appendix 2 Error Information and measurement

It describes the error information of the motion control module and how to deal with it.

(1) Function block error information

Error code		Error Description	Action
Hex	Dec		
0005	5	The current motion control module does not support the corresponding function block.	The motion function block is not executed in the version of current motion control module. Check the version in which the motion function block can be executed.
0006	6	Axis number of motion function block (Axis input) or encoder number (Encoder input) exceeded allowable range.	Set axis and encoder numbers with a range by product.
0007	7	Axis group number of function block (AxesGroup input) exceeded allowable range.	Set axis group number to a value between 1 and 16.
0008	8	NC channel of function block exceeded allowable range.	Check the range of NC channel, and set again.
0009	9	Slave number of function block (Slave input) exceeded allowable range.	Check the range of slave number, and set again.
000B	11	Input of function block exceeded allowable range.	Check the input range of function block, and set again.
000C	12	Array input of function block exceeded allowable range.	Check the array input size of function block, and set again.
0012	18	Internal execution error of function block occurred during the execution of the function block.	The current controller version may cause problems. Please check the supported version of XG5000 and controller.
0013	19	Motion response error occurred during the execution of function block.	The current controller version may cause problems. Please check the supported version of XG5000 and controller.
0014	20	CAM ID (CamTableID input) of function block exceeded allowable range.	Check the CAM ID range of function block, and set again.
0015	21	Jog operation was stopped by another command.	If re-execution of jog operation is necessary, turn Enable input off and then on.

(2) System error information

Error code		Error Description	Action
Hex	Dec		
000E	14	System error	A / S request if repeated on power up
0017	23	Program error	Startup after program modification and reloading
0019	25	Basic parameter error	Check retention status after uploading basic parameters If it is broken, fix it and download it again to check the operation. Exchange if the problem persists.
002B	43	Built-in parameter - Encoder error	Check retention status after uploading built-in parameters If it is broken, fix it and download it again to check the operation. Exchange if the problem persists.
002C	44	Axis parameter error	Re-download after modifying parameters
002D	45	Axis group parameter error	Re-download after modifying parameters

Error code		Error Description	Action
Hex	Dec		
002E	46	EtherCAT parameter error	Re-download after modifying parameters
002F	47	NC parameter error	Re-download after modifying parameters
0030	48	NC Program check error	Re-download after checking the program
0032	50	Significant error detection in external device	Repair and restart the wrong device by referring to the fault detection flag of the external device (according to the parameter).
0038	56	Main task period error	Check the main task and cycle flag and re-download after modifying the main task program or download by increasing the main task cycle of the basic parameter.
0039	57	Cycle task period error	Check the cycle task and cycle flag and re-download after modifying the cycle task program or download by increasing the cycle task cycle of the basic parameter
003A	58	Task program occupancy excess error	1) Secure the time for the system service to operate by reducing the amount of the user program execution within the main task/cycle task. 2) Please set the execution cycle of the main task/cycle task in the basic parameter higher to secure time for the internal service of the system to operate.
0080	128	The shared variable parameter error	Set the shared variable parameters of the motion control module again in the I/O parameters of XG5000. If you have added motion sharing variables, please download the corresponding program.
0FF9	4089	Failed to read shared variable parameter from CPU	Please check whether the motion control module has a supportable CPU version In the case of a CPU supporting motion control module, if it occurs repeatedly when power is re-applied, request A/S.
0FFE	4094	End due to abnormal operation	The module was shut down due to abnormal operation (module dropout, failure, etc.).
0FFF	4095	Backup ram data corruption	Retransmit data using XG5000 and operate. If it occurs repeatedly, please exchange the module.

(3) SD memory card error information

Error code		Error Description	Action
Hex	Dec		
Overall Error Codes			
0001	1	SD Card Recognition Error	Format in FAT32 format and connect SD memory.
0002	2	Partition Information Error	Format in FAT32 format and connect SD memory.
0003	3	File system error	Format in FAT32 format and connect SD memory.
0004	4	SD Card Not Supported	Please connect SD Card with storage of 2GB~512GB.
0005	5	SD card capacity check error	SD memory capacity test failed, and thus SD cannot be used. Please replace SD memory or format the memory before reconnecting.
0006	6	SD Card Capacity Exceeded	SD memory storage is fully used, and data cannot be saved. Please replace SD memory or format the memory before reconnecting. Available storage is less than 20%
0007	7	Folder Creation Failed	Failed to create folder on SD memory card (Please replace SD memory or format the memory before reconnecting.

(4) Analog error information

Error code		Error Description	Action
Hex	Dec		
0064	0100	Input Channel 0 range setting error	Set it to a configurable input range.
0065	0101	Input Channel 1 range setting error	Set it to a configurable input range.
00C8	0200	Input Channel 0 filter value setting error	Set it to a configurable filter value.
00C9	0201	Input Channel 1 filter value setting error	Set it to a configurable filter value.
012C	0300	Input Channel 0 average value setting error	Set it to a configurable average value.
012D	0301	Input Channel 1 average value setting error	Set it to a configurable average value.
0190	0400	Output Channel 0 range setting error	Set it to a configurable output range.
0191	0401	Output Channel 1 range setting error	Set it to a configurable output range.
01F4	0500	Output Channel 0 Input value setting error	Set it to a configurable input value.
01F5	0501	Output Channel 1 Input value setting error	Set it to a configurable input value.
0258	0600	Output Channel 0 Interpolation method setting error	Set it to a configurable Interpolation method range.
0259	0601	Output Channel 1 Interpolation method setting error	Set it to a configurable Interpolation method range.

(5) Cnet error information

1) XGT server error information

Error code		Error Description	Action
Hex	Dec		
0003	0003	Block number excess error	Set request block count to 16 or less
0004	0004	Variable length error	Set variable length to 16 or less
0007	0007	Data type error	Set the data type to X,B,W,D,L.
0011	0017	Data error	Set with configurable data type, area and length.
0090	0144	Monitor execution error	Please register for monitor execution.
0190	0400	Monitor execution error	When executing the monitor, set the registration number range requested by the client to the configurable input range.
0290	0656	Monitor registration error	When registering monitor, set the registration number range requested by the client to the configurable input range.
1132	4402	Device memory error	Set it to an available device.
1232	4658	Data size error	Set the data size to 60 words or less.
1234	4660	Spare frame error	Remove unnecessary content of slack frames.
1332	4914	Data type mismatch error	Remove unnecessary content of slack frames
1432	5170	Data value error	Check if the data value can be converted to Hex.
7132	28978	Variable request area excess error	Set the device to an available zone.

2) Modbus server error information

Error code		Error Description	Action
Hex	Dec		
0001	0001	Function code error	This is a function code that the server device does not support. Set it after checking whether the function code is supported by the Modbus server.
0002	0002	Address error	Set the address range supported by the server device.
0003	0003	Data setting error	Set the data type supported by the server device.
0004	0004	Server station abnormality error	Check the error status of the server (slave) station.
0005	0005	Server station retry request	The server is in a state where it cannot be processed due to a large amount of processing. Request redelivery when client-side processing time is available.
0006	0006	Server station processing time delay	It is a state that takes time for the server station to process. A re-request must be made on the client side.

3) P2P client error information

Error code		Error Description	Action
Hex	Dec		
0005	0005	P2P block Time out error	Check server connection status and media settings.

4) PLC CPU self error information

Error code		Error Description	Action
Hex	Dec		
0015	0021	P2P client timeout error	There is no response within 5 seconds after the PLC makes a P2P request to the communication module. Check the communication module status.
0016	0022	P2P client device error	An invalid device range was used. Please reset your device.

(5) Motion error information

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
0E00	3584	The command data range sent from XG5000 is out of the allowed value.	The current controller version may cause problems. Please check the supported version of XG5000 and controller.	Warning
0E01	3585	XG5000 test run function cannot be executed when the controller is in RUN status.	Please execute the XG5000 test operation after putting the controller in the STOP state.	Warning
0E02	3586	If there is an axis in operation, cam data cannot be written.	Write cam data while all axes are not in operation.	Warning
0E03	3587	If there is an axis in operation, encoder parameter cannot be written.	Write encoder parameter while all axes are not in operation.	Warning
0E04	3588	When EtherCAT communication is connected, EtherCAT parameters cannot be written.	After disconnecting EtherCAT communication, write network data.	Warning
0E10	3600	Encoder parameter data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
0E11	3601	Encoder 1 pulse input type of encoder parameter exceeded the setting range.	Set encoder 1 input signal of Encoder parameter to be between 0 and 5.	Alarm
0E12	3602	Encoder 1 maximum value of encoder parameter is out of pulse unit expression value.	For encoder 1 maximum value of encoder parameter, input 1 or more based on pulse unit.	Alarm
0E13	3603	Encoder 1 minimum value of encoder parameter is out of pulse unit expression value.	For encoder 1 minimum value of encoder parameter, input 1 or more based on pulse unit.	Alarm
0E14	3604	Encoder 1 pulse input Max/Min of encoder parameter exceeded the setting range.	Set the minimum value of encoder 1 of encoder parameter to be smaller than the maximum value.	Alarm
0E15	3605	Encoder 2 pulse input type of encoder parameter exceeded the range.	Set encoder 2 input signal of Encoder parameter to be between 0 and 5.	Alarm
0E16	3606	Encoder 2 maximum value of encoder parameter is out of pulse unit expression value.	For encoder 2 maximum value of encoder parameter, input 1 or more based on pulse unit.	Alarm
0E17	3607	Encoder 2 minimum value of encoder parameter is out of pulse unit expression value.	For encoder 2 minimum value of encoder parameter, input 1 or more based on pulse unit.	Alarm
0E18	3608	Encoder 2 pulse input Max/Min of encoder parameter exceeded the setting range.	Set the minimum value of encoder 2 in the encoder parameter to be smaller than the maximum value.	Alarm
0E19	3609	It is not possible to set the encoder input more than the encoder setting of the encoder parameter.	Check the encoder related items in the encoder parameter and set it to a value within the range.	Alarm
0E1A	3610	Encoder 1 pulse count per rotation in the encoder parameter exceeded the setting range.	Set pulse count per rotation of encoder1 in the encoder parameter to be greater than 0 and less than or equal to 4294967295.	Alarm
0E1B	3611	Encoder 1 travel distance per rotation in the encoder parameter exceeded the setting range.	Set travel distance per rotation of encoder1 in the encoder parameter to be greater than 0.000000001 and less than or equal to 4294967295.	Alarm
0E1C	3612	Encoder 2 pulse count per rotation in the encoder parameter exceeded the setting range.	Set pulse count per rotation of encoder2 in the encoder parameter to be greater than 0 and less than or equal to 4294967295.	Alarm
0E1D	3613	Encoder 2 travel distance per rotation in the encoder parameter exceeded the setting range.	Set travel distance per rotation of encoder2 in the encoder parameter to be greater than 0.000000001 and less than or equal to 4294967295.	Alarm
0E1E	3614	Encoder 1 input filter value in the encoder parameter exceeded the setting range.	Set encoder 1 input signal in the encoder parameter to be between 0 and 6.	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
0E1F	3615	Encoder 2 input filter value in the encoder parameter exceeded the setting range.	Set encoder 2 input signal in the encoder parameter to be between 0 and 6.	Alarm
0E20	3616	Encoder 1 maximum and minimum values of encoder parameters are set not to include the current position of encoder 1.	Encoder 1 maximum and minimum values of encoder parameters are set to include the current position of encoder 1. Or to operate with the set parameters, use the encoder preset command to change the encoder current position to a value within the parameter range.	Alarm
0E21	3617	Encoder 2 maximum and minimum values of encoder parameters are set not to include the current position of encoder 2.	Encoder 2 maximum and minimum values of encoder parameters are set to include the current position User. Or to operate with the set parameters, use the encoder preset command to change the encoder current position to a value within the parameter range.	Alarm
0E22	3618	Encoder 1 position latch value in the encoder parameter exceeded the setting range.	Set encoder 1 position latch in the encoder parameter to be between 0 and 1.	Alarm
0E23	3619	Encoder 2 position latch value in the encoder parameter exceeded the setting range.	Set encoder 2 position latch in the encoder parameter to be between 0 and 1.	Alarm
0E30	3632	EtherCAT parameter data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
0E31	3633	The number of constant period communication timeouts of the EtherCAT parameter exceeded the range.	Set the constant period communication timeout number of the EtherCAT parameter to a value between 1 and 8.	Alarm
0E32	3634	An error occurred while parsing EtherCAT parameters.	Check the EtherCAT parameters, and set again.	Alarm
0E40	3648	Connection command cannot be executed with EtherCAT parameter error.	Check the EtherCAT parameters, and set again.	Warning
0E41	3649	Network connection command is running.	Check that the network connection command is not entered again while the network connection command is being executed.	Warning
0E42	3650	A network disconnect command is running.	Check that the network disconnection command is not entered again while the network disconnection command is being executed.	Warning
0E43	3651	The connect/disconnect command cannot be executed with mode switching.	Check that no mode switching is executed during network connect/disconnect command operation.	Warning
0E44	3652	The connect/disconnect command cannot be executed with ESTOP Command.	Check that no ESTOP Command is executed during network connect/disconnect command operation.	Warning
0E50	3664	Unable to execute encoder preset command due to encoder parameter error.	Check the encoder related items of the encoder parameter to see if it is set to a value within the range, and set the encoder parameter to a normal value using XG5000.	Warning
0E51	3665	Preset command cannot be done because of the axis which using relevant encoder as a main axis	When there is an axis that operates the corresponding encoder as the main axis, check if the encoder preset command is input to the corresponding encoder.	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
0E52	3666	The encoder preset position is out of the range of the maximum or minimum value of relevant encoder.	Set the encoder preset position to a range greater than or equal to the minimum value of relevant encoder and less than or equal to the maximum value.	Warning
0E53	3667	The encoder selection of the encoder preset command exceeded the range.	For encoder selection, set a value between 0 and 1 (0: Encoder 1, 1: Encoder 2).	Warning
0E54	3668	The command to set encoder current position cannot be executed because the built-in ENC1 is set as the encoder of the spindle axis automatically controlled by the NC function module.	Check that the built-in ENC1 is not set as the encoder connected to the spindle axis in the 'Spindle encoder selection' item of the axis parameters.	Warning
0E55	3669	The command a set encoder current position cannot be executed because the built-in ENC2 is set as the encoder of the spindle axis automatically controlled by the NC function module.	Check that the built-in ENC2 is not set as the encoder connected the spindle axis in the 'Spindle encoder selection' item of the axis parameters.	Warning
0E60	3680	A command cannot be executed with basic parameter data error.	Download the basic parameters from XG5000 again, and if it occurs repeatedly basic parameter error after re-executing, request A/S.	Alarm
0E61	3681	CAM data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
0E80	3712	LS_SaveMotionData command is running.	Please execute it again after the previously executed LS_SaveMotionData command is completed.	Alarm
0E81	3713	An error occurred while executing the LS_SaveMotionData command.	Execute the LS_SaveMotionData command again in XG5000, and if it occurs repeatedly after re-executing, request A/S.	Alarm
0E82	3714	Exceeded the mode range of the LS_SaveMotionData command.	Set Mode to 0 or 1 (0: save only changed motion data, 1: save all motion data)	Alarm
0E90	3728	Error executing all axis command MC_PowerAll.	Check if the axis is in a state where the command can be executed and execute the command again.	Warning
0E91	3729	Error executing all axis command MC_HomeAll.	Check if the axis is in a state where the command can be executed and execute the command again.	Warning
0E92	3730	Error executing all axis command LS_HomeAll.	Check if the axis is in a state where the command can be executed and execute the command again.	Warning
0E93	3731	Error executing all axis command MC_StopAll.	Check if the axis is in a state where the command can be executed and execute the command again.	Warning
0E94	3732	Error executing all axis command MC_HaltAll.	Check if the axis is in a state where the command can be executed and execute the command again.	Warning
0E95	3733	Error executing all axis command MC_Reset2All.	Check if the axis is in a state where the command can be executed and execute the command again.	Warning
0E96	3734	Error executing all axis command MC_SetPositionAll.	Check if the axis is in a state where the command can be executed and execute the command again.	Warning
0F00	3840	Failed to change to EtherCAT INIT state.	Check the communication cable connection status and slave operation status (power on and error occurrence). Make sure communication cables are not exposed to noise.	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
0F06	3846	EtherCAT INIT status initialization (DC_INIT) error.	Check the communication cable connection status and slave operation status (power on and error occurrence). Make sure communication cables are not exposed to noise.	Alarm
0F09	3849	There is no EtherCAT slave connected to the controller.	If there is a slave connected to the controller, check that the communication cable between the controller and the EtherCAT slave is properly installed, that the power is normally supplied to the EtherCAT slave, or that the communication cable is not exposed to noise.	Alarm
0F0A	3850	The maximum number of connected slaves has been exceeded.	Check that there are not exceed 64 EtherCAT slaves connected to the controller	Alarm
0F0E	3854	There is a difference in node ID and network settings.	Check that the network cable connection sequence matches the network settings.	Alarm
0F0F	3855	There is wrong with the node ID setting.	Check if the node ID is duplicated or there is an error in the setting.	Alarm
0F10	3856	Failed to change to EtherCAT PREOP state.	Check the communication cable connection status and slave operation status (power on and error occurrence). Make sure communication cables are not exposed to noise.	Alarm
0F1E	3870	No network setting data.	Send slave parameters to the controller using XG5000.	Alarm
0F1F	3871	The network setting data and the connected slave are different.	Transmit the slave parameters after connecting the slave using the network slave automatic connection using XG5000.	Alarm
0F20	3872	Failed to change to EtherCAT SAFEOP state.	Check the communication cable connection status and slave operation status (power on and error occurrence). Make sure communication cables are not exposed to noise.	Alarm
0F30	3888	Failed to change to EtherCAT OP state.	Check the communication cable connection status and slave operation status (power on and error occurrence). Make sure communication cables are not exposed to noise.	Alarm
0F40	3904	Failed to change EtherCAT OP state to INIT state.	Check the communication cable connection status and slave operation status (power on and error occurrence). Make sure communication cables are not exposed to noise.	Alarm
0F50	3920	There is no response in the communication connection state.	Check the communication cable connection status and slave operation status (power on and error occurrence). Make sure communication cables are not exposed to noise.	Alarm
0F51	3921	A constant cycle communication error has occurred. (A communication error exceeding the number of master parameter constant cycle communication timeouts occurred.)	Check that the servo power is not turned off during communication, the communication cable is installed normally, or the communication cable is not exposed to noise.	Alarm
0F52	3922	A constant cycle communication error has occurred. (AlStatus error occurred in slave)	Please check the AL Status Code of the slave.	Alarm

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
0F60	3936	The slave device address (Adp) setting value of the ESC read command exceeded the range.	Check and set the slave device address (Adp) range according to the EtherCAT command code (EcatCmd) setting value.	Warning
0F61	3937	The data size setting value of the ESC read command exceeded the range.	Set the data size setting value of the ESC read command to 1 to 4 (BYTE)	Warning
0F62	3938	The EtherCAT command code (EcatCmd) setting value of the ESC read command is incorrect.	Set the EtherCAT command code to one of 1 (APRD), 4 (FPRD), or 7 (BRD).	Warning
0F63	3939	There is no response from the slave device to the ESC read command.	Check whether the slave device designated as Adp is installed normally or if the Ado address value is in the read-allowed area.	Warning
0F64	3940	ESC read command cannot be executed when the slave is in Init state.	Change the status of the slave to PreOP, SafeOP, or OP, then execute again.	Warning
0F70	3952	The slave device address (Adp) setting value of the ESC write command exceeded the range.	Check and set the slave device address (Adp) range according to the EtherCAT command code (EcatCmd) setting value.	Warning
0F71	3953	The data size setting value of the ESC write command exceeded the range.	Set the data size setting value of the ESC write command to 1 to 4 (BYTE)	Warning
0F72	3954	The EtherCAT command code (EcatCmd) setting value of the ESC write command is incorrect.	Set the EtherCAT command code to one of 2 (APWR), 5 (FPWR), or 8 (BWR).	Warning
0F73	3955	There is no response from the slave device to the ESC write command.	Check whether the slave device designated as Adp is installed normally or if the Ado address value is in the write allowed area.	Warning
0F74	3956	The area designated by the ESC address (Ado) cannot be written while the communication connection/disconnection command is being executed or in the communication connection state.	In the ESC write command, set the writable ESC address (Ado) during communication connection/disconnection command execution or communication connection status.	Warning
0F75	3957	ESC write command cannot be executed when the slave is in Init state.	Change the status of the slave to PreOP, SafeOP, or OP, then execute again.	Warning
0FF2	4082	Normal operation related to encoder input cannot be executed due to the controller H/W error.	If it occurs repeatedly during retry, please request A/S.	Alarm
1000	4096	Axes are not ready to operating (No is not connected.)	Execute the command when the axis is ready for operation.	Warning
1001	4097	Cannot run in "Disabled" state.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
1002	4098	Cannot run in "Standstill" state.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
1003	4099	Cannot run in "Discrete" state.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
1004	4100	Cannot run in "Continuous" state.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
1005	4101	Cannot run in "Synchronized" state.	Check the operable axis status of the command, and	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
			execute the command in the status where the command can be operated.	
1006	4102	Cannot run in "Homing" state.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
1007	4103	Cannot run in "stopped" state.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
1008	4104	Cannot run in "ErrorStop" state.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
100A	4106	Motion command cannot be executed when the affiliated axis group is active.	Execute the command after changing the axis group to GroupDisabled with the axis group disable command.	Warning
100B	4107	This command cannot be given to the virtual axis.	This command cannot be executed on the virtual axis. Check that the command is not executed on the virtual axis.	Warning
100C	4108	If it is registered as NC channel/axis and NC control operation is in progress, the corresponding command cannot be executed.	Check the operable axis status of the command, and execute the command in the status where the command can be operated.	Warning
100D	4109	The command cannot be executed because the axis is not active.	Check if the setting axis of the relevant command is registered in the axis parameter. Axis can be registered in axis parameter among motion data items of XG5000.	Warning
100E	4110	While the motion test operation command is being executed, it is changed to 'Run' and cannot continue operation.	Check that the controller has not changed to 'Run' state while the axis is running.	Warning
100F	4111	The controller is stopped by the ESTOP command and the axis operation cannot be continued.	Check that the controller is not stopped by ESTOP command during axis operation.	Alarm
1010	4112	The controller has changed to 'Stop' or 'Error' status and operation cannot be continued.	Check that the controller has not changed to 'Stop' or 'Error' state while the axis is running.	Alarm
1011	4113	Since the network connection was disconnected, operation cannot be continued.	Check if the network connection is not disconnected due to slave power error, network cable error, or noise inflow on the network cable while the axis is running.	Alarm
1012	4114	The absolute target position where the position setting value of the command is reflected is out of the pulse unit position expression range.	Set the command position so that it does not exceed the 32-bit INT range (-2147483648 ~ 2147483647) when the absolute target position value reflecting the position setting value of the command is converted into pulse units. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Warning
1013	4115	The operating speed value is less than 0 or exceeds the maximum speed value.	Set the operation speed value greater than 0 and less than the maximum speed value set for the axis.	Warning
1014	4116	Acceleration is set to negative.	Set the acceleration value to a value greater than or equal to 0.	Warning
1015	4117	Deceleration is set to negative.	Set the deceleration value to a value greater than or	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
			equal to 0.	
1016	4118	Jerk is set to negative.	Set the Jerk value to a value greater than or equal to 0.	Warning
1017	4119	The direction setting value is out of range.	Check the range of the direction setting value of the command, and set it within the range. (Refer to Chapter 6 Commands and Functions)	Warning
1018	4120	The torque setting value is out of range.	Set the torque setting value within 1000%.	Warning
1019	4121	The torque ramp setting value is out of range.	Set torque ramp setting value to a value greater than or equal to 0.	Warning
101A	4122	The Buffer Mode setting value is out of input range.	Set a settable value (0 to 5) in the buffer mode.	Warning
101B	4123	The Execution Mode setting value is out of input range.	Set a settable value (0 to 1) in the Execution Mode.	Warning
101C	4124	A tracking error excess alarm occurs, and operation cannot be continued.	The deviation between the command position and the current position is out of the 'Exceeding value of tracking error'. In order not to occur an alarm, tune the servo drive or set the 'Exceeding value of tracking error' larger.	Alarm
101D	4125	Exceeding value of tracking error warning occurred.	The deviation between the command position and the current position is out of the 'Exceeding value of tracking error'. In order not to occur an alarm, tune the servo drive or set the 'Exceeding value of tracking error' larger.	Warning
101F	4127	The command position value transmitted to the servo drive is out of the pulse unit expression value.	When the command position value was converted into pulse units, it was out of the 32-bit area When the command position values is converted to a pulse, it set within the range of -2147483648 to 2147483647) (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
1020	4128	Undefined axis command	The command is not executed in the current version of the controller. Check the version in which the motion function block can be executed.	Warning
1021	4129	The same command was executed, thus canceling the previously executed command.	The corresponding motion command can only be executed once per scan. Change the operating condition of the program so that one motion command is executed per scan.	Warning
1022	4130	The number of Buffered commands that can be executed has been exceeded.	The command cannot be executed because the command buffer of the corresponding axis is full. The number of Buffered Command that can be executed is 100. Adjust the command execution timing.	Warning
1023	4131	The value of the parameter exceeded the input range.	Check the parameter input range and execute the command.	Warning
1030	4144	Axis parameters cannot be written while the axis is running.	Write parameters when the relevant axis is not in operation.	Warning
1040	4160	Axis parameter data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
1041	4161	Operation cannot be executed due to the axis	Check the Axis parameters, and set again.	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		parameters error.		
1042	4162	The speed limit of the axis parameter cannot be set to a value less than 0.	Set the speed limit of the basic parameter to a value greater than zero.	Warning
1043	4163	Axis parameter software upper/lower value exceeds range.	Set the soft upper limit of the axis parameter to be greater than or equal to the soft lower limit.	Warning
1044	4164	The current speed filter time constant value of the axis parameter exceeds the range.	Set the parameter setting value from 0 to 100.	Warning
1045	4165	Error reset monitoring time of axis parameter exceeded the range.	Set the parameter setting value from 1 to 1000.	Warning
1046	4166	The set value of transfer distance per rotation exceeds the range	Set the parameter setting value to 0.000000001 or more and 4294967295 or less.	Warning
1047	4167	Infinite length repeat position setting value exceeded the range.	Set the parameter setting value greater than 0 and less than or equal to 2147483647 in pulse units.	Warning
1048	4168	The in-position width setting value is out of range.	Set the parameter setting value to be greater than or equal to 0 and less than or equal to 2147483647 in pulse units.	Warning
1049	4169	The set value of Tracking error over range exceeds the range	Set the parameter setting value to be greater than or equal to 0 and less than or equal to 2147483647 in pulse units.	Warning
104A	4170	The current position display compensation amount setting value exceeds the range.	Set the parameter setting value to be greater than or equal to 0 and less than or equal to 2147483647 in pulse units.	Warning
104B	4171	The set value of JOG high speed speed exceeds the range	Set the parameter setting value greater than 0, greater than the jog low speed value, and less than the speed limit value.	Warning
104C	4172	The set value of jog low speed speed exceeds the range	Set the parameter setting value greater than 0, less than the jog high speed speed value, and less than the speed limit value.	Warning
104D	4173	The set value of jog acceleration exceeds the range	Set the parameter setting value to 0 or higher.	Warning
104E	4174	The set value of jog deceleration exceeds the range	Set the parameter setting value to 0 or higher.	Warning
104F	4175	The set value of jog jerk exceeds the range	Set the parameter setting value to 0 or higher.	Warning
1050	4176	The set value of gear ratio of motor side exceeds the range	Set the parameter setting value from 1 to 65535.	Warning
1051	4177	The set value of gear ratio of mechanical side exceeds the range	Set the parameter setting value from 1 to 65535.	Warning
1052	4178	The set value of pulse per rotation exceeds the range	Set the parameter setting value greater than 0 and less than or equal to 4294967295.	Warning
1053	4179	The set value of connection device exceeds the range	Set the device number of the slave that can be supported. The node setting range is 0 (no connection device), 1 to 64.	Warning
1054	4180	The set value of axis type exceeds the range	Set the parameter setting value to '0: real axis' or '1: virtual axis'.	Warning
1055	4181	The set value of speed command unit exceeds the range	Set the parameter setting value to '0: unit /sec', '1: unit /min', '2: rpm'.	Warning
1056	4182	The set value of blacklash compenstion amount	Set the parameter setting value to be greater than or	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		exceeds the range.	equal to 0 and less than or equal to 65535 in pulse units.	
1060	4192	Servo On cannot be executed due to a servo drive error.	Check the error factor of the servo drive, remove the servo drive error, and then turn on the servo.	Alarm
1061	4193	Servo-on execution command was executed again during servo-on processing.	Check if the servo-on command is not executed again in the program or XG5000 during servo-on processing.	Warning
1062	4194	Servo-on could not be completed because the servo drive could not change to "ReadyToSwitchON" status.	Check the status of the servo drive. Servo-on command may not be executed under certain conditions.	Alarm
1063	4195	Servo on could not be completed because the servo drive could not be changed to "Switched on" status.	Check the status of the servo drive. Servo-on command may not be executed under certain conditions.	Alarm
1064	4196	Servo on could not be completed because the servo drive could not be changed to "operation enable" status.	Check the status of the servo drive. Servo-on command may not be executed under certain conditions.	Alarm
1065	4197	Servo-on cannot be completed because the "Quick Stop" function of the servo drive is activated.	Check the status of the servo drive. Servo-on command may not be executed under certain conditions.	Alarm
1066	4198	Servo off execution command was executed again during servo- Servo-Off processing.	Check if the servo off command is not executed again in the program or XG5000 during servo- Servo off processing.	Warning
1067	4199	Servo off execution command execution is not completed.	Check the the servo drive status.	Alarm
1070	4208	The servo error reset monitoring time has been exceeded.	The servo drive error was not removed after the error reset monitoring time set in the axis parameter passed. After removing the cause of the servo drive error, execute the error reset command again.	Warning
1080	4224	Commands using absolute coordinates cannot be executed in the absolute coordinates of the undecided homing state.	Execute the absolute coordinate operation command after making the homing state with the home return command or the current position setting command.	Warning
1081	4225	In the state where infinite running repeat is allowed, the target position is out of the infinite running repeat position range in the direction designation.	Set the target position within the infinite running repeat position in the direction designation	Warning
1082	4226	SuperImposed commands cannot be executed while operating with speed control or torque control.	Execute the SuperImposed command while not operating with speed control or torque control.	Warning
1083	4227	A SuperImposed stop command cannot be executed while not operating a SuperImposed.	Execute the SuperImposed stop operation command while operating SuperImposed.	Warning
1090	4240	The position value of the current position change command exceeded the range.	Execute the current position preset command after setting the position setting value above the soft lower limit value and below the soft upper limit value of the extended parameter.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
1091	4241	The current position change command cannot be executed during operation by homing, speed synchronization, cam, or torque control.	Execute the current position change command when the relevant axis is not in operation of one of homing, speed synchronization, cam, and torque control.	Warning
1092	4242	When the spindle axis is automatically controlled by the NC function module, if the 'Spindle encoder selection' item of the axis parameter is '0: Disabled', the axis current position setting command cannot be executed.	After correctly setting the encoder connection method connected to the spindle axis in the 'Spindle encoder selection' item of the axis parameter, execute the axis current position setting command.	Warning
1093	4243	If the 'Spindle encoder selection' item of the axis parameter is '1: Motor ENC', the 'Position actual value (0x6064)' object does not exist in the TxPDO setting of the EtherCAT slave connected to the spindle axis, so the axis current position setting command cannot be executed.	If the 'Spindle encoder selection' item of the axis parameter is '1: Motor ENC', add the 'Position actual value (0x6064)' object to the TxPDO setting of the EtherCAT slave connected to the spindle axis, resume EtherCAT connection, and then execute the axis current position setting command.	Warning
1094	4244	If the 'spindle encoder selection' item of the axis parameter is '2: Built-in ENC1', the axis current position setting command cannot be executed because the encoder 1 parameter setting is incorrect.	If the 'Spindle encoder selection' item of the axis parameter is '2: Built-in ENC1', set the encoder 1 unit = pulse, Encoder 1 maximum value = 2147483647, Encoder 1 minimum value = -2147483648 and then execute the axis current position setting command. Please run.	Warning
1095	4245	If the 'spindle encoder selection' item of the axis parameter is '3: Built-in ENC2', the axis current position setting command cannot be executed because the encoder 2 parameter setting is incorrect.	If the 'Spindle encoder selection' item of the axis parameter is '3: Built-in ENC2', set the encoder 2 unit = pulse, Encoder 2 maximum value = 2147483647, Encoder 2 minimum value = -2147483648 and then execute the axis current position setting command. Please run.	Warning
10A0	4256	Servo drive does not support torque control mode.	Execute torque control using a servo drive that supports CST mode of EtherCAT CoE.	Warning
10A1	4257	There is no target torque object (0x6071) setting that can execute torque control in the RxPDO entry setting in the slave parameter.	In the slave parameter of XG5000, set the target torque object (0x6071) that supports torque control in the RxPDO entry and send it to the controller.	Warning
10B0	4272	Servo drive does not support homing mode.	Execute homing using a servo drive that supports is 'homing mode of EtherCAT CoE.	Warning
10B1	4273	An error occurred during homing in the servo drive.	Check the error factor of the servo drive, remove the servo drive error, and then execute homing.	Warning
10B2	4274	The homing command cannot be executed while the axis is running.	After the axis stops operating, execute the homing command again in the Standstill state.	Warning
10B3	4275	The switch search speed value of the extended homing command exceeded the range.	When converting the switch search speed value to pulse unit, set it to a value between 0 and 1073741824.	Warning
10B4	4276	The ZERO search speed value of the extended homing command exceeded the range.	When converting the ZERO search speed value to pulse unit, set it to a value between 0 and 1073741824.	Warning
10B5	4277	The Homing acceleration value of the extended homing command exceeded the range.	When converting homing acceleration value to pulse unit, set it to a value between 0 and 1073741824.	Warning
10B6	4278	The Home offset value of the extended homing command exceeded the range.	When converting home offset value to pulse unit, set it to a value between -2147483648 and 2147483647.	Warning

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
10B7	4279	Failed to write extended homing command parameters to the drive.	Among the parameters of the extended homing command, there is an item that is not supported by the drive. (Refer to the user manual of the drive) Use the basic homing command (MC_Home).	Warning
10C0	4288	Override command cannot be executed if it is not operating with position/speed control.	Execute the override command while operating by position control or speed control.	Warning
10C1	4289	The override factor of the override instruction exceeded the range.	Set the VelFactor, AccFactor, and JerkFactor values of the override command to 0 or more and execute the override command.	Warning
10C2	4290	After reflecting factor of the override command, the operation speed value exceeded the maximum speed value.	Override within the range that does not exceed the maximum speed value of the relevant axis.	Warning
10D0	4304	Gear ratio denominator od gear operation cannot be zero.	Set the gear ratio denominator to a non-zero value and execute the command.	Warning
10D1	4305	Gear operation MasterValueSource setting value exceeds the range.	Set the MasterValueSource input values to 0~1 and execute the command.	Warning
10D2	4306	Gear operation main axis setting is out of range.	Set axis and encoder numbers with a range by product.	Warning
10D3	4307	The gear operation main axis setting is the same as that of the sub axis.	Set the main axis as an axis different from the sub axis (command axis) and execute the command.	Warning
10D4	4308	Gear operation main axis are not ready to operating	Execute the command when the main axis is ready for operation.	Warning
10D5	4309	If the gear operation main axis is set as an encoder, the command cannot be executed due to the occurrence of a common parameter error.	Check the encoder related items of the encoder parameter to see if it is set to a value within the range, and set the encoder parameter to a normal value using XG5000.	Warning
10D6	4310	MC_GearInPos command cannot be executed when the main axis is operating under torque control.	Execute the MC_GearInPos command while the main axis is not operating under torque control.	Warning
10D7	4311	The speed of the gear operation sub-axis has exceeded the speed limit.	Reduce the speed of the main axis or set a different gear ratio so that the speed of the sub-axis during gear operation does not exceed the speed limit set for the sub-axis.	Warning
10D8	4312	If it is not in gear operation, the gear release command cannot be executed.	The gear release command can only be used during gear operation.	Warning
10D9	4313	The command cannot be executed because the target speed setting value of the positioning gear operation command is smaller than the current operation speed or gear operation speed.	Execute the command after setting the target speed setting value of the positioning gear operation command to be higher than the current operation speed or gear operation speed.	Alarm
10DA	4314	During the positioning gear operation operation, the subordinate axis cannot reach the subordinate axis synchronous position at the set target speed within the time the main axis operates to the main axis synchronous position.	Increase the target speed setting value of the positioning gear operation command or execute the command after adjusting MasterStartDistance so that the subordinate axis can move to the subordinate axis synchronous position within the time that the main axis operates to the main axis synchronous position.	Alarm
10DB	4315	Synchronous operation commands (gear, cam, etc.) cannot be executed when the main axis is in	Execute the synchronous operation command (gear, cam, etc.) while the main axis is not in homing	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		homing operation.	operation.	
10E0	4320	There is no object setting that can execute the touch probe in the PDO entry setting in the slave parameter.	After setting the object that supports touch probe in the PDO entry in the slave parameter of XG5000, transmit it to the controller.	Warning
10E1	4321	TriggerInput input setting value is out of range.	Please set the TriggerInput setting value to 0 (Touch Probe1) or 1 (Touch Probe2).	Warning
10E2	4322	TriggerInput input setting value of expansion touch probe command is out of range.	Set the TriggerInput setting value of the expansion touch probe command to a value between 0 (Rising edge of TouchProbe1) and 5 (Index pulse of TouchProbe2).	Warning
10E3	4323	TriggerMode input setting value of expansion touch probe command is out of range.	Set the TriggerMode setting value of the expansion touch probe command to 0 (single trigger) or 1 (continuous trigger).	Warning
10F0	4336	Parameter number setting value of parameter read/write command is out of range.	Execute the command after setting the parameter number setting value of the parameter read/write command between 0~28, 100~108, and 200~208.	Warning
10F1	4337	The data set value of the parameter set in the parameter write command is out of range.	Check the data setting range of the parameter you want to set.	Warning
10F2	4338	When changing the corresponding encoder parameter, the parameter cannot be changed because the encoder 1 maximum value is out of the pulse unit expression value.	Change the relevant parameter in advance so that no error occurs when converting the encoder 1 maximum value into pulse units.	Warning
10F3	4339	When changing the corresponding encoder parameter, the parameter cannot be changed because the encoder 1 minimum value is out of the pulse unit expression value.	Change the relevant parameter in advance so that no error occurs when converting the encoder 1 minimum value into pulse units.	Warning
10F4	4340	When changing the corresponding encoder parameter, the parameter cannot be changed because the encoder 2 maximum value is out of the pulse unit expression value.	Change the relevant parameter in advance so that no error occurs when converting the encoder 2 maximum value into pulse units.	Warning
10F5	4341	When changing the corresponding encoder parameter, the parameter cannot be changed because the encoder 2 minimum value is out of the pulse unit expression value.	Change the relevant parameter in advance so that no error occurs when converting the encoder 2 minimum value into pulse units.	Warning
1100	4352	Jog operation command cannot be executed while the axis is in operation.	Execute the jog command while the axis is stopped.	Warning
1101	4353	If the tool retract command is being executed in the NC channel, the jog operation command cannot be executed in more than one axis at the same time.	When canceling the tool retract command of the NC channel or using the tool retract command, execute the jog operation command on only one axis at a time.	Warning
1110	4368	There is an error in the cam operation MasterScaling input value.	A value of 0 cannot be entered in the MasterScaling input value.	Warning
1111	4369	There is an error in the cam operation MasterStartDistance input value.	Set the MasterStartDistance input values to a value greater than or equal to 0 and execute the command.	Warning
1112	4370	There is an error in the cam operation MasterSyncPosition input value.	Set the MasterSyncPosition input values to a value greater than or equal to 0 and execute the command.	Warning

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
1113	4371	Cam operation StartMode input value exceeded the range.	Set the StartMode input values to 0~1 and execute the command.	Warning
1114	4372	Cam operation MasterValueSource input value exceeded the range.	Set the MasterValueSource input values to 0~1 and execute the command.	Warning
1115	4373	The specified cam table does not exist.	Adjust the cam table number to a valid cam table number and execute the command.	Warning
1116	4374	Cam operation main axis setting is out of range.	Set axis and encoder numbers with a range by product. If the main axis is a variable, check that the VarOffset value does not exceed the memory area.	Warning
1117	4375	The cam operation main axis setting is the same as that of the sub axis.	Set the main axis as an axis different from the sub axis (command axis) and execute the command.	Warning
1118	4376	Cam operation main axis are not ready to operating	Execute the command when the main axis is ready for operation.	Warning
1119	4377	If the cam operation main axis is set as an encoder, the command cannot be executed due to the occurrence of a common parameter error.	Check the encoder related items of the encoder parameter to see if it is set to a value within the range, and set the encoder parameter to a normal value using XG5000.	Warning
111A	4378	The speed of the cam operation sub-axis has exceeded the speed limit.	Reduce the speed of the main axis or adjust cam table so that the speed of the sub-axis during cam operation does not exceed the speed limit set for the sub-axis.	Warning
111B	4379	If it is not in cam operation, the cam release command cannot be executed.	The cam release command can only be used during cam operation.	Warning
111C	4380	The cam data count setting value of the cam data write command exceeded the range.	Set the cam data number setting value of write cam data command between 1 and 100.	Warning
111D	4381	The specified cam table data of the cam data read command is abnormal.	Set the cam data again, and if it occurs repeatedly after re-executing, request A/S.	Warning
111E	4382	CAM skip command cannot be executed when it is not CAM operation.	Execute the CAM skip command when operating CAM.	Warning
111F	4383	The number of cam cycles to skip in the cam skip command is set to 0.	Set the number of CAM cycles to skip in the CAM skip command to be greater than 0.	Warning
1121	4385	The skip mode setting value of the cam skip command exceeded the range.	Set the skip mode setting value of the cam skip command to a value between 0 and 2 and execute the command.	Warning
1122	4386	Cam table not registered.	Register the cam table or reset the data and execute the command.	Warning
1123	4387	The cam data of the cam data write command is abnormal.	Set the data of the cam data write command correctly.	Warning
1124	4388	Cam main axis value does not exist within the specified range	Check the MasterStartPos and MasterEndPos values, and execute the command again.	Warning
1130	4400	The phase compensation command cannot be executed if the command axis is not in InSync or InGear state of synchronous control (cam, gear operation) operation.	If the command axis of the phase compensation command is in synchronous control operation, execute the phase compensation command in InSync or InGear status.	Warning
1131	4401	There is an error in the main axis setting of the phase compensation command.	Execute the command after setting the main axis setting of the phase compensation command to be the same as the actual main axis of the current	Warning

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
			synchronous operation.	
1132	4402	The phase compensation amount of the phase compensation command is out of the position expression range.	Execute the command after setting the phase compensation amount so that it falls within the range of -2147483648 to 2147483647 or less in pulse units.	Warning
1133	4403	The speed setting values of the phase compensation command is out of the range.	Execute the command after setting the speed value of the phase compensation command to be greater than 0 and less than the speed limit of the main axis.	Warning
1140	4416	The connected slave device does not support speed control mode.	Use a slave device that supports the Velocity mode of EtherCAT CoE to execute speed control.	Warning
1150	4432	The connected slave device does not support position control mode.	Use a slave device that supports the CSP mode of EtherCAT CoE to execute position control.	Warning
1151	4433	The operation mode of the connected slave device (0x6060, Mode Of Operation) SDO read service failed.	Check if the slave is in a state that can execute SDO service. If an error occurs repeatedly by re-executing the command, reconnect EtherCAT with "EtherCAT Disconnect – Connect Command" before use.	Warning
1152	4434	Operation mode (0x6060, Mode Of Operation) of the connected slave device Timeout for SDO read request occurred.	Check if the slave is in a state that can execute SDO service. If an error occurs repeatedly by re-executing the command, reconnect EtherCAT with "EtherCAT Disconnect – Connect Command" before use.	Warning
1160	4448	The connected slave device does not support synchronous speed control (CSV) mode.	Use a slave device that supports the synchronous speed control (CSV) mode of EtherCAT CoE to execute speed control.	Warning
1161	4449	There is no target velocity object (0x60FF) setting that can execute synchronous speed control CSV of the RxPDO entry setting in the slave parameter.	After setting the target speed object (0x60FF) supporting synchronous speed control (CSV) in the RxPDO entry of EtherCAT parameter slave data in XG5000, transmit it to the controller.	Warning
1162	4450	The CmdPosMode setting value of synchronous speed control (CSV) operation exceeds the input range.	CmdPosMode only supports a value of 0 (applies current position as command position). Execute the command again after setting CmdPosMode to 0.	Warning
1170	4464	Part length value is not valid	Part length value cannot be set less than 0.	Warning
1171	4465	The perimeter value is not valid.	The perimeter cannot be set to a value less than 0.	Warning
1172	4466	The cutting start position value is not valid.	The cutting start position cannot be set to a value smaller than 1/4 of the circumference value.	Warning
1173	4467	The cutting end position value is not valid.	The cutting end position cannot be set smaller than the cutting start position or larger than 3/4 of the perimeter.	Warning
1174	4468	The synchronous speed ratio value is not valid.	The value of the speed ratio of the cutting section must be set between 50 and 200.	Warning
1175	4469	The 0 speed section ratio value is not valid.	The value of 0 speed section must be set between 0 and 50.	Warning
1176	4470	The cam profile type value is not valid.	Change the camp profile type value and run the command.	Warning
1177	4471	The cam point count value is invalid.	Change the camp point count and run the command.	Warning
1178	4472	The cam curve type value is not valid.	Change the cam curve type value and run the command.	Warning
1179	4473	The cutting area is too wide.	Change the length of the cutting section or change the speed ratio value of the cutting section and execute	Warning

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
			the command.	
1200	4608	A hardware upper limit error has occurred.	Clear the error by executing the error reset command after out of the external upper limit signal range using the reverse jog command.	Alarm
1201	4609	A hardware lower limit error has occurred.	Clear the error by executing the error reset command after out of the external lower limit signal range using the reverse direction jog command.	Alarm
1203	4611	The command cannot be executed due to an error in the servo driver during operation.	After removing the cause of the servo error, clear the servo error with the error reset command.	Alarm
1204	4612	The command cannot be executed because the servo is off during operation.	Execute the command again after making the command axis into the servo-on state with the servo-on command.	Alarm
1205	4613	A software upper limit error has occurred.	Clear the error by executing the error reset command after out of the software upper limit range using the reverse jog command.	Alarm
1206	4614	A software lower limit error has occurred.	Clear the error by executing the error reset command after out of the software lower limit range using the forward direction jog command.	Alarm
1210	4624	Motion commands related to traversing cannot be executed if the spindle axis is automatically controlled by the NC function module.	After checking the motion commands that can be executed for the axis assigned as the NC spindle axis, execute the motion commands allowed for the NC spindle axis.	Warning
1220	4640	Parameter number setting value of motion information read/write command is out of range.	Execute the command after setting the parameter number value within of the motion information read command between 0 and 5.	Warning
1230	4656	Master position loop control cannot be enabled/disabled based operation.	Execute/release master position loop control when the corresponding axis is not in operation.	Warning
1231	4657	The P gain input value of the master position loop control function block is negative.	Enter a value of 0 or more in the P gain of the master position loop control function block.	Warning
1232	4658	The I gain input value of the master position loop control function block is negative.	Enter a value of 0 or more in I gain of the master position loop control function block.	Warning
1233	4659	The control output limit value of the master position loop control function block is negative.	Enter a value of 0 or more in the control output limit value of the master position loop control function block.	Warning
1234	4660	Cross- coupled control cannot be enabled/disabled during operation.	Execute/release Cross-coupled control when the corresponding axis is not in operation.	Warning
1235	4661	The command axis and the connection axis of the cross-coupled control function block must be different axes.	Set different axes for the command axis and the connection axis of the cross-coupled control function block.	Warning
1236	4662	In order to operate cross-coupled control, the master position loop controller must be running.	In order to drive cross-coupled control, execute master position loop control for the relevant axis first.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
1237	4663	The P gain input value of the Cross-coupled control function block is negative.	Enter a value of 0 or more in the P gain of the Cross-coupled control function block.	Warning
1238	4664	The command was re-executed on an axis for which cross-coupled control is not running.	When re-executing the cross-coupled control function block, set the command axis and connection axis to the previously executed values. The value of P gain can be changed without re-executing the function block.	Warning
1F00	7936	A constant cycle communication error has occurred. (A communication error exceeding the number of master parameter constant cycle communication timeouts occurred.)	Check that the servo power is not turned off during communication, the communication cable is installed normally, or the communication cable is not exposed to noise.	Alarm
1F10	7952	SDO command can no longer be executed due to previously executed slave device SDO processing failure	Please reset the connection after checking that the status of the slave device is normal	Warning
1F11	7953	SDO parameter write command cannot be executed while its operating.	Execute SDO parameter write command while the relevant axis is not in operation.	Warning
1F12	7954	The data range of the SDO parameter Index, SubIndex, etc. is out of the range.	SDO parameter Index is set to 0x0000~0x9FFF, SubIndex is 0x00 to 0xFF, and data size is set within 4 byte and read/write SDO parameter.	Alarm
1F13	7955	Abort occurred during SDO parameter write command.	Slave device stopped writing operation while writing SDO parameter. Check the write setting data and the status of the slave device.	Alarm
1F14	7956	There is no response from the slave device the SDO parameter write command.	Slave device no response while writing SDO parameter. Check the slave device status.	Alarm
1F16	7958	Abort occurred during SDO parameter EEPROM save.	Slave device canceled while saving SDO parameter EEPROM. Check the slave device status.	Alarm
1F17	7959	There is no response from the slave device the SDO parameter EEPROM save command.	Slave no response while saving SDO parameter EEPROM. Check the slave device status.	Alarm
1F19	7961	Other commands cannot be executed while writing SDO parameters or saving SDO parameters to EEPROM.	Execute another command after all SDO parameter EEPROM saving is completed.	Alarm
1F20	7968	Abort occurred during SDO parameter read command.	Slave device stopped reading operation while reading SDO parameter. Check the read setting data and the status of the slave device.	Alarm
1F21	7969	There is no response from the slave device the SDO parameter read command.	Slave device no response while reading SDO parameter. Check the slave device status.	Alarm
1F22	7970	SDO parameter read/write command cannot be executed while SDO parameter read/write command is being executed.	Execute the command after all currently executing SDO parameter read/write is completed.	Warning
1F33	7987	Failed to change the operation mode of the servo drive to position control (CSP) mode.	Check whether the corresponding servo drive supports EtherCAT CoE CSP mode, and check the status of the servo drive.	Alarm
1F34	7988	Failed to change the operation mode of the servo drive to home return (Homing) mode.	Check whether the corresponding servo drive supports EtherCAT CoE Homing mode, and check the status of the servo drive.	Alarm
1F35	7989	Failed to change the operation mode of the servo	Check whether the corresponding servo drive	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		drive to torque control (CST) mode.	supports EtherCAT CoE CST mode, and check the status of the servo drive.	
1F36	7990	Failed to change the operation mode of the servo drive to speed control (CST) mode.	Check whether the corresponding servo drive supports EtherCAT CoE CSV mode, and check the status of the servo drive.	Alarm
1F50	8016	XG5000 manual tuning function cannot be executed when the controller is in RUN status.	After changing the controller to STOP status, execute manual tuning of XG5000	Warning
1F60	8032	This command is not available in the slave's current EtherCAT state.	Set the slave's EtherCAT status to Boot status and execute the command.	Warning
1F61	8033	Send timeout occurred during file transfer	Check the status of the transfer line or slave and execute the command.	Warning
1F62	8034	Send timeout occurred during file transfer	Check the status of the transfer line or slave and execute the command.	Warning
1F63	8035	A packet error occurred during file transfer.	Check the status of the transfer line or slave and execute the command.	Warning
1F64	8036	Slave is out of memory.	Check the transferred file and execute the command.	Warning
1F65	8037	The device does not exist.	Execute the command after checking whether the FOE function is available as a slave.	Warning
1F66	8038	Access denied to slave.	Execute the command after checking whether the FOE function is available as a slave.	Warning
1F67	8039	Passwords do not match.	Confirm the password and execute the command.	Warning
1F68	8040	The data to be downloaded with the FoE function was not sent to the controller.	Check the communication cable connection status and control operation status.	Warning
1F6F	8047	Slave error occurred during file transfer	Remove the error of the slave and execute the command	Warning
2000	8192	The axis group is not ready for operation.	Execute the command while the axis group is ready for operation.	Warning
2001	8193	Axis group cannot be executed in "Disabled" state.	Check the operable axis group status of the command, and execute the command in the status where the command can be operated.	Warning
2002	8194	Axis group cannot be executed in "Standby" state.	Check the operable axis group status of the command, and execute the command in the status where the command can be operated.	Warning
2003	8195	Axis group cannot be executed in "Moving" state.	Check the operable axis group status of the command, and execute the command in the status where the command can be operated.	Warning
2004	8196	Axis group cannot be executed in "Homing" state.	Check the operable axis group status of the command, and execute the command in the status where the command can be operated.	Warning
2005	8197	Axis group cannot be executed in "Stopping" state.	Check the operable axis group status of the command, and execute the command in the status where the command can be operated.	Warning
2006	8198	Axis group cannot be executed in "Errorstop" state.	Check the operable axis group status of the command, and execute the command in the status where the command can be operated.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
2007	8199	The configuration axis of the axis group is not servo-on	Check the operable axis group status of the command, and execute the command in the status where the command can be operated.	Warning
200F	8207	The controller is stopped by the ESTOP command and the axis group operation cannot be continued.	Check that the controller is not stopped by ESTOP command during axis group operation.	Alarm
2010	8208	The controller has changed to 'Stop' or 'Error' status and operation cannot be continued.	Check that the controller has not changed to 'Stop' or 'Error' state while the axis group is running.	Alarm
2011	8209	Since the network connection was disconnected, operation cannot be continued.	Check if the network connection is not disconnected due to slave power error, network cable error, or noise inflow on the network cable while the axis group is running.	Alarm
2012	8210	The absolute target position where the position setting value of the command is reflected is out of the pulse unit position expression range.	Set the command position so that it does not exceed the 32-bit INT range (-2147483648 ~ 2147483647) when the absolute target position value reflecting the position setting value of the command is converted into pulse units. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Warning
2013	8211	The operating speed value is less than 0 or exceeds the maximum speed value.	Set the operation speed value greater than 0 and less than the maximum speed value set for the axis group.	Warning
2014	8212	Acceleration is set to negative.	Set the acceleration value to a value greater than or equal to 0.	Warning
2015	8213	Deceleration is set to negative.	Set the deceleration value to a value greater than or equal to 0.	Warning
2016	8214	Jerk is set to negative.	Set the Jerk value to a value greater than or equal to 0.	Warning
201A	8218	The Buffer Mode setting value is out of range.	Set a settable value in the buffer mode.	Warning
201B	8219	The Execution Mode setting value is out of input range.	Set a settable value (0 to 1) in the Execution Mode.	Warning
201C	8220	The Transition Mode setting value is out of range.	Set a settable value in the Transition Mode.	Warning
201D	8221	The Transition Parameter setting value is out of range.	Set a settable value in the Transition Parameter.	Warning
201E	8222	The axis group operation was stopped due to an axis group configuration error.	After removing the error cause, clear the error with the axis or axis group error reset command and execute the command.	Warning
201F	8223	The command position value transmitted to the servo drive is out of the pulse unit expression value.	When the command position value was converted into pulse units, it was out of the 32-bit area When the command position values is converted to a pulse, it set within the range of -2147483648 to 2147483647) (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Warning
2020	8224	It is an undefined axis group command.	The command is not executed in the current version of the controller. Check the version in which the motion function block can be executed.	Warning
2021	8225	The same command was executed, thus canceling the previously executed command.	Make sure that the command is not executed again during the execution of the same command.	Warning

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
2022	8226	The number of Buffered commands that can be executed has been exceeded.	The command cannot be executed because the command buffer of the corresponding axis group is full. The number of Buffered Command that can be executed is 100. Adjust the command execution timing.	Warning
2030	8240	Axis group parameters cannot be written while the axis group is running.	Write the axis group parameter when the axis group is not in operation.	Warning
2040	8256	Axis group parameter data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
2041	8257	Operation cannot be executed due to the axis group parameters error.	Check the axis group parameters, and set again.	Alarm
2042	8258	The speed limit of the axis group parameter cannot be set to a value less than 0.	Set the speed limit value to greater than zero.	Warning
2043	8259	The configuration axis number setting value of the axis group parameter exceeded the range.	Set the configuration axis of axis group (axis and encoder numbers) in the range by product.	Warning
2051	8273	The axis to be added is already registered in the axis group.	Please set another axis after checking whether the axis group has the same axis number.	Warning
2052	8274	The current axis group is active, and the axis you want to add is already included in another active axis group.	Execute the command after changing the active axis group including the relevant axis to the GroupDisabled state.	Warning
2053	8275	IdentInGroup setting value of axis group add/remove command exceeded the range.	Set the IdentInGroup setting value from 1 to 10.	Warning
2060	8288	There is no axis setting in the specified axis group of the enable/disable axis group command.	Set one or more axes in the relevant axis group and execute the corresponding command.	Warning
2061	8289	The axis group cannot be activated because there is an operating axis among the axes of the current axis group.	Execute the command in the state that all axes belonging to the relevant axis group are not in operation.	Warning
2062	8290	The designated axis group cannot be activated because the current axis group's configuration axis is a configuration axis of another active axis group.	Check that the axis belonging to the relevant axis group does not belong to another active axis group.	Warning
2063	8291	Axis group operation cannot be executed because the units of axis group configuration axes are different.	To execute the corresponding operation, set the unit of the configuration axes belonging to the axis group to be the same.	Warning
2064	8292	The axis group cannot be activated because parameter of axis group configuration occur error.	Set the parameters of the configuration axes belonging to the axis group within the normal range.	Warning
2065	8293	The axis group cannot be activated because the speed command unit of axis group configuration axes are different.	Set the speed command unit of the configuration axes belonging to the axis group to be the same.	Warning
2066	8294	Axis group cannot be activated because there is an axis whose speed command unit is rpm among the axes of the axis group.	The speed command unit of the configuration axes belonging to the axis group cannot be set as rpm. Set it to a value other than rpm.	Warning
2067	8295	Coordinates operation cannot be executed because the units of axis group configuration axes are different to coordinate system type.	To execute the corresponding operation, set the unit of the configuration axes belonging to the axis group to match with coordinate system type.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
206F	8303	If the axis group configuration axis is in NC control operation, the axis group cannot be activated.	Execute the command while the configuration axes belonging to the axis group are not in NC control operation.	Warning
2070	8304	The servo drive of the configuration axis does not support the homing mode.	Check whether the corresponding servo drive supports EtherCAT CoE Homing mode, and check the status of the servo drive.	Warning
2071	8305	There is an axis for which homing has not been completed normally among the configuration axes.	Check the component axis error code, remove the cause of the error of the relevant axis, and then execute the command again.	Warning
2072	8306	The axis group homing command cannot be executed while the axis group is running.	After the axis group stops operating, execute the axis group homing command again in the GroupStandby state.	Warning
2080	8320	There is an axis where an error occurred while setting the current position among the configuration axes.	Check the component axis error code, remove the cause of the error of the relevant axis, and then execute the command again.	Warning
2090	8336	Absolute coordinate linear interpolation command cannot be executed in undecided homing status.	Execute the command after making the homing state with the home return command or the current position setting command.	Warning
2091	8337	The speed limit of the linear interpolation axis has been exceeded.	Execute the command by lowering the command speed so as not to exceed the speed limit of the component axis.	Warning
2092	8338	In the case of a transition with a designated corner distance, the transition operation cannot be executed because the designated corner distance is greater than the moving distance to the target position.	Set the corner distance value specified in the transition parameter to be smaller than the moving distance to the target position.	Warning
2093	8339	In the case of a transition with a specified corner distance, transition operation cannot be executed because the radius of the arc to be inserted exceeds 2147483647pulse.	Execute linear interpolation by resetting the target position or changing the transition mode so that the two lines are not nearly on a straight line.	Warning
2094	8340	Linear interpolation operation cannot be executed when the main axis or sub axis is "allowed" for infinite running repeat.	Execute the command after changing the setting of infinite running repeat of main axis or sub axis to "0: Prohibited".	Warning
20A0	8352	Circular Interpolation command cannot be executed in undecided homing status.	Execute the command after making the homing state with the home return command or the current position setting command.	Warning
20A1	8353	Circular interpolation mode setting value exceeded the range.	Set the circular interpolation mode to a value between 0 and 2 (0: auxiliary point, 1: center point, 2: radius).	Warning
20A2	8354	The circular interpolation pass selection setting value exceeds the range.	Set the circular interpolation pass selection setting value between 0 and 1 (0: CW, 1: CCW).	Warning
20A3	8355	The radius setting exceeded the range in the circular interpolation radius method.	Set radius setting from circular interpolation main axis operating data for 80% bigger than its half distance of beginning point to end point.	Warning
20A4	8356	Cannot be operated if circular interpolation start point =center point (middle point) or center point (middle point)= end point.	Execute circular interpolation after setting the center point (or middle point) to a position different from the starting point (or end point) in circular interpolation.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
20A5	8357	The start point and end point is not possible to be same in the middle point (radius) mode of circular interpolation.	If the circular interpolation method is set to the middle point (or radius), execute the circular interpolation after setting the position of the start point and the position of the end point differently.	Warning
20A6	8358	Radius setting error in circular interpolation.	The radius of the circle to carry out circular interpolation operation is from 0 to 2,147,483,2147483647pulse. Execute the command after setting the input value so that the radius can be calculated within the setting range. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Warning
20A7	8359	Not possible to carry out the operation as linear profile comes out of circular interpolation.	In case of circular interpolation middle point, execute circular interpolation after changing the middle point so that it is not located on a straight line between the start point and the endi point.	Warning
20A8	8360	Circular interpolation operation cannot be executed when the main axis or sub axis is "allowed" for infinite running repeat.	Execute the command after changing the setting of infinite running repeat of main axis or sub axis to "0: Prohibited".	Warning
20A9	8361	Circular interpolation cannot be executed when the number of axis group configuration axes is 4 or more.	In the case of circular interpolation, set the axis group to 2 axes, and in the case of helical interpolation, set the axis group to 3 axes.	Warning
20AA	8362	Circular interpolation cannot be executed if the axis configuration of the axis group is not configured in order.	In case of circular interpolation, set the axis group in order from the first.	Warning
20AB	8363	The speed limit of the circular interpolation axis has been exceeded.	Execute the command by lowering the command speed so as not to exceed the speed limit of the component axis.	Warning
20AC	8364	In circular interpolation, in the middle point (or radius) method, the middle point (center point) must exist in the same XY plane as the start point.	Execute circular interpolation after setting the center point (or middle point) to the same XY plane as the start point (or end point) in circular interpolation.	Warning
20C0	8384	Coordinates operation command cannot be executed in undecided homing status.	Execute the command after making the homing state with the home return command or the current position setting command.	Warning
20C1	8385	The PCS setting parameter data of the coordinate system parameter is abnormal.	Check the PCS setting parameters, and set again.	Warning
20C2	8386	The coordinate system type parameter data of the coordinate system parameter is abnormal.	Check the coordinate system type parameters, and set again.	Warning
20C3	8387	The coordinate system parameter data of the coordinate system parameter is abnormal.	Check the machine parameters, and set again.	Warning
20C4	8388	Workspace type data of coordinate system parameter is abnormal.	Check the workspace type parameters, and set again.	Warning
20C5	8389	The workspace parameter data of the coordinate system parameter is abnormal.	Check the workspace parameters, and set again.	Warning
20C6	8390	It is a position where coordinate system operation cannot be started.	Move to the position where you can start the coordinate system operation and execute the command.	Warning
20C7	8391	It is a position where coordinate system operation	Check if there are any abnormalities in the target	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		cannot be reach.	position or coordinate system parameters and set again.	
20C8	8392	Operating outside the workspace.	Check if there are any abnormalities in workspace parameter position at and set again.	Warning
20C9	8393	The axis group cannot be activated because the unit of axis group configuration is different to coordinate system type.	Set the unit of the configuration axes belonging to the axis group to match the coordinate system type.	Warning
20CA	8394	Coordinate system operation exceeded the maximum interpolation speed.	Execute the command by lowering the command speed so as not to exceed the maximum interpolation speed.	Warning
20CB	8395	Coordinate system operation cannot be executed when the configuration axis is "allowed" for infinite running repeat.	Execute the command after changing the setting of infinite running repeat of configuration axis to "0: Prohibited".	Warning
20CC	8396	Unsupported CoordSystem.	Set up a supported CoordSystem and execute the command.	Warning
20CD	8397	Unsupported TrajType.	Set up a supported TrajType and execute the command.	Warning
20D0	8400	The conveyor axis setting value is out of range.	Set conveyor axis (axis and encoder numbers) in the range by product.	Warning
20D1	8401	The axis set as the conveyor axis is set as the axis group configuration axis.	Execute the command with the conveyor axis set to another axis.	Warning
20D2	8402	There is an error in the unit setting of the conveyor axis.	Set the unit of the conveyor shaft to mm/inch.	Warning
20D3	8403	Conveyor axis are not ready to operating	Execute the command when the conveyor axis is ready for operation.	Warning
20D4	8404	Conveyor synchronous command cannot be executed when the main axis is in homing operation.	Execute the command when the conveyor axis is not in operation by homing.	Warning
20D5	8405	Conveyor synchronous command cannot be executed when the main axis is in torque control.	Execute the command when the conveyor axis is not in operation by torque control.	Warning
20D6	8406	Conveyor synchronous function conveyor axis cannot be executed when the configuration axis is "disabled" for infinite running repeat.	Execute the command after changing the setting of infinite running repeat of conveyor axis to "1: enabled".	Warning
20E0	8416	The step value of the coordinate system path operation exceeds the range.	Set the step value of the coordinates path operation to a value from 0 to 99 and execute the command.	Warning
20E1	8417	The commandType value of the coordinate system path operation exceeds the range.	Set thecommandType of the coordinates path operation to a value from 0 to 4 and execute the command.	Warning
20E2	8418	The Mode value of the coordinate system path operation exceeds the range.	Set the Mode value of the coordinates path operation to a value from 0 to 2 and execute the command.	Warning
20E3	8419	Coordinate system path operation exceeded the number of possible paths.	Set the step value of the coordinates path operation to a value from 0 to 99 and execute the command.	Warning
2110	8464	Axis group override command cannot be executed if it is not operating with axis group interpolation or jog operation.	Execute the axis group override command during axis group interpolation or jog operation.	Warning
2111	8465	The override factor of the axis group override command exceeded the range.	Set the VelFactor, AccFactor, and JerkFactor values of the axis group override command to 0 or more and	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
			execute the command.	
2112	8466	After reflecting factor of the axis group override command, the axis group operation speed value exceeded the maximum speed value.	Execute override within the range that does not exceed the maximum speed value of the relevant axis.	Warning
2113	8467	After reflecting factor of the axis group override command, the operation speed values belonging to axis group exceeded the maximum speed value.	Execute override within the range that does not exceed the axis included maximum speed value of the axis included in the relevant axis group.	Warning
2120	8480	Parameter number set value of read/write command of axis group parameter is out of range.	Execute the command after the parameter number setting value of the axis group parameters read parameter/write command set between 1 and 41.	Warning
2121	8481	Max interpolation speed setting values of axis group parameters write command is out of range.	Set the maximum speed interpolation value of the axis group as a positive number.	Warning
2122	8482	Coordinate system parameter setting values of axis group parameters write command is out of range.	Check the setting value of the parameter in the 'Coordinate system setting' item among the axis group parameters.	Warning
2123	8483	Workspace setting values of axis group parameter write command is out of range.	Check the setting value of the parameter in the 'workspace setting' item among the axis group parameters.	Warning
2124	8484	PCS setting values of axis group parameter write command is out of range.	Check the setting value of the parameter in the 'PCS setting' item among the axis group parameters.	Warning
2125	8485	The jog speed setting values of axis group parameters write command is out of range.	The jog speed setting value is a positive number, and please set it to less than the maximum interpolation speed setting value.	Warning
2126	8486	Configuration axis setting value of axis group parameter write command is out of range.	Execute the command after setting the configuration axis setting value of the axis group parameter to a value between 1 and 36.	Warning
2130	8496	Among the configuration axes, there is an axis that has not completed servo on/off normally.	Check the component axis error code, remove the cause of the error of the relevant axis, and then execute the command again.	Warning

(6) NC error information

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3000	12288	The NC channel is not ready for operation.	Check if the NC channel is ready for operation. To use NC channel, NC channel must be registered in NC parameter in XG5000.	Warning
3001	12289	NC Program data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
3002	12290	If the NC channel is in automatic operation, the program cannot be written.	If the NC channel is in automatic operation, write the program while automatic operation is stopped.	Warning
3003	12291	NC program writing was not completed normally. (File processing (DELETE) failure when writing NC program)	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
3004	12292	NC program writing was not completed normally. (File processing (OPEN) failure when writing NC program)	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
3005	12293	NC program writing was not completed normally. (File processing (WRITE) failure when writing NC program)	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
3006	12294	NC program writing was not completed normally. (File processing (CLOSE) failure when writing NC program)	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
3007	12295	If the NC channel is in automatic operation, the parameter cannot be written.	If the NC channel is in automatic operation, write the parameter while automatic operation is stopped.	Warning
3008	12296	During NC channel automatic operation, the mode of the controller is changed to STOP or ERROR status, so automatic operation cannot be continued.	Check if the mode of the controller is not changed to STOP or ERROR status while the NC channel is in automatic operation.	Alarm
3009	12297	Automatic operation cannot be continued because the EtherCAT network connection is cut during NC channel automatic operation.	Check if the network connection is not disconnected due to slave power error, network cable error, or noise inflow on the network cable while the NC channel is automatic running.	Alarm
300A	12298	During NC channel automatic operation, the controller is stopped by the ESTOP command and automatic operation cannot be continued.	Check that the controller is not stopped by the ESTOP command while the NC channel is in automatic operation.	Alarm
3011	12305	Automatic operation cannot be continued because the NC X axis is not ready for operation.	Check whether the NC X axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
3012	12306	Automatic operation cannot be continued because the NC Y axis is not ready for operation.	Check whether the NC Y axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3013	12307	Automatic operation cannot be continued because the NC Z axis is not ready for operation.	Check whether the NC Z axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3014	12308	Automatic operation cannot be continued because the NC A axis is not ready for operation.	Check whether the NC A axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3015	12309	Automatic operation cannot be continued because the NC B axis is not ready for operation.	Check whether the NC B axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3016	12310	Automatic operation cannot be continued because the NC C axis is not ready for operation.	Check whether the NC C axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3017	12311	Automatic operation cannot be continued because the NC U axis is not ready for operation.	Check whether the NC U axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3018	12312	Automatic operation cannot be continued because the NC V axis is not ready for operation.	Check whether the NC V axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3019	12313	Automatic operation cannot be continued because the NC W axis is not ready for operation.	Check whether the NC W axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
301A	12314	Automatic operation cannot be continued because the NC S axis is not ready for operation.	Check whether the NC S axis is in servo-off status or drive alarm has occurred. NC channel automatic operation can be executed when the configuration axis is servo-on and drive alarm does not occur.	Warning
3020	12320	It is an undefined NC channel command.	The NC command is not supported in the current version of the controller. Check the version in which the command can be executed.	Warning
3021	12321	The same NC Channel command was executed, thus canceling the previously executed command.	The corresponding NC channel command can only be executed once per scan. Change the operating condition of the program so that one NC channel command is executed per scan.	Warning
3030	12336	Interpreter (IPR) alarm occurred during NC channel automatic operation, so	Check the interpreter (IPR) error code among the NC channel flags. After resetting the NC channel with the NC	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		automatic operation cannot be continued.	reset command (NC_Reset), execute the automatic operation start command (NC_CycleStart) again.	
3031	12337	Program processor (PA) alarm occurred during NC channel automatic operation, so automatic operation cannot be continued.	Check the iProgram processor (PA) error code among the NC channel flags. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation start command (NC_CycleStart) again.	Alarm
3040	12352	During NC channel automatic operation, the command position setting value is out of the pulse unit expression value.	When the command position value was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3041	12353	The command position setting value of NC X axis is out of the pulse unit expression value.	When the command position value of NC X axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3042	12354	The command position setting value of NC Y axis is out of the pulse unit expression value.	When the command position value of NC Y axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3043	12355	The command position setting value of NC Z axis is out of the pulse unit expression value.	When the command position value of NC Z axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3044	12356	The command position setting value of NC A axis is out of the pulse unit expression value.	When the command position value of NC A axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3045	12357	The command position setting value of NC B axis is out of the pulse unit expression value.	When the command position value of NC B axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3046	12358	The command position setting value of NC C axis is out of the pulse unit expression value.	When the command position value of NC C axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3047	12359	The command position setting value of NC U axis is out of the pulse unit expression value.	When the command position value of NC U axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3048	12360	The command position setting value of NC V axis is out of the pulse unit expression value.	When the command position value of NC V axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3049	12361	The command position setting value of NC W axis is out of the pulse unit expression value.	When the command position value of NC W axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
304A	12362	The command position setting value of NC S axis is out of the pulse unit expression value.	When the command position value of NC S axis was converted into pulse units, it was out of the 32-bit area When the command position value is converted to a pulse, it check for within the range of -2147483648 to 2147483647. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Alarm
3050	12368	The command position of the NC channel configuration axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC configuration axis where the error occurred.	Alarm
3051	12369	The command position of the NC X axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC X.	Alarm
3052	12370	The command position of the NC Y axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC Y.	Alarm
3053	12371	The command position of the NC Z axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC Z	Alarm
3054	12372	The command position of the NC A axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC A.	Alarm
3055	12373	The command position of the NC B axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC B.	Alarm
3056	12374	The command position of the NC C axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC C.	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3057	12375	The command position of the NC U axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC U.	Alarm
3058	12376	The command position of the NC V axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC V.	Alarm
3059	12377	The command position of the NC W axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC W.	Alarm
305A	12378	The command position of the NC S axis is out of the software upper limit position.	Clear the error by executing the error reset command after out of the software upper limit by using the reverse jog command on the NC S.	Alarm
305B	12379	The command position of the NC channel/axis is out of the inner range of the G22 no-feed zone.	Clear the error by executing the error reset command after out of G22 No Traveling Area by using the jog command on the NC configuration axis where the error occurred.	Alarm
305C	12380	The command position of the NC channel/axis is out of the outer range of the G22 no-feed zone.	Clear the error by executing the error reset command after out of G22 No Traveling Area by using the jog command on the NC configuration axis where the error occurred.	Alarm
305D	12381	The command position of the NC channel/axis is out of range of the 3rd no-feed zone.	Clear the error by executing the error reset command after out of the 3rd no-feed zone by using the jog command on the NC configuration axis where the error occurred.	Alarm
3060	12384	The command position of the NC channel configuration axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC configuration axis where the error occurred.	Alarm
3061	12385	The command position of the NC X axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC X.	Alarm
3062	12386	The command position of the NC Y axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC Y.	Alarm
3063	12387	The command position of the NC Z axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC Z.	Alarm
3064	12388	The command position of the NC A axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC A.	Alarm
3065	12389	The command position of the NC B axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC B.	Alarm
3066	12390	The command position of the NC C axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC C.	Alarm
3067	12391	The command position of the NC U axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC U.	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3068	12392	The command position of the NC V axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC V.	Alarm
3069	12393	The command position of the NC W axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC W.	Alarm
306A	12394	The command position of the NC S axis is out of the software lower limit position.	Clear the error by executing the error reset command after out of the software lower limit by using the forward direction jog command on the NC S.	Alarm
3071	12401	Automatic operation cannot be continued because the NC X axis is not completion for homing.	Check if the NC X axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3072	12402	Automatic operation cannot be continued because the NC Y axis is not completion for homing.	Check if the NC Y axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3073	12403	Automatic operation cannot be continued because the NC Z axis is not completion for homing.	Check if the NC Z axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3074	12404	Automatic operation cannot be continued because the NC A axis is not completion for homing.	Check if the NC A axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3075	12405	Automatic operation cannot be continued because the NC B axis is not completion for homing.	Check if the NC B axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3076	12406	Automatic operation cannot be continued because the NC C axis is not completion for homing.	Check if the NC C axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3077	12407	Automatic operation cannot be continued because the NC U axis is not completion for homing.	Check if the NC U axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3078	12408	Automatic operation cannot be continued because the NC V axis is not completion for homing.	Check if the NC V axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3079	12409	Automatic operation cannot be continued because the NC W axis is not completion for homing.	Check if the NC W axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
307A	12410	Automatic operation cannot be continued because the NC S axis is not completion for homing.	Check if the NC S axis is not completion for homing. Use the homing command (MC_Home, NC_Home) to change the axis to the homing completion state.	Warning
3080	12416	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC component axis.	Remove the cause of the abnormal state after checking that the drive status of the NC configuration axis is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3081	12417	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition	Remove the cause of the abnormal state after checking that the drive status of the NC X is not changed to high/low limit, alarm or servo-off status during automatic operation	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		(high/low limit, alarm, servo-off) of the NC X.	of NC channel.	
3082	12418	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC Y.	Remove the cause of the abnormal state after checking that the drive status of the NC Y. is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3083	12419	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC Z.	Remove the cause of the abnormal state after checking that the drive status of the NC Z is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3084	12420	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC A.	Remove the cause of the abnormal state after checking that the drive status of the NC A is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3085	12421	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC B.	Remove the cause of the abnormal state after checking that the drive status of the NC B is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3086	12422	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC C.	Remove the cause of the abnormal state after checking that the drive status of the NC C is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3087	12423	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC U.	Remove the cause of the abnormal state after checking that the drive status of the NC U is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3088	12424	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC V.	Remove the cause of the abnormal state after checking that the drive status of the NC V is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
3089	12425	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC W.	Remove the cause of the abnormal state after checking that the drive status of the NC W is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning
308A	12426	During NC channel automatic operation, automatic operation cannot be continued due to an abnormal drive condition (high/low limit, alarm, servo-off) of the NC S.	Remove the cause of the abnormal state after checking that the drive status of the NC S is not changed to high/low limit, alarm or servo-off status during automatic operation of NC channel.	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
3100	12544	NC Channel parameter data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Alarm
3101	12545	Operation cannot be executed due to the NC parameters error.	Check the NC parameters, and if the settings such as the data range are abnormal, set them again.	Alarm
3102	12546	Upper/Lower speed limit of the cutting feed of NC channel parameter exceeded the range.	Upper/Lower speed limit of the cutting feed of NC channel parameter set to a values greater than 0. Set the Upper speed limit of the cutting feed higher than the lower speed limit of the cutting feed.	Alarm
3103	12547	The upper/lower limit cutting speed setting value of the circular milling speed limit in the NC channel parameter exceeded the range.exceeded the range.	Set the upper/lower limit cutting speed values of the circular milling speed limit in the NC channel parameter to a value greater than 0. Set the Upper speed limit of circular interpolation higher than the lower speed limit of circular interpolation.	Alarm
3104	12548	Max/Min Speed of Spindle at Constant Speed Control setting of NC channel parameter exceeded the range.	Set Max. Speed of Spindle at Constant Speed Control setting of NC channel parameter higher than Min Speed of Spindle at Constant Speed Control setting of NC channel parameter.	Alarm
3200	12800	NC Channel /axis parameter data is abnormal.	Download the data from XG5000 again, and if it occurs repeatedly after re-executing, request A/S.	Warning
3310	13072	The NC Feed Hold command was executed in a state other than automatic operation, or the currently executing program block is in a state where NC Feed Hold cannot be executed.	Execute the NC Feed Hold command (NC_FeedHold) while the NC channel is in automatic operation. If automatic operation is currently in progress, please check whether the currently executing program block is in a state where Feed Hold is possible.	Warning
3311	13073	Feed Hold function was canceled because reset by NC_Reset or NC_Emergency command was executed.	If the Feed Hold function is required, execute the command again in the automatic operating status after the NC_Reset or NC_Emergency command is finished.	Warning
3312	13074	NC_FeedHold command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3320	13088	The override factor of the NC rapid traverse override command exceeded the range.	Set the VelFactor, AccFactor, and JerkFactor values of the override command to 0 or more and execute the override command.	Warning
3321	13089	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC X axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC X axis, override the speed limit value within the range.	Warning
3322	13090	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC Y axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC Y axis, override the speed limit value within the range.	Warning
3323	13091	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC Z axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC Z axis, override the speed limit value within the range.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3324	13092	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC A axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC A axis, override the speed limit value within the range.	Warning
3325	13093	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC B axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC B axis, override the speed limit value within the range.	Warning
3326	13094	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC C axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC C axis, override the speed limit value within the range.	Warning
3327	13095	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC U axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC U axis, override the speed limit value within the range.	Warning
3328	13096	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC V axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC V axis, override the speed limit value within the range.	Warning
3329	13097	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC W axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC W axis, override the speed limit value within the range.	Warning
332A	13098	After the override factor of the NC rapid traverse override command is reflected, the operating speed value of the NC S axis exceeds the speed limit.	After checking the speed limit value of the axis connected to the NC S axis, override the speed limit value within the range.	Warning
332B	13099	Rapid Traverse override function was canceled because reset by NC_Reset or NC_Emergency command was executed.	If the Rapid Traverse override function is required, execute the command again after the NC_Reset or NC_Emergency command is finished.	Warning
332C	13100	NC_RapidTraverseOverride command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3330	13104	The override factor of the NC cutting feed override command exceeded the range.	Set the VelFactor, AccFactor, and JerkFactor values of the override command to 0 or more and execute the override command.	Warning
3331	13105	After the override factor of the NC cutting feed override command was reflected, the operating speed value exceeded the cutting feed upper limit speed value.	After checking the cutting feed upper limit speed value of the NC channel parameter, override it within the range that does not exceed the cutting feed upper limit speed value.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3332	13106	Cutting feed override function was canceled because reset by NC_Reset or NC_Emergency command was executed.	If the cutting feed override function is required, execute the command again after the NC_Reset or NC_Emergency command is finished.	Warning
3333	13107	NC_CuttingFeedOverride command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3340	13120	The override factor of the NC Spindle override command exceeded the range.	Set the VelFactor, AccFactor, and JerkFactor values of the override command to 0 or more and execute the override command.	Warning
3341	13121	After the override factor of the NC Spindle override command is reflected, the operating speed value of Spindle exceeds the speed limit.	After checking the speed limit value of the axis connected to spindle axis, override the speed limit value within the range.	Warning
3342	13122	Spindle override function was canceled because reset by NC_Reset or NC_Emergency command was executed.	If the Spindle override function is required, execute the command again after the NC_Reset or NC_Emergency command is finished.	Warning
3343	13123	NC_SpindleOverride command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3344	13124	The spindle override command cannot be executed because the spindle axis (NC S axis) is not ready for operation.	The spindle axis (NC S axis) is in servo off state or a drive alarm occurs	Warning
3350	13136	The setting axis of the NC parameter read command is not activated as an NC axis.	Check if the setting axis of the NC parameter read command is registered as the NC channel/axis parameter. NC channel/ Axis can be registered in NC Channel parameter among motion data items of XG5000.	Warning
3351	13137	The axis setting value of the NC parameter read command exceeded the allowable range.	Set axis number to a value between 1 and 10. If the axis value is 0, read channel parameters, and if 1 to 10, read NC axis X to NC axis S.	Warning
3352	13138	The Parameter group setting value of the NC parameter read command exceeded the allowable range.	The setting range of the parameter group is 1 to 17 for channel parameters and 1 to 5 for channel/axis parameters. After checking the group number to which the parameter you want to read belongs, execute the parameter read command (NC_ReadParameter).	Warning
3353	13139	The parameter number set in the parameter group of the NC parameter read command is not supported.	Check whether the parameter number set in the channel parameter or channel/axis parameter group is supported. After checking the group number ans parameter number to which the parameter you want to read belongs, execute the parameter read command (NC_ReadParameter).	Warning
3362	13154	The NC_MirrorImage command was executed in a non-SingleBlock state.	To execute the mirror image (NC_MirrorImage) command during automatic operation, execute it after creating a single block state with the block operation designation (NC_BlockControl) command.	Warning
3370	13168	If the reverse operation buffer is 0, it cannot be executed.	Set the reverse operation buffer value of the NC channel parameter to a value between 1 and 50.	Warning
3380	13184	Block skip function was canceled because	If the block skip function is required, execute the command	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		reset by NC_Reset or NC_Emergency command was executed.	again after the NC_Reset or NC_Emergency command is finished.	
3381	13185	The tool position measurable position is out of range. (G37)	Execute the automatic tool measurement (NC_BlockSkip) command after checking the position where the tool position can be measured.	Warning
3500	13568	If the NC channel is in automatic operation, the automatic operation start command cannot be executed.	Check if the NC channel is in automatic operation. Start automatic operation again after the automatic operation is finished,	Warning
3501	13569	If the NC eed Hold command is enabled status, the automatic operation start command cannot be executed.	After releasing the Enable input of the NC Feed Hold command (NC_FeedHold), execute the automatic operation start command (NC_CycleStart) again.	Warning
3502	13570	If the NC emergency stop command is enabled status, the automatic operation start command cannot be executed.	After releasing the Enable input of the NC emergency stop command (NC_Emergency), execute the automatic operation start command (NC_CycleStart) again.	Warning
3503	13571	The automatic operation start command cannot be executed without the NC Interpreter (IPR) normally end.	After resetting the NC channel with the NC reset command (NC_Reset), and run automatic operation start command (NC_CycleStart) again.	Warning
3504	13572	The NC Interpreter (IPR) or Program Processor (PA) cannot execute the automatic operation start command in an error state.	Check the Interpreter (IPR) or Program Processor (PA) error code among the NC channel flags. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation start command (NC_CycleStart) again.	Warning
3505	13573	If the NC channel is not setting to execute program, the automatic operation start command cannot be executed.	After designating the program to be executed with the NC program designation command (NC_LoadProgram), execute the automatic operation start command (NC_CycleStart) again.	Warning
3506	13574	The automatic operation start command cannot be executed because the NC channel has reached the target machining quantity or the target machining quantity in M99 repeat machining.	After checking the machining quantity of the NC channel flag or the machining quantity in case of repeated machining of M99, check whether the target machining quantity has been reached. Execute the automatic operation start command (NC_CycleStart) again after resetting the machining quantity flag or M99 machining quantity flag in case of repeated machining.	Warning
3507	13575	When the NC M/S/T-code output strobe signal is On, the automatic operation start command cannot be executed.	After completing the NC M/S/T-code output strobe signal, execute the automatic operation start command (NC_CycleStart).	Warning
3508	13576	The NC Channel Interpreter (IPR) did not run normally.	After resetting the NC channel with the NC reset command (NC_Reset), and run automatic operation start command (NC_CycleStart) again.	Warning
3509	13577	If entering the NC feed prohibited zone, the automatic operation start command cannot be executed.	Clear the error by executing the error reset command after out of feed prohibited zone by using the jog command on the NC configuration axis where the error occurred.	Warning
350A	13578	NC_CycleStart command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
350B	13579	The automatic operation start command cannot be executed because the NC	Check whether the slave connected to the spindle axis supports csv or vl operation mode. It cannot be operated	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
		spindle device does not support csv or vl mode.	as a spindle axis because the spindle device does not support csv or vl mode.	
350C	13580	The automatic operation start command cannot be executed because there is no required object required for spindle operation in the PDO setting of the EtherCAT slave connected to the spindle axis.	Execute the automatic operation start command (NC_CycleStart) after resuming the EtherCAT connection by setting the necessary objects required for spindle operation in the PDO setting of the EtherCAT slave connected to the spindle axis. (Refer to the manual spindle device required PDO setting)	Warning
350D	13581	The automatic operation start command cannot be executed because the reverse operation buffer is all used during reverse operation.	After releasing the enable input of NC_Reset or NC_TetraceMove commands, execute the automatic operation start command (NC_CycleStart) again.	Warning
350F	13583	The automatic operation start command cannot be executed while writing the downloaded NC program during run.	Execute the automatic operation start command after the NC program write operation downloaded during run is completed.	Warning
3510	13584	If the NC channel is not ready for configuration axis, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3511	13585	If the NC X axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3512	13586	If the NC Y axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3513	13587	If the NC Z axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3514	13588	If the NC A axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3515	13589	If the NC B axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
3516	13590	If the NC C axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3517	13591	If the NC U axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3518	13592	If the NC V axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3519	13593	If the NC W axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
351A	13594	If the NC S axis is not ready, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when all NC channel configuration axes are ready. To start automatic operation, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3520	13600	If the NC channel configuration axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3521	13601	If the NC X axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3522	13602	If the NC Y axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3523	13603	If the NC Z axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3524	13604	If the NC A axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3525	13605	If the NC B axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3526	13606	If the NC C axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3527	13607	If the NC U axis is in operation, the	Execute the automatic operation start command	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		automatic operation start command cannot be executed.	(NC_CycleStart) when NC channel configuration axes are stop.	
3528	13608	If the NC V axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3529	13609	If the NC W axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
352A	13610	If the NC S axis is in operation, the automatic operation start command cannot be executed.	Execute the automatic operation start command (NC_CycleStart) when NC channel configuration axes are stop.	Warning
3530	13616	The automatic operation start command cannot be executed because the NC channel configuration axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3531	13617	The automatic operation start command cannot be executed because the NC X axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3532	13618	The automatic operation start command cannot be executed because the NC Y axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3533	13619	The automatic operation start command cannot be executed because the NC Z axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3534	13620	The automatic operation start command cannot be executed because the NC A axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3535	13621	The automatic operation start command cannot be executed because the NC B axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3536	13622	The automatic operation start command cannot be executed because the NC C axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3537	13623	The automatic operation start command cannot be executed because the NC U axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3538	13624	The automatic operation start command cannot be executed because the NC V axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3539	13625	The automatic operation start command cannot be executed because the NC W axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
353A	13626	The automatic operation start command cannot be executed because the NC S axis is activated as a motion axis group configuration axis.	Execute the automatic operation start command (NC_CycleStart) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3540	13632	The position unit or velocity unit setting of the NC channel configuration axis is incorrect.	For NC operation, set the unit of NC channel configuration axis (except spindle) to mm or deg. Set the speed unit in RPM for the spindle axis (S axis) and unit/min for other axes (X, Y, Z, A, B, C, U, V, W).	Warning
3541	13633	The position unit or velocity unit setting of the NC X axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3542	13634	The position unit or velocity unit setting of the NC Y axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3543	13635	The position unit or velocity unit setting of the NC Z axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3544	13636	The position unit or velocity unit setting of the NC A axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3545	13637	The position unit or velocity unit setting of the NC B axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3546	13638	The position unit or velocity unit setting of the NC C axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3547	13639	The position unit or velocity unit setting of the NC U axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3548	13640	The position unit or velocity unit setting of the NC V axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
3549	13641	The position unit or velocity unit setting of the NC W axis is incorrect.	For NC operation, set the unit of NC channel configuration axis to mm or deg. Set the speed unit as unit/min.	Warning
354A	13642	The position unit or velocity unit setting of the NC S axis is incorrect.	For NC operation, set the speed unit of spindle axis to RPM.	Warning
3600	13824	Cannot load program because the program set in the NC program designation command does not exist in the controller.	After writing the NC program in XG5000, write it to the controller and then execute the program designation command (NC_LoadProgram) again.	Warning
3601	13825	If the NC channel is in automatic operation, the program designation command cannot be executed.	Check if the NC channel is in automatic operation. Designate a new program after the automatic operation is finished,	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
3602	13826	If the NC program data is error, the program designation command cannot be executed.	After checking whether an abnormal NC program data error (0x3001) has occurred, download the data again from XG5000, and if it occurs repeatedly after re-executing, request A/S.	Warning
3603	13827	Invalid LoadMode of NC programming instruction.	After entering a value of 0 in LoadMode of the NC program designation command (NC_LoadProgram), execute the program designation command (NC_LoadProgram) again.	Warning
3604	13828	NC_LoadProgram command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3605	13829	While writing the downloaded NC program during run, the program designation command could not be executed.	Execute the program designation command after the NC program write operation downloaded during run is completed.	Warning
3610	13840	The NC Channel Interpreter (IPR) did not normally reset.	Reset the NC channel again with the NC reset command (NC_Reset). If it occurs repeatedly after retry, please request A/S.	Warning
3620	13856	The NC_Emergency command was executed without automatic operation.	Execute the NC emergency stop command (NC_Emergency) while the NC channel is in automatic operation.	Warning
3630	13872	It is not the range of homing that can be specified in NC homing operation.	The range of the home (ReferenceNum) is from the 1st home to the 4th home. Specify a value between 1 and 4.	Warning
3631	13873	The Nc_home return command cannot be executed when the channel is in automatic operation.	After the automatic operation is finished, execute the origin return command.	Warning
3632	13874	If the NC emergency stop command is enabled status, the home return command cannot be executed.	After releasing the Enable input of the NC emergency stop command (NC_Emergency), execute the home return command (NC_Home) again.	Warning
3633	13875	If the NC channel is not ready for configuration axis, the home return command cannot be executed.	Execute the home return command (NC_Home) when all NC channel configuration axes are ready. To execute home return command, the NC channel configuration axis must be connected to the network or set as a virtual axis.	Warning
3634	13876	The home return command cannot be executed because the NC channel configuration axis is activated as a motion axis group configuration axis.	Execute the home return command (NC_Home) while the NC channel configuration axis is inactive as a motion axis group configuration axis.	Warning
3635	13877	An error occurred during home return in the the servo drive.	Check the error factor of the servo drive, remove the servo drive error, and then execute homing.	Warning
3636	13878	NC_Home command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3637	13879	When the spindle axis is automatically controlled by the NC function module, if the 'Spindle encoder selection' item of the axis parameter is '0: Disabled', the home operation cannot be executed.	After correctly homing operation encoder connection method connected to the spindle axis in the 'Spindle encoder selection' item of the axis parameter, execute home operation command.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3638	13880	If the 'Spindle encoder selection' item of the axis parameter is '1: Motor ENC', the 'Position actual value (0x6064)' object does not exist in the TxPDO setting of the EtherCAT slave connected to the spindle axis, so the home operation cannot be executed.	If the 'Spindle encoder selection' item of the axis parameter is '1: Motor ENC', add the 'Position actual value (0x6064)' object to the TxPDO setting of the EtherCAT slave connected to the spindle axis, resume EtherCAT connection, and then execute home operation command.	Warning
3639	13881	If the 'Spindle encoder selection' item of the axis parameter is '2: Built-in ENC1', the setting of the encoder 1 parameter is incorrect and the home operation cannot be executed.	If the 'Spindle encoder selection' item of the axis parameter is '2: Built-in ENC1', set the encoder 1 unit = pulse, Encoder 1 maximum value = 2147483647, Encoder 1 minimum value = -2147483648 and then execute the home operation command.	Warning
363A	13882	If the 'Spindle encoder selection' item of the axis parameter is '3: Built-in ENC2', the setting of the encoder 2 parameter is incorrect and the home operation cannot be executed.	If the 'Spindle encoder selection' item of the axis parameter is '3: Built-in ENC2', set the encoder 2 unit = pulse, Encoder 2 maximum value = 2147483647, Encoder 2 minimum value = -2147483648 and then execute the home operation command.	Warning
3640	13888	NC M-code operation completion command cannot be executed when M-code output strobe signal is Off.	After checking the status of the M-code output strobe signal among the NC channel flags, execute the M-code operation completion command (NC_McodeComplete) when the M-code output strobe signal is On.	Warning
3641	13889	NC_McodeComplete command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3650	13904	NC S-code operation completion command cannot be executed when S-code output strobe signal is Off.	After checking the status of the S-code output strobe signal among the NC channel flags, execute the S-code operation completion command (NC_ScodeComplete) when the S-code output strobe signal is On.	Warning
3651	13905	NC_ScodeComplete command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3660	13920	NC T-code operation completion command cannot be executed when T-code output strobe signal is Off.	After checking the status of the T-code output strobe signal among the NC channel flags, execute the T-code operation completion command (NC_TcodeComplete) when the S-code output strobe signal is On.	Warning
3661	13921	NC_TcodeComplete command cannot be executed while reset by NC_Reset or NC_Emergency command is in progress.	Execute the NC command after the NC_Reset or NC_Emergency command is finished.	Warning
3670	13936	The parameter write command cannot be executed when the channel is in automatic operation.	Check if the NC channel is in automatic operation. Execute the NC parameter write command (NC_WriteParameter) in the stop state after automatic operation is finished.	Warning
3671	13937	The setting axis of the NC parameter write command is not activated as an NC axis.	Check if the setting axis of the NC parameter write command is registered as the NC channel/axis parameter. NC channel/ Axis can be registered in NC Channel parameter among motion data items of XG5000.	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
3672	13938	The axis setting value of the NC parameter write command exceeded the allowable range.	Set axis number to a value between 1 and 10. If the axis value is 0, write channel parameters, and if 1 to 10, write NC axis X to NC axis S.	Warning
3673	13939	The Parameter group setting value of the NC parameter write command exceeded the allowable range.	The setting range of the parameter group is 1 to 17 for channel parameters and 1 to 5 for channel/axis parameters. After checking the group number to which the parameter you want to write belongs, execute the parameter write command (NC_WriteParameter).	Warning
3674	13940	The parameter number set in the parameter group of the NC parameter write command is not supported.	Check whether the parameter number set in the channel parameter or channel/axis parameter group is supported. After checking the group number and parameter number to which the parameter you want to write belongs, execute the parameter write command (NC_WriteParameter).	Warning
3675	13941	NC The data set value of the parameter set in the NC parameter write command is out of range.	After checking the data setting range of the parameter you want to set, execute the parameter write command (NC_WriteParameter) with a value within the range.	Warning
3690	13968	Only the Cancel Tool Retract/Recover Mode (0), Tool Retract Mode (1) or Tool Recover Mode (2) commands can be commanded.	After specifying Cancel Tool Retract/Recover Mode (0), Tool Retract Mode (1), or Tool Recover Mode (2) in the ToolMode input, execute the Tool Retract/Recover (NC_ToolMode) command.	Warning
3691	13969	Tool Retract cannot be commanded if it is not in tool Recover mode.	Execute the Tool Recover Mode (2) command after tool Retract is completed in Tool Retract Mode (1).	Warning
3692	13970	If the tool Retract operation is not performed in the tool Retract mode, the tool Recover command cannot be commanded.	Execute tool Recover mode (2) command after executing tool Retract operation with JOG operation in tool Retract mode.	Warning
36A0	13984	Block Optional Skip number is out of range. (0~9)	Set the value to be set for SkipNum from 0 to 9 and execute the command.	Warning
36B0	14000	The axis setting value of the manual tool correction command exceeded the allowable range. (X, Y, Z)	Set one of the X~Z (1~3) axes in NcAxis and execute the command.	Warning
36B1	14001	The command cannot be executed because the axis of the manual tool correction command is not prepared as a configuration axis.	Set the axis set in NcAxis as a configuration axis and execute the command.	Warning
36B2	14002	The axis of the manual tool correction command is not activated as an NC axis.	Assign the axis to the NC axis set in NcAxis and execute the command.	Warning
36B3	14003	The manual tool correction cannot be executed when the channel is in automatic operation.	After the automatic operation is finished, execute the origin return command.	Warning
36C0	14016	Spindle shaft gear change speed setting value exceeds axis speed limit.	Set the speed to be set in ChgVelocity of the NC_ChgSpindleGear command within the speed limit of the relevant axis and execute the command.	Warning
36C1	14017	Spindle axis maximum speed value is less than or equal to zero.	Set the MaxVelocity value of the NC_ChgSpindleGear command to a value greater than 0 and execute the command.	Warning
36C2	14018	The gear ratio setting value on the motor	Set the GearOfMotor value of the NC_ChgSpindleGear	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		side is less than or equal to 0.	command to a value greater than 0 and execute the command.	
36C3	14019	The gear ratio setting value on the mechanical side is less than or equal to 0.	Set the GearOfMachine value of the NC_ChgSpindleGear command to a value greater than 0 and execute the command.	Warning
36C4	14020	Backlash setting is less than 0.	Set the Backlash value of the NC_ChgSpindleGear command to a value greater than or equal to 0 and execute the command.	Warning
36C5	14021	The P Gain setting is less than 0 or greater than 500.	Set the P gain value of the NC_ChgSpindleGear command to a value in the range 0 to 500 and execute the command.	Warning
36C6	14022	The FF Gain setting is less than 0 or greater than 100.	Set the FF gain value of the NC_ChgSpindleGear command to a value in the range 0 to 100 and execute the command.	Warning
36C7	14023	Spindle axis is not in operation with NC automatic operation.	Operate the spindle axis with NC automatic operation and perform NC_ChgSpindleGear operation.	Warning
36D0	14032	The spindle control command cannot be executed because the main spindle axis is not set to be automatically controlled by the NC function module.	After setting the spindle axis (S axis) in the NC channel/axis of the NC parameter, set the motor axis number connected to the spindle axis (S axis) in the 'Main spindle axis number' of the NC channel parameter.	Warning
36D1	14033	The spindle control command cannot be executed because the spindle axis is not ready for operation.	The motor axis connected to the spindle axis is not currently ready for operation. After making the axis ready for operation with the LS_Connect command, execute the spindle control command.	Warning
36E0	14048	The SD card is not installed properly.	Please check whether the SD card is properly installed and ready for use.	Warning
36E1	14049	SrcProgramName program does not exist.	Check if the program of SrcProgramName is saved, and enter the program name of the command correctly.	Warning
36E2	14050	DstProgramName program is set to overwrite protection.	To modify the program in DstProgramName, set it to allow overwriting when executing the command.	Warning
36E3	14051	An error occurred while reading the file.	Please check if the status of the saved file is normal.	Warning
3800	14336	In the NC rapid traverse command, the operating speed value of the configuration axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC configuration axis in which the error occurred, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3801	14337	In the NC rapid traverse command, the operating speed value of the NC X axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC X axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3802	14338	In the NC rapid traverse command, the operating speed value of the NC Y exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC Y axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3803	14339	In the NC rapid traverse command, the operating speed value of the NC Z exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC Z axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3804	14340	In the NC rapid traverse command, the operating speed value of the NC A exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC A axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3805	14341	In the NC rapid traverse command, the operating speed value of the NC B exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC B axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3806	14342	In the NC rapid traverse command, the operating speed value of the NC C exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC C axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3807	14343	In the NC rapid traverse command, the operating speed value of the NC U exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC U axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3808	14344	In the NC rapid traverse command, the operating speed value of the NC V exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC V axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
3809	14345	In the NC rapid traverse command, the operating speed value of the NC W exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC W axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
380A	14346	In the NC rapid traverse command, the operating speed value of the NC S exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC S axis, set the rapid traverse speed within the range that does not exceed the speed limit.	Warning
380B	14347	During NC rapid traverse operation, the in position of the axes of rapid traverse was not completed within the in position completion monitoring time.	Check the command in position width and the in position completion monitoring time of the NC channel parameters among the parameters of the axes connected to the NC axis.	Warning
380C	14348	During NC rapid traverse operation, automatic operation cannot be continued because there is an error among the NC configuration axes.	Check the axis in which the error occurred among the NC configuration axes. Check the error occurred in the NC axis from the axis error code number of the NC channel/axis flag.	Alarm
3810	14352	In NC feed-per-revolution mode, the speed of the cutting feed operation is specified as zero.	Set the speed of cutting feed operation to a non-zero value in NC feed-per-revolution mode.	Warning
3811	14353	The operating speed of the NC cutting feed command exceeded the cutting feed upper limit speed value of the NC channel parameter.	After checking the cutting feed upper limit value of the NC channel parameter, set the cutting feed rate value (F) within the range that does not exceed the parameter value.	Warning
3812	14354	During NC cutting traverse operation, the in position of the axes of cutting traverse was not completed within the in position completion monitoring time.	Check the command in position width and the in position completion monitoring time of the NC channel parameters among the parameters of the axes connected to the NC axis.	Warning
3820	14368	In the NC cutting feed command, the operating speed value of the configuration	After checking the speed limit value of the axis connected to the NC configuration axis in which the error occurred,	Warning

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
		axis exceeded the speed limit.	set the cutting traverse speed within the range that does not exceed the speed limit.	
3821	14369	In the NC cutting traverse command, the operating speed value of the NC X axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC X axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3822	14370	In the NC cutting traverse command, the operating speed value of the NC Y axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC Y axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3823	14371	In the NC cutting traverse command, the operating speed value of the NC Z axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC Z axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3824	14372	In the NC cutting traverse command, the operating speed value of the NC A axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC A axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3825	14373	In the NC cutting traverse command, the operating speed value of the NC B axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC B axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3826	14374	In the NC cutting traverse command, the operating speed value of the NC C axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC C axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3827	14375	In the NC cutting traverse command, the operating speed value of the NC U axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC U axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3828	14376	In the NC cutting traverse command, the operating speed value of the NC V axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC V axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3829	14377	In the NC cutting traverse command, the operating speed value of the NC W axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC W axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
382A	14378	In the NC cutting traverse command, the operating speed value of the NC S axis exceeded the speed limit.	After checking the speed limit value of the axis connected to the NC S axis, set the cutting traverse speed within the range that does not exceed the speed limit.	Warning
3840	14400	NC cannot be operated if NC circular interpolation start point =center point or center point= end point.	In NC circular interpolation, set the position of the center point to a position different from the start point (or end point).	Warning
3841	14401	Invalid radius setting in NC circular interpolation.	The circular radius value in which NC circular interpolation operation can be executed is greater than 0 and less than 2147483647pulse based on the pulse unit. Set the center point or radius input value so that the radius can be calculated within the setting range. (When using the 'Position Control Range Expansion' function, 48-bit INT range can be set.)	Warning

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
3850	14416	Invalid axis designation in NC cylindrical interpolation.	When performing circular interpolation operation in NC cylindrical interpolation, the Y axis in the XY plane, the Z axis in the YZ plane, and the Z axis in the ZX plane must be designated.	Warning
3860	14432	The Dwell method is specified as the number of revolutions, but the number of revolutions is zero.	Drive the S axis with MC_MoveVelocity in the NC program.	Warning
3870	14448	An error occurred in the spindle axis during NC channel automatic operation, so automatic operation cannot be continued.	Check the error code that occurred in the spindle axis, change the spindle axis to an operable state, and then execute the automatic operation start command (NC_CycleStart).	Warning
3871	14449	The operation mode of the slave connected to the spindle axis cannot be changed.	Check whether the slave connected to the spindle axis supports csv or vl operation mode. Check if the operation status of the slave is normal.	Alarm
3872	14450	Orientation operation cannot be executed because the spindle axis is not home completion.	Execute the NC_Home or MC_SetPosition command on the spindle axis to make the home completion state and then execute the spindle orientation operation.	Warning
3873	14451	When the spindle axis is automatically controlled by the NC function module, if the 'Spindle encoder selection' item of the axis parameter is '0: Disabled', the orientation operation (M19) cannot be executed.	After correctly homing operation encoder connection method connected to the spindle axis in the 'Spindle encoder selection' item of the axis parameter, execute orientation operation command (M19).	Warning
3874	14452	If the 'Spindle encoder selection' item of the axis parameter is '1: Motor ENC', the 'Position actual value (0x6064)' object does not exist in the TxPDO setting of the EtherCAT slave connected to the spindle axis, so orientation operation (M19) cannot be executed.	Add 'Position actual value (0x6064)' object to the TxPDO setting of the EtherCAT slave connected to the spindle axis, resume EtherCAT connection, and then execute Orientation operation (M19).	Warning
3875	14453	If the 'Spindle encoder selection' item of the axis parameter is '2: Built-in ENC1', the setting of the encoder ENC1 is incorrect and Orientation operation (M19) cannot be executed.	If the 'Spindle encoder selection' item of the axis parameter is '2: Built-in ENC1', set the encoder 1 unit = pulse, Encoder 1 maximum value = 2147483647, Encoder 1 minimum value = -2147483648 and then execute the orientation operation command (M19), it.	Warning
3876	14454	If the 'Spindle encoder selection' item of the axis parameter is '3: Built-in ENC2', the setting of the encoder ENC2 is incorrect and Orientation operation (M19) cannot be executed.	If the 'Spindle encoder selection' item of the axis parameter is '3: Built-in ENC2', set the encoder 2 unit = pulse, Encoder 2 maximum value = 2147483647, Encoder 2 minimum value = -2147483648 and then execute the orientation operation command (M19), it.	Warning
3880	14464	The reference axis of constant speed control is not set. Cannot switch to constant speed control (G96) mode	Execute the command after setting the 'reference axis for constant speed control' setting value of the NC channel parameter to a value within the range of '1: X~9: W'.	Warning

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
3F00	16128	Interpreter (IPR) Parsing Error - Invalid LEX MAIN TABLE configuration	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F01	16129	Interpreter (IPR) Parsing Error - Undefined character exists.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F02	16130	Interpreter (IPR) parsing error - the number has exceeded the maximum buffer.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F03	16131	Interpreter (IPR) parsing error - tThe number of LEX tokens exceeds the maximum buffer.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F04	16132	Interpreter (IPR) parsing error - more than one decimal point.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F05	16133	Interpreter (IPR) parsing error - the number of parentheses in the formula does not match.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F06	16134	Interpreter (IPR) Parsing Error - an invalid character exists in the formula.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F07	16135	Interpreter (IPR) parsing error - the grammar of the formula is not correct.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F08	16136	Interpreter (IPR) parsing error - this is not an allowed macro variable.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F09	16137	Interpreter (IPR) parsing error - TANGENT operation error.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with	Alarm

Error code		Error Description	Action	Alarm/ Warning
Hex	**Dec**			
			the NC reset command (NC_Reset), execute the automatic operation again.	
3F0A	16138	Interpreter (IPR) parsing error - SQUARE ROOT operation error.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F0B	16139	Interpreter (IPR) parsing error - the denominator of division cannot be zero	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F0C	16140	Interpreter (IPR) parsing error - the grammar is not correct.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F0D	16141	Interpreter (IPR) Parsing Error - Invalid YACC MAIN TABLE configuration	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F0E	16142	Interpreter (IPR) parsing error - tThe number of YACC tokens exceeds the maximum buffer.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F0F	16143	Interpreter (IPR) parsing error - Unable to open IPR semaphore.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F10	16144	Interpreter (IPR) parsing error - exited without M02 or M30.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F11	16145	Interpreter (IPR) parsing error - command can be given only at the beginning of the block.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F12	16146	Interpreter (IPR) parsing error - the same progress block exists.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3F13	16147	Interpreter (IPR) parsing error - The number of statements exceeds the maximum buffer	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F14	16148	Interpreter (IPR) parsing error - Could not find next block to proceed.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F15	16149	Interpreter (IPR) parsing error - The subprogram call syntax is incorrect.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F16	16150	Interpreter (IPR) parsing error - The maximum subprogram calls have been exceeded.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F17	16151	Interpreter (IPR) parsing error - The program has already been called.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F18	16152	Interpreter (IPR) parsing error - there is no M99 in the subprogram.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset),execute the automatic operation again.	Alarm
3F19	16153	Interpreter (IPR) parsing error - the grammar of the M99 is not correct.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F1A	16154	Interpreter (IPR) parsing error - the number of loops is too large.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F1B	16155	Interpreter (IPR) parsing error - there is no start of the loop.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F1C	16156	Interpreter (IPR) parsing error - invalid connection of loop.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
			the NC reset command (NC_Reset), execute the automatic operation again.	
3F1D	16157	Interpreter (IPR) parsing error - exceeded the M command limit within one block	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F1E	16158	Interpreter (IPR) parsing error - this is an unused G code.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F1F	16159	Interpreter (IPR) parsing error - Simultaneous commands cannot be given to one block.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F20	16160	Interpreter (IPR) parsing error - The center point of the arc could not be found.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F21	16161	Interpreter (IPR) parsing error - It is not possible to create a path for cycle code.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F22	16162	Interpreter (IPR) parsing error - the taper amount of the cycle code is too large.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F23	16163	Interpreter (IPR) parsing error - It cannot be commanded within the cycle shape block.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F24	16164	Interpreter (IPR) parsing error - there is a problem with the cycle shape block command	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F26	16166	Interpreter (IPR) parsing error - the tool offset number is not valid.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm

| Error code | | Error Description | Action | Alarm/ |
Hex	Dec			Warning
3F27	16167	Interpreter (IPR) parsing error - the position of the center point of the arc does not match	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F28	16168	Interpreter (IPR) parsing error - It cannot make subprogram calls in MDI mode.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F29	16169	Interpreter (IPR) parsing error - Chamfering and rounding are only applied to the cutting feed command.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F2A	16170	Interpreter (IPR) parsing error - Chamfering and rounding have been duplicated.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F2B	16171	Interpreter (IPR) parsing error - For chamfering and rounding, only single axis command is available.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F2C	16172	Interpreter (IPR) parsing error - the command value for chamfering and rounding is greater than the feed amount.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F2D	16173	Interpreter (IPR) parsing error - When chamfering and rounding, the following block information cannot be obtained.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F2E	16174	Interpreter (IPR) parsing error - When chamfering and rounding, an arc cannot come to the next block.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F2F	16175	Interpreter (IPR) parsing error - Rounding cannot be performed in the same straight line feed.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F30	16176	Interpreter (IPR) parsing error - compensation start and end can only be traversed in a straight line.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
			the NC reset command (NC_Reset), execute the automatic operation again.	
3F31	16177	Interpreter (IPR) parsing error - there is no feed command in the cycle shape end block.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F32	16178	Interpreter (IPR) parsing error - there is an axis command independent of the plane during chamfering and rounding.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F33	16179	Interpreter (IPR) parsing error - IJK command limit was exceeded within one block when calling macro.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F34	16180	Interpreter (IPR) parsing error - Modal macros cannot be called from subprograms.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F35	16181	Interpreter (IPR) parsing error - there is exceeded the limit of multiple calls to a modal macro.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F36	16182	Interpreter (IPR) parsing error -This is an unused M code.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F37	16183	Interpreter (IPR) parsing error -Pitch cannot be calculated for rigid tapping	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F38	16184	Interpreter (IPR) parsing error - string exceeded max buffer.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F39	16185	Interpreter (IPR) parsing error - string construction syntax is incorrect.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm

Error code		Error Description	Action	Alarm/ Warning
Hex	Dec			
3F3A	16186	Interpreter (IPR) parsing error - target processing quantity has been reached.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F3B	16187	Interpreter (IPR) parsing error - user stop of macro program.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F3C	16188	Interpreter (IPR) parsing error - It is not possible to create a path for a compound screw cycle.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F3D	16189	Interpreter (IPR) parsing error - It cannot be commanded during polar coordinate interpolation.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F3E	16190	Interpreter (IPR) parsing error - it cannot feed to 0 during polar coordinate interpolation.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F3F	16191	Interpreter (IPR) parsing error - it is a grammatical error when commanding a circular interpolation.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F40	16192	Interpreter (IPR) parsing error - It cannot be commanded during circular interpolation.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F41	16193	Interpreter (IPR) parsing error - It is a constant peripheral speed control mode in polar coordinates and cylindrical interpolation.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3F42	16194	Interpreter (IPR) parsing error - It is not the homing.	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm

Error code		Error Description	Action	Alarm/
Hex	Dec			Warning
3F43	16195	Interpreter (IPR) parsing error - tool interference has occurred	After checking the 'error block number' among the NC channel flags, check that there is no program error in the corresponding block. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3FE0	16352	Program processor (PA) error - The corresponding pointer location in the program file does not exist.	After resetting the NC channel with the NC reset command (NC_Reset), and run automatic operation again.	Alarm
3FE1	16353	Program processor (PA) error - unable to read from program file.	After resetting the NC channel with the NC reset command (NC_Reset), and run automatic operation again. Alarm	Alarm
3FE2	16354	Program processor (PA) error - there is no program file selected.	Check if the specified program is saved in the controller. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm
3FE3	16355	Program processor (PA) error - Unable to open NcAccess semaphore.	After resetting the NC channel with the NC reset command (NC_Reset), and run automatic operation again.	Alarm
3FE4	16356	Program processor (PA) error - The number of characters in one block is limited to 300.	Check that the number of characters in one block of the specified program does not exceed 300. After resetting the NC channel with the NC reset command (NC_Reset), execute the automatic operation again.	Alarm