A ALARM LIST

Appendix A, "ALARM LIST", consists of the following sections:

A.1 ALARM LIST (CNC)

(1) Alarms on program and operation (PS alarm)
(2) Background edit alarms (BG alarm)
(3) Communication alarms (SR alarm)

Alarm numbers are common to all these alarm types.
Depending on the state, an alarm is displayed as in the following examples:

PS"alarm number" Example: PS0003
BG"alarm number" Example: BG0085
SR"alarm number" Example: SR0001

Number	Message	Description
0001	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
0002	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting TVC parameter No. 0000#0 to “0”.
0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
0004	ADDRESS NOT FOUND	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.

Number	Message	Description
0005	NO DATA AFTER ADDRESS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0006	ILLEGAL USE OF MINUS SIGN	A minus sign (–) was specified at an NC instruction word or system variable where no minus signal may be specified.
0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two decimal points were specified.
0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.
0010	IMPROPER G-CODE	An unusable G code is specified.
0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead.
0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either divide the number of programmed move axes into two blocks.
0020	OVER TOLERANCE OF RADIUS	An arc was specified for which the difference in the radius at the start and end points exceeds the value set in parameter No. 3410. Check arc center codes I, J and K in the program. The tool path when parameter No. 3410 is set to a large value is spiral.
0021	ILLEGAL PLANE SELECT	The plane selection instructions G17 to G19 are in error. Reprogram so that same 3 basic parallel axes are not specified simultaneously. This alarm is also generated when an axis that should not be specified for plane machining is specified, for example, for circular interpolation. T In the 0*i* -TD, the helical interpolation option is needed to enable the specification of 3 or more axes for the G02/G03 block.
0022	R OR I,J,K COMMAND NOT FOUND	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.
0025	CIRCLE CUT IN RAPID (F0)	M F0 (rapid traverse in one-digit F code feed or inverse feed) was specified during circular interpolation (G02, G03).
0027	NO AXES COMMANDED IN G43/G44	M No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Multiple axes were specified for the same block when the tool length compensation type is C.

Number	Message	Description
0028	ILLEGAL PLANE SELECT	The plane selection instructions G17 to G19 are in error. Reprogram so that same 3 basic parallel axes are not specified simultaneously. This alarm is also generated when an axis that should not be specified for plane machining is specified, for example, for circular interpolation. T In the 0*i* -TD, the helical interpolation option is needed to enable the specification of 3 or more axes for the G02/G03 block.
0029	ILLEGAL OFFSET VALUE	Illegal offset No.
0030	ILLEGAL OFFSET NUMBER	An illegal offset No. was specified.
0031	ILLEGAL P COMMAND IN G10	Data input for the L No. of G10 or the corresponding function is not enabled. A data setting address such as P or R is not specified. An address command not concerned with data setting was specified. An address varies with the L No. The sign or decimal point of the specified address is in error, or the specified address is out of range.
0032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
0033	NO INTERSECTION AT G41/G42	The intersection cannot be obtained by the intersection calculation in tool radius/tool nose radius compensation. Modify the program.
0034	ONLY G00/G01 ALLOWED IN STUP/EXT BLK	An attempt was made to perform a start-up or cancel of cutter compensation or tool nose radius compensation not in the G00/G01 mode. Modify the program.
0035	CAN NOT COMMANDED G31	1) G31 cannot be specified. This alarm is generated when a G code (such as for tool radius/tool nose radius compensation) of group 07 is not canceled. 2) A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like.
0037	CAN NOT CHANGE PLANE IN G41/G42	The compensation plane G17/G18/G19 was changed during cutter or tool-nose radius compensation. Modify the program.
0038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool radius/tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
0039	CHF/CNR NOT ALLOWED IN G41,G42	T Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in G41 and G42 commands (tool nose radius compensation). The program may cause overcutting to occur in chamfering or corner R. Modify the program.
0041	INTERFERENCE IN G41/G42	In tool radius/tool nose radius compensation, excessive cutting may occur. Modify the program.
0042	G45/G48 NOT ALLOWED IN CRC	M Tool offset (G45 to G48) is commanded in tool radius compensation mode. Modify the program.
0044	G27-G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 (G29 is only for the M series) is commanded in canned cycle mode. Modify the program.
0045	ADDRESS Q NOT FOUND (G73/G83)	In a high-speed peck drilling cycle or peck drilling cycle, the amount of each-time cutting is not specified by address Q, or Q0 is specified. Modify the program.

Number	Message	Description
0046	ILLEGAL REFERENCE RETURN COMMAND	A command for a return to the second, third or fourth reference position is error. (The address P command is in error.)
0050	CHF/CNR NOT ALLOWED IN THRD BLK	The chamfering or corner R block is specified in a threading block. Modify the program.
0051	MISSING MOVE AFTER CNR/CHF	The travel or travel distance is incorrect in the block next to the chamfering or corner R. Modify the program.
0052	CODE IS NOT G01 AFTER CHF/CNR	T The block next to the chamfering or corner R block is not G01 (or vertical line). Modify the program.
0053	TOO MANY ADDRESS COMMANDS	T In the chamfering and corner R commands, two or more of I, J, K and R are specified.
0054	NO TAPER ALLOWED AFTER CHF/CNR	T A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
0055	MISSING MOVE VALUE IN CHF/CNR	The travel distance specified in the chamfering or corner R block is smaller than the amount of the chamfering or corner R. Modify the program.
0056	NO END POINT & ANGLE IN CHF/CNR	T In direct dimension drawing programming, both an end point and an angle were specified in the block next to the block in which only an angle was specified (Aa). Modify the program.
0057	NO SOLUTION OF BLOCK END	T Block end point is not calculated correctly in direct dimension drawing programming. Modify the program.
0058	END POINT NOT FOUND	T Block end point is not found in direct dimension drawing programming. Modify the program.
0060	SEQUENCE NUMBER NOT FOUND	[External data input/output] The specified number could not be found for program number and sequence number searches. A request was issued for input/output of an offset amount for tool data, but a tool number has never been entered after power-up. The tool data corresponding to the entered tool number could not be found. [External workpiece number search] The program corresponding to the specified workpiece number could not be found. [Program restart] In the program restart sequence number specification, the specified sequence number could not be found.
0061	P OR Q COMMAND IS NOT IN THE MULTIPLE REPETIVE CYCLES BLOCK	T Address P or Q is not specified in multiple repetitive cycle (G70, G71, G72, or G73) command.
0062	THE CUTTING AMOUNT IS ILLEGAL IN THE ROUGH CUTTING CYCLE	T A zero or a negative value was specified in a multiple repetitive canned rough-cutting cycle (G71 or G72) as the depth of cut.
0063	THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND	T The sequence number specified by addresses P and Q in multiple repetitive cycle (G70, G71, G72, or G73) command cannot be searched.

Number	Message	Description
0064	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)	T In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command for the first plane axis was not a monotonous increase or decrease.
0065	G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM	T In the first block of the shape program specified by P of the multiple repetitive canned cycle (G70, G71, G72, or G73), G00 or G01 was not specified.
0066	UNAVAILABLE COMMAND IS IN THE MULTIPLE REPETIVE CYCLES BLOCK	T An unavailable command was found in a multiple repetitive canned cycle (G70, G71, G72, or G73) command block.
0067	THE MULTIPLE REPETIVE CYCLES IS NOT IN THE PART PROGRAM STORAGE	T A multiple repetitive canned cycle (G70, G71, G72, or G73) command is not registered in a tape memory area.
0069	LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND	T In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle.
0070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
0071	DATA NOT FOUND	1) The address to be searched was not found. 2) The program with specified program number was not found in program number search. 3) In the program restart block number specification, the specified block number could not be found. Check the data.
0072	DATA NOT FOUND	The number of programs to be stored exceeded 400 (1-path system) or 800 (2-path system of T series). Delete unnecessary programs and execute program registration again.
0073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
0074	PROGRAM NUMBER ALREADY IN USE	The program number is other than 1 to 9999. Modify the program number.
0075	PROTECT	An attempt was made to register a program whose number was protected. In program matching, the password for the encoded program was not correct. An attempt was made to select a program being edited in the background as the main program. An attempt was made to call a program being edited in the background as a subprogram.
0076	PROGRAM NOT FOUND	The specified program is not found in the subprogram call or macro call. The M, G, or T codes are called by a P instruction other than that in an M98, M198, G65, G66, or interrupt type custom macro, and a program is called by a specific address. This alarm is also generated when a program is not found by these calls.
0077	TOO MANY SUB,MACRO NESTING	The total number of subprogram and macro calls exceeds the permissible range. Another subprogram call was executed during an external memory subprogram call.

Number	Message	Description
0078	SEQUENCE NUMBER NOT FOUND	The specified sequence No. was not found during sequence number search. The sequence No. specified as the jump destination in GOTO— and M99P— was not found.
0079	PROGRAM NOT MATCH	The program in memory does not match the program stored on tape. Multiple programs cannot be matched continuously when bit 6 (NPE) of parameter No. 3201 is set to “1”. Set bit 6 (NPE) of parameter No. 3201 to “0” before executing a match.
0080	G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT	M When the tool length measurement function (G37) is performed, a measuring position reached signal goes 1 in front of the area determined by the ε value specified in parameter No.6254. Alternatively, the signal does not go 1. T When the automatic tool compensation function (G36, G37) is used, a measuring position reached signals (XAE1, XAE2) does not go 1 within the range determined by the ε value specified in parameters Nos.6254 and 6255.
0081	G37 OFFSET NO. UNASSIGNED	M The tool length measurement function (G37) is specified without specifying an H code. Correct the program. T The automatic tool compensation function (G36, G37) is specified without specifying an T code. Correct the program.
0082	G37 SPECIFIED WITH H CODE	M The tool length measurement function (G37) is specified together with an H code in the same block. Correct the program. T The automatic tool compensation function (G36, G37) is specified together with an T code in the same block. Correct the program.
0083	G37 IMPROPER AXIS COMMAND	M An error has been found in axis specification of the tool length measurement function (G37). Alternatively, a move command is specified as an incremental command. Correct the program. T An error has been found in axis specification of the automatic tool compensation function (G36, G37). Alternatively, a command is specified as an incremental command. Correct the program.
0085	OVERRUN ERROR	The next character was received from the I/O device connected to reader/punch interface 1 before it could read a previously received character. An overrun, parity error, or framing error occurred during the reading by reader/punch interface 1. The number of bits in the entered data, the baud rate setting, or the I/O unit specification number is incorrect.

Number	Message	Description
0086	DR OFF	During I/O process by reader/punch interface 1, the data set ready input signal of the I/O device (DR) was OFF. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
0087	BUFFER OVERFLOW	During a read by reader/punch interface 1, although a read stop command was issued, more than 10 characters were input. The I/O device or printed circuit board was defective.
0090	REFERENCE RETURN INCOMPLETE	1) The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2) An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
0091	MANUAL REFERENCE POSITION RETURN IS NOT PERFORMED IN FEED HOLD	Manual return to the reference position cannot be performed when automatic operation is halted. Perform the manual return to the reference position when automatic operation is stopped or reset.
0092	ZERO RETURN CHECK (G27) ERROR	The axis specified in G27 has not returned to reference position. Reprogram so that the axis returns to reference position.
0094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the User's manual.
0095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece origin offset amount changed.) Perform the correct operation according to the User's manual.
0096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece origin offset amount changed.) Perform the correct operation according to the User's manual.
0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON or alarms 0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
0099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
0109	FORMAT ERROR IN G08	T A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
0110	OVERFLOW :INTEGER	An integer went out of range during arithmetic calculations.
0111	OVERFLOW :FLOATING	A decimal point (floating point number format data) went out of range during arithmetic calculations.
0112	ZERO DIVIDE	An attempt was made to divide by zero in a custom macro.
0113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
0114	ILLEGAL EXPRESSION FORMAT	The format used in an expression in a custom macro statement is in error. The parameter tape format is in error.
0115	VARIABLE NO. OUT OF RANGE	A number that cannot be used for a local variable, common variable, or system variable in a custom macro is specified.

Number	Message	Description
0116	WRITE PROTECTED VARIABLE	An attempt was made in a custom macro to use on the left side of an expression a variable that can only be used on the right side of an expression.
0118	TOO MANY BRACKET NESTING	Too many brackets “[ ]” were nested in a custom macro. The nesting level including function brackets is 5.
0119	ARGUMENT VALUE OUT OF RANGE	The value of an argument in a custom macro function is out of range.
0122	TOO MANY MACRO NESTING	Too many macro calls were nested in a custom macro.
0123	ILLEGAL MODE FOR GOTO/WHILE/DO	A GOTO statement or WHILE–DO statement was found in the main program in the MDI or DNC mode.
0124	MISSING END STATEMENT	The END instruction corresponding to the DO instruction was missing in a custom macro.
0125	MACRO STATEMENT FORMAT ERROR	The format used in a macro statement in a custom macro is in error.
0126	ILLEGAL LOOP NUMBER	DO and END Nos. in a custom macro are in error, or exceed the permissible range (valid range: 1 to 3).
0127	DUPLICATE NC,MACRO STATEMENT	An NC statement and macro statement were specified in the same block.
0128	ILLEGAL MACRO SEQUENCE NUMBER	The specified sequence No. could not be found for sequence number search. The sequence No. specified as the jump destination in GOTO-- and M99P-- could not be found.
0129	USE 'G' AS ARGUMENT	G is used as an argument in a custom macro call. G cannot be used as an argument.
0130	NC AND PMC AXIS ARE CONFLICTED	The NC command and the PMC axis control command were conflicted. Modify the program or ladder.
0136	SPOS AXIS - OTHER AXIS SAME TIME	T The M code for spindle positioning and an axis address not for a spindle positioning axis were specified simultaneously. Alternatively, the axis addresses of a spindle positioning axis and non-spindle positioning axis were specified simultaneously in the spindle positioning mode.
0137	M-CODE & MOVE CMD IN SAME BLK.	T The M code for spindle positioning and the axis address of a spindle positioning axis were specified simultaneously.
0139	CANNOT CHANGE PMC CONTROL AXIS	The PMC axis was selected for the axis for which the PMC axis is being controlled.
0140	PROGRAM NUMBER ALREADY IN USE	In the background, an attempt was made to select or delete the program being selected in the foreground. Perform the correct operation for the background edition.
0142	ILLEGAL SCALE RATE	M The scaling rate is 0 times or 10000 times or more. Modify the setting of the scaling rate. (G51P_ … or G51I_J_K_ … or parameter No. 5411 or 5421)
0143	COMMAND DATA OVERFLOW	An overflow occurred in the storage length of the CNC internal data. This alarm is also generated when the result of internal calculation of scaling (M series), coordinate rotation (M series), and cylindrical interpolation overflows the data storage. It also is generated during input of the manual intervention amount.
0144	ILLEGAL PLANE SELECTED	M The coordinate rotation plane and arc or cutter compensation plane must be the same. Modify the program.

Number	Message	Description
0145	ILLEGAL USE OF G12.1/G13.1	T The axis No. of plane selection parameter No. 5460 (linear axis) and No. 5461(axis of rotation) in the polar coordinate interpolation mode is out of range (1 to number of controlled axes).
0146	ILLEGAL USE OF G-CODE	T The G code must be G40 modal when the polar coordinate interpolation mode is set or canceled. An illegal G code was specified while in the polar coordinate interpolation mode. Only the following G codes can be specified in this mode. G01,G02,G03,G04,G40,G41,G42,G65,G66,G67, (G90 and G91 for the G code system B or C), G98,G99
0148	SETTING ERROR	M Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters Nos.1710 to 1714.
0149	FORMAT ERROR IN G10L3	In registration (G10L3 to G11) of tool life management data, an address other than Q1, Q2, P1, and P2 or an unusable address was specified.
0150	ILLEGAL LIFE GROUP NUMBER	The tool group number exceeded the maximum allowable value. The tool group number (P after specification of G10 L3;) or the group number given by the tool life management T code in a machining program.
0151	GROUP NOT FOUND AT LIFE DATA	The tool group specified in a machining program is not set in tool life management data.
0152	OVER MAXIMUM TOOL NUMBER	The number of tools registered in one group exceeded the maximum allowable registration tool number.
0153	T-CODE NOT FOUND	In registration of tool life data, a block in which the T code needs to be specified does not include the T code. Alternatively, in tool exchange method D, M06 is specified solely. Modify the program.
0154	NOT USING TOOL IN LIFE GROUP	The H99 command, D99 command, or the H/D code set by parameters Nos. 13265 and 13266 was specified when no tool belonging to a group is used.
0155	ILLEGAL T-CODE COMMAND	In a machining program, the T code specified in the same block as M06 does not correspond to the group in current use. Modify the program.
0156	P/L COMMAND NOT FOUND	The P and L commands are not specified in the beginning of a program for setting a tool group. Modify the program.
0157	TOO MANY TOOL GROUPS	In registration of tool life management data, the group setting command block counts of P (group number) and L (tool life) exceeded the maximum group count.
0158	TOOL LIFE VALUE OUT OF RANGE	The life value that is being set is too large. Change the setting.
0159	ILLEGAL TOOL LIFE DATA	Tool life management data is corrupted for some reason. Register the tool data in the tool group or the tool data in the group again by G10L3; or MDI input.
0160	MISMATCH WAITING M-CODE	A waiting M-code is in error. Different waiting M codes are specified for paths 1 and 2.
0163	ILLEGAL COMMAND IN G68/G69	T G68 and G69 are not independently commanded in balance cut.
0169	ILLEGAL TOOL GEOMETRY DATA	T Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.

Number	Message	Description
0175	ILLEGAL G07.1 AXIS	An axis which cannot perform cylindrical interpolation was specified. More than one axis was specified in a G07.1 block. An attempt was made to cancel cylindrical interpolation for an axis that was not in the cylindrical interpolation mode. In cylindrical interpolation mode, to specify circular interpolation including a rotation axis (when bit 0 (ROT) of parameter No. 1006 is 1 and parameter No. 1260 is set), the value of rotation axis parameter No. 1022 must not be 0 but 5, 6, or 7 for parallel axis specification.
0176	ILLEGAL G-CODE USE(G07.1 MODE)	A G code was specified that cannot be specified in the cylindrical interpolation mode. This alarm also is generated when an 01 group G code was in the G00 modal or code G00 was instructed. Cancel the cylindrical interpolation mode before instructing code G00.
0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 3770.
0194	SPINDLE COMMAND IN SYNCHRO-MODE	T A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode. M A Cs contour control mode or rigid tapping mode was specified during the spindle synchronous control mode or simple spindle synchronous control mode.
0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
0199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
0200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value was out of range or was not specified. The parameters Nos. 5241 to 5243 setting is an S value which can be specified for the rigid tap. Correct the parameters or modify the program.
0201	FEEDRATE NOT FOUND IN RIGID TAP	The command F code for a cutting feedrate is a zero. If the value of F command is much smaller than that of the S command, when a rigid tap command is specified, this alarm is generated. This is because cutting is not possible by the lead specified by the program.
0202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. (System error)
0203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
0204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (or G74) block. Modify the program.
0205	RIGID MODE DI SIGNAL OFF	Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (DGN G061.0) is not ON during execution of the G84 (or G74) block. Check the PMC ladder diagram to find the reason why the DI signal is not turned on.
0206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Modify the program.
0207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.

Number	Message	Description
0210	CAN NOT COMMAND M198/M99	1) The execution of an M198 or M99 command was attempted during scheduled operation. Alternatively, the execution of an M198 command was attempted during DNC operation. Modify the program. T 2) The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.
0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In feed axis control , the following errors occurred during the synchronous operation. 1) The program issued the move command to the slave axis. 2) The program issued the manual operation to the slave axis. 3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.
0214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool length compensation (M series) of the shift type is executed in the synchronous control. Correct the program.
0217	DUPLICATE G51.2(COMMANDS)	T G51.2 is further commanded in the G51.2 mode. Modify the program.
0218	NOT FOUND P/Q COMMAND	T P or Q is not commanded in the G51.2 block, or the command value is out of the range. Modify the program. For a polygon turning between spindles, more information as to why this alarm occurred is indicated in DGN No. 471.
0219	COMMAND G51.2/G50.2 INDEPENDENTLY	T G51.2 and 50.2 were specified in the same block for other commands. Modify the program in another block.
0220	ILLEGAL COMMAND IN SYNCHR-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. Modify the program or check the PMC ladder.
0221	ILLEGAL COMMAND IN SYNCHR-MODE	T Polygon machining synchronous operation and Cs axis contour control or balance cutting are executed at a time. Modify the program.
0222	DNC OP. NOT ALLOWED IN BG-EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.
0224	ZERO RETURN NOT FINISHED	1) A reference position return has not been performed before the start of automatic operation. (Only when bit 0 (ZRNx) of parameter No. 1005 is 0) Perform a reference position return. T 2) A command was specified for a spindle positioning axis not in the spindle positioning mode. Perform spindle orientation.
0230	R CODE NOT FOUND	M Cut depth R is not specified in the block including G161. Alternatively, the value specified for R is negative. Modify the program.
0231	ILLEGAL FORMAT IN G10 L52	Errors occurred in the specified format at the programmable-parameter input.
0232	TOO MANY HELICAL AXIS COMMAND	Three or more axes were specified as helical axes in the helical interpolation mode.

Number	Message	Description
0233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS-232-C interface, other users were using it.
0245	T-CODE NOT ALLOWED IN THIS BLOCK	One of the G codes, G04,G10,G28,G29 (M series),G30,G50 (T series), and G53, which cannot be specified in the same block as a T code, was specified with a T code.
0247	THE MISTAKE IS FOUND IN THE OUTPUT CODE OF DATA.	When an encrypted program is output, EIA is set for the output code. Specify ISO.
0250	TOOL CHANGE ILLEGAL Z AXIS COMMAND	A Z-axis move command was performed in the same block for M06 command.
0251	TOOL CHANGE ILLEGAL T COMMAND	An unusable T code was specified in M06Txx.
0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program.
0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although bit 0 (IDGx) of parameter No. 1012 was set to 1 to inhibit the reference position from being set again for a return to the reference position without a dog, an attempt was made to perform a manual return to the reference position.
0302	SETTING THE REFERENCE POSITION WITHOUT DOG IS NOT PERFORMED	The reference position could not be set for a return to the reference position without a dog. Possible causes are: - The axis was not moved in the direction of a return to the reference position for jog feeding. - The axis was moved in the direction opposite to the direction of a manual return to the reference position.
0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
0305	INTERMEDIATE POSITION IS NOT ASSIGNED	M Although a G28 (automatic return to the reference position) or G30 (return to the second, third, or fourth reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.
0306	MISMATCH AXIS WITH CNR/CHF	T The correspondence between the moving axis and the I, J, or K command is incorrect in a block in which chamfering is specified.
0307	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	An attempt was made to set a butt-type reference position for an axis for which to use the function to set a reference position without a dog.
0310	FILE NOT FOUND	The specified file could not be found during a subprogram or macro call.
0311	CALLED BY FILE NAME FORMAT ERROR	An invalid format was specified to call a subprogram or macro using a file name.
0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	T Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. Two or more blocks not to be moved exist in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (parameter No. 3405#4 = 1) was specified for direct input of drawing dimensions, a comma was specified.
0313	ILLEGAL LEAD COMMAND	T The variable-lead threading increment specified in address K exceeds the specified maximum value in variable-lead threading. Or, a negative lead value was specified.

Number	Message	Description
0314	ILLEGAL SETTING OF POLYGONAL AXIS	T An axis was specified invalidly in polygon turning. For polygon turning: 1) A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning between spindles: 1) Valid spindles are not specified. (Parameters Nos. 7640 to 7643) 2) A spindle other than the serial spindle. 3) A spindle is not connected.
0315	ILLEGAL NOSE ANGLE COMMAND IS IN THE THREAD CUTTING CYCLE	T An invalid tool tip angle is specified in a multiple repetitive canned threading cycle (G76).
0316	ILLEGAL CUTTING AMOUNT IS IN THE THREAD CUTTING CYCLE	T An minimum depth of cut higher than the thread height is specified in a multiple repetitive canned threading cycle (G76).
0317	ILLEGAL THREAD COMMAND IS IN THE THREAD CUTTING CYCLE	T A zero or a negative value is specified in a multiple repetitive canned threading cycle (G76) as the thread height or the depth of cut.
0318	ILLEGAL RELIEF AMOUNT IS IN THE DRILLING CYCLE	T Although an escape directions is set in a multiple repetitive canned cutting-off cycle (G74 or G75), a negative value is specified for Δd.
0319	THE END POINT COMMAND IS ILLEGAL IN THE DRILLING CYCLE	T Although the Δi or Δk travel distance is set to 0 in a multiple repetitive canned cutting-off cycle (G74 or G75), a value other than 0 us specified for a U or W.
0320	ILLEGAL MOVEMENT AMOUNT/CUTTING AMOUNT IS IN THE DRILLING CYCLE	T A negative value is specified in a multiple repetitive canned cutting-off cycle (G74 or G75) as Δi or Δk (travel distance/the depth of cut).
0321	ILLEGAL REPEATED TIME IS IN THE PATTERN REPEATING CYCLE	T A zero or a negative value is specified in a multiple repetitive canned closed loop cycle (G73) as a repeated time.
0322	FINISHING SHAPE WHICH OVER OF STARTING POINT	T An invalid shape which is over the cycle starting point is specified in a shape program for a multiple repetitive canned rough-cutting cycle (G71 or G72).
0323	THE FIRST BLOCK OF SHAPE PROGRAM IS A COMMAND OF TYPE II	T Type II is specified in the first block of the shape program specified by P in a multiple repetitive canned rough-cutting cycle (G71 or G72). For G71, Z(W) is specified. For G72, X(U) is specified.
0324	THE INTERRUPTION TYPE MACRO WAS DONE IN THE MULTIPLE REPETIVE CYCLES	T An interruption type macro was issued during the multiple repetitive canned cycle (G70, G71, G72, or G73).
0325	UNAVAILABLE COMMAND IS IN SHAPE PROGRAM	T An usable command was issued in a shape program for a multiple repetitive canned cycle (G70, G71, G72, or G73).
0326	LAST BLOCK OF SHAPE PROGRAM IS A DIRECT DRAWING DIMENSIONS	T In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for direct input of drawing dimensions in the last block is terminated in the middle.

Number	Message	Description
0327	MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE	T A multiple repetitive canned cycle (G70, G71, G72, or G73) was commanded in a modal state in which a multiple repetitive canned cycle could not be commanded.
0328	ILLEGAL WORK POSITION IS IN THE TOOL NOSE RADIUS COMPENSATION	T The specification for the blank side for a tool nose radius compensation (G41 or G42) is incorrect in a multiple repetitive canned cycle (G71 or G72).
0329	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)	T In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command of the second plane axis was not a monotonous increase or decrease.
0330	ILLEGAL AXIS COMMAND IS IN THE TURNING CANNED CYCLE	T An axis other than the plane is specified n a canned cycle(G90, G92, or G94).
0334	OFFSET IS OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (malfunction prevention function)
0336	TOOL COMPENSATION COMMANDED MORE TWO AXES	M For a tool length compensation C, an attempt was made to command the offset to other axes without canceling the offset. Or, for a tool length compensation C, multiple axes are specified in G43 or G44 block.
0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (malfunction prevention function)
0338	ILLEGAL EXEC SEQ OF BLOCK	An incorrect value was detected in a check sum. (malfunction prevention function)
0345	TOOL CHANGE ILLEGAL Z AXIS POS	A tool change position on the Z-axis is incorrect.
0346	TOOL CHANGE ILLEGAL TOOL NUM	A tool number for tool change is incorrect.
0347	TOOL CHANGE ILLEGAL COMMAND IN SAME BLK.	Tool changing is commanded twice or more in the same block.
0348	TOOL CHANGE Z AXIS POS NOT ESTABLISHED	A tool change spindle on the Z-axis is not set.
0349	TOOL CHANGE SPINDLE NOT STOP	A tool change spindle stop is not stopped.
0350	PARAMETER OF THE INDEX OF THE SYNCHRONOUS CONTROL AXIS SET ERROR.	T An illegal synchronous control axis number (parameter No. 8180) is set.
0351	BECAUSE THE AXIS IS MOVING, THE SYNC CONTROL IS CAN'T BE USED.	T While the axis being subject to synchronous control was moving, an attempt was made to start or cancel the synchronous control by a synchronous control axis selection signal.
0352	SYNCHRONOUS CONTROL AXIS COMPOSITION ERROR.	T This error occurred when: 1) An attempt was made to perform synchronous control for the axis during a synchronization, composite, or superimposed control. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation. 3) An attempt was made to operate synchronous control although a parent-child-grandchild relation was not set.

Number	Message	Description
0353	THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.	T This error occurred when: 1) A move command was executed for an axis for which bit 7 (NUMx) of parameter No. 8163 was 1. 2) A move command was executed for a slave axis in synchronous control. 3) A move command was executed for an axis for which bit 7 (MUMx) of parameter No. 8162 was 1 in composite control.
0354	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE	T This error occurred when G28 was specified to the master axis being parking during synchronous control, but an axis reference position is not set for the slave axis.
0355	PARAMETER OF THE INDEX OF THE COMPOSITE CONTROL AXIS SET ERROR.	T An illegal composite control axis number (parameter No. 8183) is specified.
0356	BECAUSE THE AXIS IS MOVING, THE COMP CONTROL IS CAN'T BE USED.	T While the axis being subject to composite control was moving, an attempt was made to start or cancel the composite control by a composite control axis selection signal.
0357	COMPOSITE CONTROL AXIS COMPOSITION ERROR.	T This error occurred when an attempt was made to perform composite control for the axis during a synchronous, composite, or superimposed control.
0359	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE	T This error occurred when G28 was specified to the composite axis during composite control, but a reference position is not set to the other part of the composition.
0360	PARAMETER OF THE INDEX OF THE SUPERPOS CONTROL AXIS SET ERROR.	T An illegal superimposed control axis number (parameter No. 8186) is specified.
0361	BECAUSE THE AXIS IS MOVING, THE SUPERPOS CONTROL IS CAN'T BE USED.	T While the axis being subject to superimposed control was moving, an attempt was made to start or cancel the superimposed control by a superimposed control axis selection signal.
0362	SUPERPOSITION CONTROL AXIS COMPOSITION ERROR.	T This error occurred when: 1) An attempt was made to perform superimposed control for the axis during a synchronous, composite, or superimposed control. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation.
0363	THE G28 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	T This error occurred when G28 was specified to the superimposed control slave axis during superimposed control.
0364	THE G53 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	T This error occurred when G53 was specified to the slave axis being moved during superimposed control.
0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The number of controlled axes or spindles to be used in one path is not set correctly. Check parameters No. 981 and No. 982. If this alarm is generated, the emergency stop state cannot be released.
0369	G31 FORMAT ERROR	1) No axis is specified or two or more axes are specified in the torque limit switch instruction (G31P98/P99). 2) G31P90 cannot be specified.

Number	Message	Description
0370	G31P/G04Q ERROR	1) The specified address P value for G31 is out of range. The address P range is 1 to 4 in a multistage skip function. 2) The specified address Q value for G04 is out of range. The address Q range is 1 to 4 in a multistage skip function. 3) P1-4 for G31, or Q1-4 for G04 was commanded without a multistage skip function option. T 4) In G72 or G74 in grinding canned cycles, the specified address P value is out of range. Address P ranges from 1 to 4 in the multistage skip function. P1-4 was specified in G72 or G74 even though the multistage skip function option is not present.
0372	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the orthogonal axis.
0373	ILLEGAL HIGH-SPEED SKIP SIGNAL	In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), the same high-speed signal is selected in different paths.
0375	CAN NOT ANGULAR CONTROL(SYNC:MIX:OVL)	Angular axis control is disabled for this axis configuration. 1) All related axes in angular axis control are not in synchronous control mode. Alternatively, settings must be made to provide synchronous control between angular axes, and also between orthogonal axes. 2) All related axes in angular axis control are not in composite control mode. Alternatively, settings must be made to provide composite control between angular axes, and also between orthogonal axes. 3) The related axes in angular axis control are in superimposed control mode.
0376	SERIAL DCL: ILLEGAL PARAMETER	1) When bit 1 of parameter No. 1815 is set to “1”, bit 3 of parameter No. 2002 is set to “0” 2) The absolute-position detection function is enabled. (Bit 5 (APCx) of parameter No.1815 is set to “1”.)
0412	ILLEGAL G CODE	An unusable G code was used.
0445	ILLEGAL AXIS OPERATION	T The positioning command was issued in the speed control mode. Check the SV speed control mode signal (Fn521).
0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	T G96.1, G96.2, G96.3, and G96.4 are specified in the block that includes other commands. Modify the program.
0447	ILLEGAL SETTING DATA	The spindle controlled with the servo motor is not set correctly. Check the parameters for the function of spindle control with the servo motor.
0455	ILLEGAL COMMAND IN GRINDING	In grinding canned cycles: M 1) The signs of the I, J, and K commands do not match. 2) The amount of travel of the grinding axis is not specified.

Number	Message	Description
0456	ILLEGAL PARAMETER IN GRINDING	Parameters related to grinding canned cycles are incorrectly set. Probable causes are given below. 1) The axis number of the grinding axis is incorrectly set (parameters Nos. 5176 to 5179). **M** 2) The axis number of the dressing axis is incorrectly set (parameters Nos. 5180 to 5183). 3) The axis numbers of the cut axis, grinding axis, and dressing axis (only for the M series) overlap.
0601	ILLEGAL AXIS OPERATION FOR SERVO MOTOR SPINDLE	A move command is executer for the spindle controlled with the servomotor. Modify the program.
0602	ILLEGAL AXIS OPERATION FOR LIVE TOOL AXIS	The spindle controlled with the serve motor is not selected correctly.
1001	AXIS CONTROL MODE ILLEGAL	The axis control mode is illegal.
1013	ILLEGAL POS. OF PROGRAM NO.	Address O or N is specified where it must not (After the macro statement etc.).
1014	ILLEGAL FORMAT OF PROGRAM NO.	Address O or N is not followed by a number.
1016	EOB NOT FOUND	EOB (End of Block) code is missing at the end of a program input in the MDI mode.
1077	PROGRAM IN USE	An attempt was made in the foreground to execute a program being edited in the background. The currently edited program cannot be executed, so end editing and restart program execution.
1079	PROGRAM FILE NOT FOUND	The program of the specified file No. is not registered in an external device. (external device subprogram call)
1080	DUPLICATE DEVICE SUB PROGRAM CALL	Another external device subprogram call was made from a subprogram after the subprogram called by the external device subprogram call.
1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	The external device subprogram call is not possible in this mode.
1091	DUPLICATE SUB-CALL WORD	More than one subprogram call instruction was specified in the same block.
1092	DUPLICATE MACRO-CALL WORD	More than one macro call instruction was specified in the same block.
1093	DUPLICATE NC-WORD & M99	An address other than O, N, P or L was specified in the same block as M99 during the macro modal call state.
1095	TOO MANY TYPE-2 ARGUMENT	More than ten sets of I, J and K arguments were specified in the type–II arguments (A, B, C, I, J, K, I, J, K, ...) for custom macros.
1096	ILLEGAL VARIABLE NAME	An illegal variable name was specified. A code that cannot be specified as a variable name was specified. The command of [#_OFSxx] does not match the type (A or C) of tool compensation memory in current use.
1097	TOO LONG VARIABLE NAME	The specified variable name is too long.
1098	NO VARIABLE NAME	The specified variable name cannot be used as it is not registered.
1099	ILLLEGAL SUFFIX [ ]	A suffix was not specified to a variable name that required a suffix enclosed by [ ]. A suffix was specified to a variable name that did not require a suffix enclosed by [ ]. The value enclosed by the specified [ ] was out of range.
1100	CANCEL WITHOUT MODAL CALL	Call mode cancel (G67) was specified even though macro continuous–state call mode (G66) was not in effect.

Number	Message	Description
1101	ILLEGAL CNC STATEMENT IRT.	An interrupt was made in a state where a custom macro interrupt containing a move instruction could not be executed.
1115	READ PROTECTED VARIABLE	An attempt was made in a custom macro to use on the right side of an expression a variable that can only be used on the left side of an expression.
1120	ILLEGAL ARGUMENT FORMAT	The specified argument in the argument function (ATAN, POW) is in error.
1124	MISSING DO STATEMENT	The DO instruction corresponding to the END instruction was missing in a custom macro.
1125	ILLEGAL EXPRESSION FORMAT	The description of the expression in a custom macro statement contains an error. A parameter program format error. The screen displayed to enter periodic maintenance data or item selection menu (machine) data does not match the data type.
1128	SEQUENCE NUMBER OUT OF RANGE	The jump destination sequence No. in a custom macro statement GOTO instruction was out of range (valid range: 1 to 99999).
1131	MISSING OPEN BRACKET	The number of left brackets ([) is less than the number of right brackets (]) in a custom macro statement.
1132	MISSING CLOSE BRACKET	The number of right brackets (]) is less than the number of left brackets ([) in a custom macro statement.
1133	MISSING '='	An equal sign (=) is missing in the arithmetic calculation instruction in a custom macro statement.
1134	MISSING ','	A delimiter (,) is missing in a custom macro statement.
1137	IF STATEMENT FORMAT ERROR	The format used in the IF statement in a custom macro is in error.
1138	WHILE STATEMENT FORMAT ERROR	The format used in the WHILE statement in a custom macro is in error.
1139	SETVN STATEMENT FORMAT ERROR	The format used in the SETVN statement in a custom macro is in error.
1141	ILLEGAL CHARACTER IN VAR. NAME	The SETVN statement in a custom macro contacts a character that cannot be used in a variable name.
1142	TOO LONG V-NAME (SETVN)	The variable name used in a SETVN statement in a custom macro exceeds 8 characters.
1143	BPRNT/DPRNT STATEMENT FORMAT ERROR	The format used in the BPRINT statement or DPRINT statement is in error.
1144	G10 FORMAT ERROR	Data input for the L No. of G10 or the corresponding function is not enabled. Data setting address P or R is not specified. An address not relating to the data setting is specified. Which address to specify varies according to the L No. The sign, decimal point or range of the specified address are in error.
1160	COMMAND DATA OVERFLOW	An overflow occurred in the position data within the CNC. This alarm is also generated if the target position of a command exceeds the maximum stroke as a result of calculation such as coordinate conversion, offset, or introduction of a manual intervention amount.
1180	ALL PARALLEL AXES IN PARKING	T All of the axis specified for automatic operation are parked.
1196	ILLEGAL DRILLING AXIS SELECTED	An illegal axis was specified for drilling in a canned cycle for drilling. In the G code command block in a canned cycle, a Z point is not specified for the drilling axis.

Number	Message	Description
1200	PULSCODER INVALID ZERO RETURN	The grid position could not be calculated during grid reference position return using the grid system as the one–revolution signal was not received before leaving the deceleration dog. This alarm is also generated when the tool does not reach a feedrate that exceeds the servo error amount preset to parameter No. 1836 before the deceleration limit switch is left (deceleration signal *DEC returns to “1”).
1202	NO F COMMAND AT G93	M F codes in the inverse time specification mode (G93) are not handled as modal, and must be specified in individual blocks.
1223	ILLEGAL SPINDLE SELECT	An attempt was made to execute an instruction that uses the spindle although the spindle to be controlled has not been set correctly.
1298	ILLEGAL INCH/METRIC CONVERSION	An error occurred during inch/metric switching.
1300	ILLEGAL ADDRESS	The axis No. address was specified even though the parameter is not an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Axis No. cannot be specified in pitch error compensation data.
1301	MISSING ADDRESS	The axis No. was not specified even though the parameter is an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Or, data No. address N, or setting data address P or R are not specified.
1302	ILLEGAL DATA NUMBER	A non–existent data No. was found while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. This alarm is also generated when illegal word values are found.
1303	ILLEGAL AXIS NUMBER	An axis No. address exceeding the maximum number of controlled axes was found while loading parameters from a tape or by entry of the G10 parameter.
1304	TOO MANY DIGIT	Data with too many digits was found while loading parameters or pitch error compensation data from a tape.
1305	DATA OUT OF RANGE	Out–of–range data was found while loading parameters or pitch error compensation data from a tape. The values of the data setting addresses corresponding to L Nos. during data input by G10 was out of range. This alarm is also generated when NC programming words contain out–of–range values.
1306	MISSING AXIS NUMBER	A parameter which requires an axis to be specified was found without an axis No. (address A) while loading parameters from a tape.
1307	ILLEGAL USE OF MINUS SIGN	Data with an illegal sign was found while loading parameters or pitch error compensation data from a tape, or by entry of the G10 parameter. A sign was specified to an address that does not support the use of signs.
1308	MISSING DATA	An address not followed by a numeric value was found while loading parameters or pitch error compensation data from a tape.
1329	ILLEGAL MACHINE GROUP NUMBER	An machine group No. address exceeding the maximum number of controlled machine groups was found while loading parameters from a tape or by entry of the G10 parameter.

Number	Message	Description
1330	ILLEGAL SPINDLE NUMBER	An spindle No. address exceeding the maximum number of controlled spindles was found while loading parameters from a tape or by entry of the G10 parameter.
1331	ILLEGAL PATH NUMBER	An path No. address exceeding the maximum number of controlled path was found while loading parameters from a tape or by entry of the G10 parameter.
1332	DATA WRITE LOCK ERROR	Could not load data while loading parameters, pitch error compensation data and work coordinate data from tape.
1333	DATA WRITE ERROR	Could not write data while loading data from tape.
1470	G40.1 –G42.1 PARAMETER MISS	M A parameter setting related to normal direction control is illegal. The axis number of a normal direction controlled axis is set in parameter No. 5480, but that axis number is in the range of the number of controlled axes. The axis set as a normal direction controlled axis is not set as a rotation axis (ROTx, bit 0 of parameter No. 1006) = 1 and No.1022=0). Set the feedrate at which to insert rotation about a normal direction controlled axis in parameter No. 5481, in the range of 1 to 15000 mm/min.
1508	DUPLICATE M-CODE (INDEX TABLE REVERSING)	M A function to which the same code as this M code is set exists. (index table indexing)
1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	T A function to which the same code as this M code is set exists. (spindle positioning, orientation)
1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	T A function to which the same code as this M code is set exists. (spindle positioning, positioning)
1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	T A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
1533	ADDRESS F UNDERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too slow in the feed per single rotation mode.
1534	ADDRESS F OVERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too fast in the feed per single rotation mode.
1537	ADDRESS F UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too slow.
1538	ADDRESS F OVERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too fast.
1541	S-CODE ZERO	“0” has been instructed as the S code.
1543	ILLEGAL GEAR SETTING	T The gear ratio between the spindle and position coder, or the set position coder number of pulses is illegal in the spindle positioning function.
1544	S-CODE OVER MAX	The S command exceeds the maximum spindle rotation number.
1548	ILLGAL AXIS MODE	The spindle positioning (T series) axis/Cs contour control axis was specified during switching of the controlled axis mode.
1561	ILLEGAL INDEXING ANGLE	M The specified angle of rotation is not an integer multiple of the minimum indexing angle.
1564	INDEX TABLE AXIS – OTHER AXIS SAME TIME	M The index table indexing axis and another axis have been specified in the same block.

Number	Message	Description
1567	INDEX TABLE AXIS DUPLICATE AXIS COMMAND	M Index table indexing was specified during axis movement or on an axis for which the index table indexing sequence was not completed.
1590	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
1591	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting bit 0 (TVC) of parameter No. 0000 to “0”.
1592	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read. For the program restart function, this alarm is generated if a specified block is not found.
1593	EGB PARAMETER SETTING ERROR	M Error in setting a parameter related to the EGB 1) The setting of SYN, bit 0 of parameter No. 2011, is not correct. 2) The slave axis specified with G81 is not set as a rotation axis. (ROT, bit 0 of parameter No. 1006) 3) Number of pulses per rotation (Parameter (No. 7772 or No. 7773) is not set.)
1594	EGB FORMAT ERROR	M Error in the format of the block of an EGB command 1) T (number of teeth) is not specified in the G81 block. 2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. 3) n the G81 block, only one of P and Q is specified.
1595	ILL-COMMAND IN EGB MODE	M During synchronization with the EGB, a command that must not be issued is issued. 1) Slave axis command using G27, G28, G29, G30, G33, G53, etc. 2) Inch/metric conversion command using G20, G21, etc.
1596	EGB OVERFLOW	M An overflow occurred in the calculation of the synchronization coefficient.
1805	ILLEGAL COMMAND	[I/O Device] An attempt was made to specify an illegal command during I/O processing on an I/O device. [G30 Reference Position Return] The address P numbers for specifying the 2nd, 3rd, and 4th reference position returns are not 2, 3, and 4. [Single Rotation Dwell] The specified spindle rotation is “0” when single rotation dwell is specified.
1806	DEVICE TYPE MISS MATCH	An operation not possible on the I/O device that is currently selected in the setting was specified. This alarm is also generated when file rewind is instructed even though the I/O device is not a FANUC Cassette.
1807	PARAMETER SETTING ERROR	An illegal I/O interface is specified. The external I/O device and baud rate, stop bit and protocol selection settings are erroneous.

Number	Message	Description
1808	DEVICE DOUBLE OPENED	An attempt was made to open a device that is being accessed.
1820	ILLEGAL DI SIGNAL STATE	1) An each axis workpiece coordinate system preset signal was turned “1” in the state in which all axes on the path including the axis on which to perform preset with the each axis workpiece coordinate system were not stopped or in which a command was in execution. 2) When the M code for performing preset with an each axis workpiece coordinate system preset signal is specified, the each axis workpiece coordinate system preset signal is not input. 3) The auxiliary function lock is enabled.
1823	FRAMING ERROR(1)	The stop bit of the character received from the I/O device connected to reader/punch interface 1 was not detected.
1830	DR OFF(2)	The data set ready input signal DR of the I/O device connected to reader/punch interface 2 turned OFF.
1832	OVERRUN ERROR(2)	The next character was received from the I/O device connected to reader/punch interface 2 before it could read a previously received character.
1833	FRAMING ERROR(2)	The stop bit of the character received from the I/O device connected to reader/punch interface 2 was not detected.
1834	BUFFER OVERFLOW(2)	The NC received more than 10 characters of data from the I/O device connected to reader/punch interface 2 even though the NC sent a stop code (DC3) during data reception.
1912	V-DEVICE DRIVER ERROR (OPEN)	An error occurred during device driver control.
1960	ACCESS ERROR (MEMORY CARD)	Illegal memory card accessing This alarm is also generated during reading when reading is executed up to the end of the file without detection of the EOR code.
1961	NOT READY (MEMORY CARD)	The memory card is not ready.
1962	CARD FULL (MEMORY CARD)	The memory card has run out of space.
1963	CARD PROTECTED (MEMORY CARD)	The memory card is write–protected.
1964	NOT MOUNTED (MEMORY CARD)	The memory card could not be mounted.
1965	DIRECTORY FULL (MEMORY CARD)	The file could not be generated in the root directory for the memory card.
1966	FILE NOT FOUND (MEMORY CARD)	The specified file could not be found on the memory card.
1967	FILE PROTECTED (MEMORY CARD)	The memory card is write–protected.
1968	ILLEGAL FILE NAME (MEMORY CARD)	Illegal memory card file name
1969	ILLEGAL FORMAT (MEMORY CARD)	Check the file name.
1970	ILLEGAL CARD (MEMORY CARD)	This memory card cannot be handled.
1971	ERASE ERROR (MEMORY CARD)	An error occurred during memory card erase.
1972	BATTERY LOW (MEMORY CARD)	The memory card battery is low.
1973	FILE ALREADY EXIST	A file having the same name already exists on the memory card.
2032	EMBEDDED ETHERNET/DATA SERVER ERROR	An error was returned in the built-in Ethernet/data server function. For details, see the error message screen of the built-in Ethernet or data server.
2051	#200-#499ILLEGAL P-CODE MACRO COMMON INPUT(NO OPTION)	An attempt was made to enter a custom macro common variable not existing in the system.

Number	Message	Description
2052	#500-#549P-CODE MACRO COMMON SELECT(CANNOT USE SETVN)	The variable name cannot be entered. The SETVN command cannot be used with the P-CODE macro common variables #500 to #549.
2053	P-CODE VARIABLE NUMBER IS OUTSIDE OF RANGE	An attempt was made to enter a P-CODE-only variable not existing in the system.
2054	EXTENDED P-CODE VARIABLE NUMBER IS OUTSIDE OF RANGE	An attempt was made to enter an extended P-CODE-only variable not existing in the system.
4010	ILLEGAL REAL VALUE OF OBUF :	The real value for a output buffer is in error.
5006	TOO MANY WORD IN ONE BLOCK	The number of words in a block exceeds the maximum. The maximum is 26 words. However, this figure varies according to NC options. Divide the instruction word into two blocks.
5007	TOO LARGE DISTANCE	Due to compensation, point of intersection calculation, interpolation or similar reasons, a movement distance that exceeds the maximum permissible distance was specified. Check the programmed coordinates or compensation amounts.
5009	PARAMETER ZERO (DRY RUN)	The dry run feedrate parameter No. 1410 or maximum cutting feedrate parameter No. 1430 for each axis has been set to 0.
5010	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read.
5011	PARAMETER ZERO (CUT MAX)	The maximum cutting feedrate parameter No. 1430 has been set to 0.
5014	TRACE DATA NOT FOUND	A transfer could not be made because of no trace data.
5016	ILLEGAL COMBINATION OF M CODES	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5018	POLYGON SPINDLE SPEED ERROR	T In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained. For polygon turning between spindles: More information as to why this alarm occurred is indicated in DGN No. 471.
5020	PARAMETER OF RESTART ERROR	An invalid value is set in parameter No. 7310, which specifies the axis order in which the tool is moved along axes to the machining restart position in dry run. A value ranging from 1 to the number of controlled axes may be set in this parameter.
5046	ILLEGAL PARAMETER (S-COMP)	M The setting of a parameter related to simple straightness compensation contains an error. Possible causes include: 1) A non-existent axis number is set in a moving or compensation axis parameter. 2) The simple straightness compensation point numbers do not have correct magnitude relationships. 3) No simple straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. 4) The compensation per compensation point is either too large or too small.
5064	DIFFERRENT AXIS UNIT	Circular interpolation has been specified on a plane consisting of axes having different increment systems.

Number	Message	Description
5065	DIFFERRENT AXIS UNIT(PMC AXIS)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5110	IMPROPER G-CODE (AICC MODE)	An unspecifiable G code was specified in the advanced preview control, AI advanced preview control, or AI contour control mode.
5131	NC COMMAND IS NOT COMPATIBLE	T The PMC axis control and polar coordinate interpolation were specified simultaneously.
5195	DIRECTION CAN NOT BE JUDGED	T Measurement is invalid in the tool compensation measurement value direct input B function. [For 1-contact input] 1) The recorded pulse direction is not constant. For example, the stopped state may be set during offset write mode, the servo off state may be entered, or the direction may vary. 2) The tool is moving along the two axes (X-axis and Y-axis). [For movement direction judgment in 4-contact input] 1) The recorded pulse direction is not constant. For example, the stopped state may be set during offset write mode, the servo off state may be entered, or the direction may vary. 2) The tool is moving along the two axes (X-axis and Z-axis). 3) The direction indicated by the tool compensation write signal does not match the movement direction of the axis.
5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting bit 2 of parameter No.1819 is set to "1". Move the machine to reference position by manual operation and execute manual reference return.
5257	G41/G42 NOT ALLOWED IN MDI MODE	Tool radius/tool nose radius compensation was specified in MDI mode. (Depending on the setting of the bit 4 (MCR) of parameter No. 5008)
5303	TOUCH PANEL ERROR	The touch panel is not connected correctly, or the touch panel cannot be initialized when the power is turned on. Correct the cause then turn on the power again.

Number	Message	Description
5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple spindle control, 1) Address P is not specified. 2) A P code for selecting a spindle is not set in parameter No. 3781. 3) An illegal G code which cannot be commanded with an S_P_; command is specified. 4) Multi-spindle control is not enabled because bit 1 (EMS) of parameter No. 3702 is 1. 5) The spindle amplifier number of each spindle is not set in parameter No. 3717. 6) A spindle command is executed from a path in which this command is prohibited (parameter No. 11090). 7) The setting of parameter No. 11090 is incorrect.
5306	MODE CHANGE ERROR	A mode switchover failed at the time of activation. An attempt to activate a one-touch macro was made while not in the reset state or during a reset or emergency stop.
5329	M98 AND NC COMMAND IN SAME BLOCK	A subprogram call which is not a single block was commanded during canned cycle mode.
5339	ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.	T 1. The value of P, Q, or L specified by G51.4/G50.4/G51.5/G50.5/G51.6/G50.6 is invalid. 2. A duplicate value is specified by parameter No. 12600.
5346	RETURN TO REFERENCE POINT	The coordinate establishment of the Cs contour control axis is not made. Perform a manual reference position return. 1) When Cs coordinate establishment is made for the Cs-axis for which the Cs-axis reference position status signal CSPENx is 0 2) When positional information is not sent from the spindle amplifier 3) When the servo off state is entered during the start of Cs-axis coordinate establishment 4) When the emergency stop state is entered during Cs-axis coordinate establishment T 5) When the Cs-axis is under synchronous or superimposed control 6) When an attempt is made to release composite control for the Cs-axis during Cs-axis coordinate establishment 7) When an attempt is made to start synchronous, composite, or superimposed control for the Cs-axis during Cs-axis coordinate establishment
5362	CONVERT INCH/MM AT REF-POS	An inch/metric conversion was performed at a position other than the reference position. Perform an inch/metric conversion after returning to the reference position.
5391	CAN NOT USE G92	M Workpiece coordinate system setting G92 cannot be specified. 1) After tool length compensation was changed by tool length compensation shift type, G92 was specified when no absolute command is present. 2) G92 was specified in the block in which G49 is present.

Number	Message	Description
5395	CS AXIS NUMBER OVER	The number of axes to be set for Cs-axis contour control exceeds the maximum number allowed in the system. Check parameter No. 1023. If this alarm is generated, the emergency stop state cannot be released.
5445	CAN NOT COMMAND MOTION IN G39	Corner circular interpolation (G39) of tool radius/tool nose radius compensation is not specified alone but is specified with a move command.
5446	NO AVOIDANCE AT G41/G42	Because there is no interference avoidance vector, the interference check avoidance function of tool radius/tool nose radius compensation does not work.
5447	DANGEROUS AVOIDANCE AT G41/G42	The interference check avoidance function of tool radius/tool nose radius compensation operation will lead to danger.
5448	INTERFERENCE TO AVD. AT G41/G42	In the interference check avoidance function of tool radius/tool nose radius compensation, a further interference occurs for an already created interference avoidance vector.

(4) Parameter writing alarm (SW alarm)

Number	Message	Description
SW0100	PARAMETER ENABLE SWITCH ON	The parameter setting is enabled (bit 0 (PWE) of parameter No. 8900 is set to “1”). To set the parameter, turn this parameter ON. Otherwise, set to OFF.

(5) Servo alarms (SV alarm)

Number	Message	Description
SV0001	SYNC ALIGNMENT ERROR	In feed axis control , the amount of compensation for synchronization exceeded the parameter (No. 8325) setting value. This alarm occurs only for a slave axis.
SV0002	SYNC EXCESS ERROR ALARM 2	In feed axis control , the amount of synchronization error exceeded the parameter (No. 8332) setting value. When the synchronization is not completed after power-up, the determination is made by the parameter value (No. 8332) multiplied by the parameter (No. 8330) multiplier. This alarm occurs only for a slave axis only.
SV0003	SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL MODE CAN'T BE CONTINUED	T Since as axis in synchronization, composite, or superimposed mode caused a servo alarm, the mode could not be continued, If one of the axes in a mode causes a servo alarm, all axes relating to the axis enter the servo-off state. This alarm is generated to enable the cause of the servo-off state to be checked.
SV0004	EXCESS ERROR (G31)	The amount of positional deviation during torque limit skip command operation exceeded the limit value of the parameter No.6287.
SV0005	SYNC EXCESS ERROR (MCN)	In feed axis control , for synchronization, the difference value of the machine coordinate between a master and slave axes exceeded the parameter (No. 8314) setting value. This alarm occurs for a master or slave axis.
SV0006	ILLEGAL TANDEM AXIS	For the slave axis under tandem control, absolute position detection is set (parameter bit 5 (APC) of parameter No. 1815 = 1).

Number	Message	Description
SV0007	SV ALM ANOTHER PATH(MULTI AMP.)	T When a multiaxis amplifier was used in a 2-path system across paths, a servo alarm occurred on an axis belonging to another path. When a 2-path system and multiple servo axes between paths are controlled by a multiaxis amplifier, if a servo alarm occurs on an axis belonging to another path of the same amplifier, the MCC of the amplifier goes down and SV0401 V-READY OFF occurs on an axis belonging to the local path in the same amplifier. Since SV0401 is caused by a servo alarm occurred on an axis in another path, SV0007 is caused together to clearly indicate the fact. The axis belonging to another path in the same amplifier resolves the cause of the servo alarm.
SV0301	APC ALARM: COMMUNICATION ERROR	Since the absolute-position detector caused a communication error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0302	APC ALARM: OVER TIME ERROR	Since the absolute-position detector caused an overtime error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0303	APC ALARM: FRAMING ERROR	Since the absolute-position detector caused a framing error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0304	APC ALARM: PARITY ERROR	Since the absolute-position detector caused a parity error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0305	APC ALARM: PULSE ERROR	Since the absolute-position detector caused a pulse error, the correct machine position could not be obtained. The absolute-position detector, or cable is thought to be defective.
SV0306	APC ALARM: OVER FLOW ERROR	Since the amount of positional deviation overflowed, the correct machine position could not be obtained. Check to see the parameter No. 2084 or No. 2085.
SV0307	APC ALARM: MOVEMENT EXCESS ERROR	Since the machine moved excessively, the correct machine position could not be obtained.
SV0360	ABNORMAL CHECKSUM(INT)	The checksum alarm occurred on the built–in Pulsecoder.
SV0361	ABNORMAL PHASE DATA(INT)	The phase data abnormal alarm occurred on the built–in Pulsecoder.
SV0362	ABNORMAL REV. DATA(INT)	The speed count abnormal alarm occurred on the built-in Pulsecoder.
SV0363	ABNORMAL CLOCK(INT)	The clock alarm occurred on the built–in Pulsecoder.
SV0364	SOFT PHASE ALARM(INT)	A digital servo soft detected an abnormality on the built in Pulsecoder.
SV0365	BROKEN LED(INT)	The digital servo software detected abnormal data on the built–in Pulsecoder.
SV0366	PULSE MISS(INT)	A pulse error occurred on the built–in Pulsecoder.
SV0367	COUNT MISS(INT)	A count error occurred on the built–in Pulsecoder.

Number	Message	Description
SV0368	SERIAL DATA ERROR(INT)	The communications data could not be received from the built–in Pulsecoder.
SV0369	DATA TRANS. ERROR(INT)	A CRC error or stop bit error occurred in the communications data from the built–in Pulsecoder.
SV0380	BROKEN LED(EXT)	Separate detector error
SV0381	ABNORMAL PHASE (EXT)	An abnormal alarm in the position data occurred on the separate detector.
SV0382	COUNT MISS(EXT)	A count error occurred on the separate detector.
SV0383	PULSE MISS(EXT)	A pulse error occurred on the separate detector.
SV0384	SOFT PHASE ALARM(EXT)	The digital servo software detected abnormal data on the separate detector.
SV0385	SERIAL DATA ERROR(EXT)	The communications data could not be received from the separate detector.
SV0386	DATA TRANS. ERROR(EXT)	A CRC error or stop bit error occurred in the communications data from the standalone detector.
SV0387	ABNORMAL ENCODER(EXT)	An abnormality occurred on a separate detector. For more information, contact the scale manufacturer.
SV0401	IMPROPER V_READY OFF	Although the ready signal (PRDY) of the position control was ON, the ready signal (VRDY) of the velocity control was OFF.
SV0403	CARD/SOFT MISMATCH	The combination of the axis card and the servo software is incorrect. Probable causes are given below. 1) The correct axis card is not attached. 2) The correct servo software is not installed in flash memory.
SV0404	IMPROPER V_READY ON	Although the ready signal (PRDY) of the position control was OFF, the ready signal (VRDY) of the velocity control was ON.
SV0407	EXCESS ERROR	T The difference value of the amount of positional deviation for the synchronization axis exceeded the setting value. (during synchronous control only)
SV0409	DETECT ABNORMAL TORQUE	An abnormal load was detected on the servo motor, during Cs axis, or spindle positioning (T series) axis. The alarm can be canceled by RESET.
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter (No. 1829) setting value.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter (No.1828) setting value.
SV0413	LSI OVERFLOW	The counter for the amount of positional deviation overflowed
SV0415	MOTION VALUE OVERFLOW	The velocity exceeding the travel velocity limit was commanded.

Number	Message	Description
SV0417	ILL DGTL SERVO PARAMETER	A digital serve parameter setting is incorrect. [When bit 4 of diagnosis information No. 203 is 1.] An illegal parameter was detected by the servo software. Identify the cause with reference to diagnosis information No. 352. [When bit 4 of diagnosis information No. 203 is 0.] The CNC software detected an illegal parameter. Probable causes are given below (see diagnosis information No. 280). 1) The value specified in parameter No. 2020 as the motor model falls outside the specified range. 2) The motor rotation direction in parameter No. 2022 is not set to a correct value (111 or -111). 3) The speed feedback pulse count per motor rotation in parameter No. 2023 is set to a negative or other incorrect value. 4) The position feedback pulse count per motor rotation in parameter No. 2024 is set to a negative or other incorrect value.
SV0420	SYNC TORQUE EXCESS	In feed axis control , for synchronization, the difference value of torque between a master and slave axes exceeded the parameter (No. 2031) setting value. This alarm occurs for a master axis.
SV0421	EXCESS ERROR(SEMI-FULL)	The difference between the feedback from the semi and full sides exceeded the setting of parameter No.2118.
SV0422	EXCESS VELOCITY IN TORQUE	In torque control, the commanded permissible velocity was exceeded.
SV0423	EXCESS ERROR IN TORQUE	In torque control, the total permissible move value specified as a parameter was exceeded.
SV0430	SV MOTOR OVERHEAT	The servo motor has overheated.
SV0431	CNV. OVERLOAD	Power Supply (PS) : Overheat Servo Amplifier : Overheat
SV0432	CNV. LOW VOLT CONTROL	Power Supply (PS) : The control power supply voltage has dropped. Servo Amplifier : The control power supply voltage has dropped.
SV0433	CNV. LOW VOLT DC LINK	Power Supply (PS) : Low DC link voltage Servo Amplifier : Low DC link voltage
SV0434	INV. LOW VOLT CONTROL	Servo Amplifier : Low control power voltage
SV0435	INV. LOW VOLT DC LINK	Servo Amplifier : Low DC link voltage
SV0436	SOFTTHERMAL(OVC)	The digital servo software detected a software thermal (OVC).
SV0437	CNV. OVERCURRENT POWER	Power Supply (PS) : Overcurrent on input circuit section.
SV0438	INV. ABNORMAL CURRENT	Servo Amplifier : Motor overcurrent
SV0439	CNV. OVER VOLT DC LINK	Power Supply (PS) : The DC link voltage is too high. Servo Amplifier : The DC link voltage is too high.
SV0440	CNV. EX DECELERATION POW.	Power Supply (PS) : Excessive generative discharge Servo Amplifier : Excessive generative discharge, or abnormal error in generative power circuit
SV0441	ABNORMAL CURRENT OFFSET	The digital servo software detected an abnormality in the motor current detection circuit.
SV0442	CNV. CHARGE FAILURE	Power Supply (PS) : The spare charge circuit for the DC link is abnormal.
SV0443	CNV. COOLING FAN FAILURE	Power Supply (PS) : Internal cooling fan failure. Servo Amplifier : Internal cooling fan failure.

Number	Message	Description
SV0444	INV. COOLING FAN FAILURE	Servo Amplifier : Internal cooling fan failure.
SV0445	SOFT DISCONNECT ALARM	The digital servo software detected a disconnected Pulsecoder.
SV0446	HARD DISCONNECT ALARM	The hardware detected a disconnected built–in Pulsecoder.
SV0447	HARD DISCONNECT(EXT)	The hardware detected a disconnected separate detector.
SV0448	UNMATCHED FEEDBACK ALARM	The sign of the feedback signal from the standalone detector is opposite to that from the feedback signal from the built–on Pulsecoder.
SV0449	INV. IPM ALARM	Servo Amplifier : The IPM (Intelligent Power Module) detected an alarm.
SV0453	SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the α Pulsecoder. Turn off the power to the CNC, then remove and insert the Pulsecoder cable. If this alarm is issued again, replace the Pulsecoder.
SV0454	ILLEGAL ROTOR POS DETECT	The magnetic pole detection function terminated abnormally. The magnetic pole could not be detected because the motor did not run.
SV0456	ILLEGAL CURRENT LOOP	An attempt was made to set the current loop that could not be set. The amplifier pulse module in use does not comply with HIGH SPEED HRV. Or, requirements to control are not satisfied in the system.
SV0458	CURRENT LOOP ERROR	The specified current loop differs from the actual current loop.
SV0459	HI HRV SETTING ERROR	For two axes whose servo axis numbers (parameter No. 1023) are consecutively even and odd numbers, HIGH SPEED HRV control is possible for one axis and impossible for the other.
SV0460	FSSB DISCONNECT	The FSSB connection was discontinued. Probable causes are: 1) The FSSB connection cable was disconnected or broken. 2) The amplifier was turned off . 3) In the amplifier, the low-voltage alarm occurred.
SV0462	SEND CNC DATA FAILED	The correct data could not be received on a slave side because of the FSSB communication error.
SV0463	SEND SLAVE DATA FAILED	The correct data could not be received in the servo software side because of the FSSB communication error.
SV0465	READ ID DATA FAILED	A read of the ID information for the amplifier has failed at power-on.
SV0466	MOTOR/AMP. COMBINATION	The maximum current of an amplifier is different to that of a motor. Probable causes are: 1) The connection command for an amplifier is incorrect. 2) The parameter (No.2165) setting is incorrect
SV0468	HI HRV SETTING ERROR(AMP)	An attempt was made to set up HIGH SPEED HRV control for use when the controlled axis of an amplifier for which HIGH SPEED HRV control could not be used.
SV0600	INV. DC LINK OVER CURRENT	DC link overcurrent.
SV0601	INV. RADIATOR FAN FAILURE	External radiator cooling fan failure.
SV0602	INV. OVERHEAT	The servo motor has overheated.
SV0603	INV. IPM ALARM(OH)	The IPM (Intelligent Power Module) detected an overheat alarm.
SV0604	AMP. COMMUNICATION ERROR	The communication between Servo amplifier and Power Supply (PS) is in error.

Number	Message	Description
SV0605	CNV. EX. DISCHARGE POW.	Power Supply (PS) : The motor regenerative power is too much.
SV0606	CNV. RADIATOR FAN FAILURE	Power Supply (PS) : External radiator cooling fan failure.
SV0607	CNV. SINGLE PHASE FAILURE	Power Supply (PS) : The input power supply has a missing phase.
SV0646	ABNORMAL ANALOG SIGNAL(EXT)	An error occurred in the analog 1Vp-p output of the separate detector. The separate detector, cable, or separate detector interface unit may be failed.
SV1025	V_READY ON (INITIALIZING)	The ready signal (VRDY) of the velocity control which should be OFF is ON while the servo control is ON.
SV1026	ILLEGAL AXIS ARRANGE	The parameter for servo axis arrange is not set correctly. A negative value, duplicate value, or greater value than the number of control axes was set to the parameter No. 1023 "The servo axis number of each axis."
SV1055	ILLEGAL TANDEM AXIS	In tandem control, the setting of the parameter No. 1023 is incorrect.
SV1056	ILLEGAL TANDEM PAIR	In tandem control, the setting of the bit 6 (TDM) of parameter No.1817 is incorrect.
SV1067	FSSB:CONFIGURATION ERROR(SOFT)	An FSSB configuration error occurred (detected by software). The connected amplifier type is incompatible with the FSSB setting value.
SV1100	S-COMP. VALUE OVERFLOW	M The amount of compensation for the simple straightness exceeded a maximum value of 32767.
SV5134	FSSB:OPEN READY TIME OUT	In the initialization, the FSSB could not be in an open ready sate. The axis card is thought to be defective.
SV5136	FSSB:NUMBER OF AMP. IS INSUFFICIENT	The number of amplifier identified by the FSSB is insufficient than the number of control axes. Or, the setting of the number of axes or the amplifier connection is in error.
SV5137	FSSB:CONFIGURATION ERROR	An FSSB configuration error occurred. The connecting amplifier type is incompatible with the FSSB setting value.
SV5139	FSSB:ERROR	Servo initialization has not completed successfully. It is probable that an optical cable failed or a connection between the amplifier and another module failed.
SV5197	FSSB:OPEN TIME OUT	The initialization of the FSSB was completed, but it could not be opened. Or, the connection between the CNC and the amplifier in is incorrect.

(6) Overtravel alarms (OT alarm)

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	Exceeded the positive side stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	Exceeded the negative side stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	Exceeded the positive side stored stroke check 2. T Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	Exceeded the negative side stored stroke check 2. T Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the negative direction.
OT0504	+ OVERTRAVEL (SOFT 3)	Exceeded the positive side stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	Exceeded the - side stored stroke check 3.

Number	Message	Description
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0508	INTERFERENCE:+	T A tool moving in the positive direction along the n axis has fouled another tool post.
OT0509	INTERFERENCE:-	T A tool moving in the negative direction along the n axis has fouled another tool post.
OT0510	+ OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the negative direction during the stroke check before movement.
OT0511	- OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the positive direction during the stroke check before movement.

(7) Memory file alarms (IO alarm)

Number	Message	Description
IO1001	FILE ACCESS ERROR	The resident–type file system could not be accessed as an error occurred in the resident–type file system.
IO1002	FILE SYSTEM ERROR	The file could not be accessed as an error occurred in the CNC file system.
IO1030	CHECK SUM ERROR	The checksum of the CNC part program storage memory is incorrect.
IO1032	MEMORY ACCESS OVER RANGE	Accessing of data occurred outside the CNC part program storage memory range.
IO1104	OVER MAXIMUM TOOL LIFE PAIRS	The maximum number of tool life management pairs is exceeded. Modify the setting of the maximum number of tool life management pairs in parameter No. 6813.

(8) Alarms requiring power to be turned off (PW alarm)

Number	Message	Description
PW0000	POWER MUST BE OFF	A parameter was set for which the power must be turned OFF then ON again.
PW0001	X-ADDRESS(*DEC) IS NOT ASSIGNED.	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: 1) During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position.
PW0002	PMC address is not correct(AXIS).	The address to assign the axis signal is incorrect. This alarm may occur in the following case: 1) The parameter No.3021 setting is incorrect.

Number	Message	Description
PW0003	PMC address is not correct(SPINDLE).	The address to assign the spindle signal is incorrect. This alarm may occur in the following case: 1) The parameter No.3022 setting is incorrect.
PW0006	POWER MUST BE OFF (ILL-EXEC-CHK)	The malfunction prevention function detected an alarm to require the power off.
PW0007	X-ADDRESS(SKIP) IS NOT ASSIGNED	The X address of PMC could not be assigned correctly. Possible causes are: 1) During the set of parameter No. 3012, the skip signal of the X address was not assigned correctly. 2) During the set of parameter No. 3019, the address other than the skip signal of the X address was not assigned correctly.
PW1102	ILLEGAL PARAMETER (I-COMP.)	The parameter for setting slope compensation is incorrect. This alarm occurs in the following cases: 1) When the size relationship between the slope compensation point Nos. is incorrect 2) When the slope compensation point is not located between the most negative side and most positive side of pitch error compensation 3) When the compensation per compensation point is too small or too great.
PW1110	ILLEGAL PARAMETER (SERVO MOTOR SPINDLE)	The parameter for the spindle controlled with the servo motor is not set correctly.
PW1111	ILLEGAL SPINDLE NUMBER (SERVO MOTOR SPINDLE)	The spindle number (parameter No. 11010) or spindle amplifier number (parameter No. 3717) for the spindle controlled with the servo motor is not set correctly.
PW5046	ILLEGAL PARAMETER (S-COMP.)	M The parameter for setting simple straightness compensation is incorrect.

(9) Spindle alarms (SP alarm)

Number	Message	Description
SP0740	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the stopped spindle has exceeded the set value during rigid tapping.
SP0741	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the moving spindle has exceeded the set value during rigid tapping.
SP0742	RIGID TAP ALARM : LSI OVERFLOW	An LSI overflow has occurred for the spindle during rigid tapping.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning (T series), rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
SP0754	ABNORMAL TORQUE	An abnormal load was detected in a spindle motor. The alarm can be canceled by RESET.
SP1202	SPINDLE SELECT ERROR	In a multi spindle control, the spindle number other than the valid spindle number was selected by a position coder select signal. An attempt was made to select the spindle number of the system having no valid spindle.
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
SP1221	ILLEGAL MOTOR NUMBER	The spindle No. and the motor No. are incorrectly matched.
SP1224	ILLEGAL SPINDLE-POSITION CODER GEAR RATIO	The spindle–position coder gear ratio was incorrect.

Number	Message	Description
SP1225	CRC ERROR (SERIAL SPINDLE)	A CRC error (communications error) occurred in communications between the CNC and the serial spindle amplifier.
SP1226	FRAMING ERROR (SERIAL SPINDLE)	A framing error occurred in communications between the CNC and the serial spindle amplifier.
SP1227	RECEIVING ERROR (SERIAL SPINDLE)	A receive error occurred in communications between the CNC and the serial spindle amplifier.
SP1228	COMMUNICATION ERROR (SERIAL SPINDLE)	A communications error occurred between the CNC and the serial spindle amplifier.
SP1229	COMMUNICATION ERROR SERIAL SPINDLE AMP.	A communications error occurred between serial spindle amplifiers (motor Nos. 1 and 2, or motor Nos. 3–4).
SP1231	SPINDLE EXCESS ERROR (MOVING)	The position deviation during spindle rotation was greater than the value set in parameters.
SP1232	SPINDLE EXCESS ERROR (STOP)	The position deviation during spindle stop was greater than the value set in parameters.
SP1233	POSITION CODER OVERFLOW	The error counter/speed instruction value of the position coder overflowed.
SP1234	GRID SHIFT OVERFLOW	Grid shift overflowed.
SP1240	DISCONNECT POSITION CODER	The analog spindle position coder is broken.
SP1241	D/A CONVERTER ERROR	The D/A converter for controlling analog spindles is erroneous.
SP1243	ILLEGAL SPINDLE PARAMETER SETTING(GAIN)	The setting for the spindle position gain is incorrect.
SP1244	MOTION VALUE OVERFLOW	The amount of distribution to a spindle is too much
SP1245	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1246	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1247	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1969	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1970	SPINDLE CONTROL ERROR	Initialization of spindle control ended in error.
SP1971	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1972	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1974	ANALOG SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1975	ANALOG SPINDLE CONTROL ERROR	An position coder error was detected on the analog spindle.
SP1976	SERIAL SPINDLE COMMUNICATION ERROR	The amplifier No. could not be set to the serial spindle amplifier.
SP1977	SERIAL SPINDLE COMMUNICATION ERROR	An error occurred in the spindle control software.
SP1978	SERIAL SPINDLE COMMUNICATION ERROR	A time–out was detected during communications with the serial spindle amplifier.
SP1979	SERIAL SPINDLE COMMUNICATION ERROR	The communications sequence was no longer correct during communications with the serial spindle amplifier.
SP1980	SERIAL SPINDLE AMP. ERROR	Defective SIC–LSI on serial spindle amplifier
SP1981	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the analog spindle amplifier side.
SP1982	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the serial spindle amplifier side.
SP1983	SERIAL SPINDLE AMP. ERROR	Could not clear on the spindle amplifier side.
SP1984	SERIAL SPINDLE AMP. ERROR	An error occurred during re–initialization of the spindle amplifier.
SP1985	SERIAL SPINDLE CONTROL ERROR	Failed to automatically set parameters
SP1986	SERIAL SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1987	SERIAL SPINDLE CONTROL ERROR	Defective SIC–LSI on the CNC
SP1988	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1989	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.

Number	Message	Description
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Alternatively, the number of spindles exceeded the maximum number allowed in the system. Check to see the following parameter. (No.3701#1,#4, 3716, 3717)
SP1998	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1999	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.

Refer to A.3 for SP9001 or later.

(10) Overheat alarms (OH alarm)

Number	Message	Description
OH0700	LOCKER OVERHEAT	CNC cabinet overheat
OH0701	FAN MOTOR STOP	PCB cooling fan motor abnormality
OH0704	OVERHEAT	T Spindle overheat due to detection of changes in the spindle speed 1) When the cutting load is large, offload the cutting conditions. 2) Check if the cutting tool became dull. 3) Check if the spindle amplifier malfunctions.

(11) Other alarms (DS alarm)

Number	Message	Description
DS0001	SYNC EXCESS ERROR (POS DEV)	In feed axis control , the difference in the amount of positional deviation between the master and slave axes exceeded the parameter (No. 8323) setting value. This alarm occurs only for the slave axis.
DS0003	SYNCHRONIZE ADJUST MODE	The system is in the synchronize adjust mode.
DS0004	EXCESS MAXIMUM FEEDRATE	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
DS0005	EXCESS MAXIMUM ACCELERATION	The malfunction prevention function detected the command in which a value exceeding the maximum acceleration was specified.
DS0006	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0007	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0008	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0009	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0010	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0011	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0012	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0013	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0014	TOOL CHANGE DETECT MACHINE LOCK	A machine lock is turned on for the Z axis for which the tool is being changed.
DS0015	TOOL CHANGE DETECT MIRROR IMAGE	A mirror image is turned on for the Z axis for which the tool is being changed.

Number	Message	Description
DS0016	SERIAL DCL:FOLLOW-UP ERROR	1) The settings of parameters No.1883 and No.1884 fall outside the range. 2) The current position at establishment of the reference position subtracted by the distance between the reference positions (detection unit) exceeded ±2147483647. Change the current position or reference position to prevent this situation.
DS0017	SERIAL DCL:REF-POS ESTABLISH ERR	The travel amount at the FL speed at establishment of the reference position exceeded the setting of parameter No. 14010.
DS0018	SERIAL DCL:MISMATCH(SSYNC CTRL)	Of the master and slave axes for feed axis control, one axis is a linear scale with the origin and the other is not a linear scale with the reference position. In such a configuration, the feed axis control selection signal (SYNC<Gn138> or SYNCJ <Gn140>) needs to be set to 0 to establish the origin.
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the perpendicular axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the perpendicular axis.
DS0021	START ERROR(ONE TOUCH MACRO)	A macro program start operation cannot be accepted. 1) The feed hold signal *SP is 0. 2) An alarm is generated. 3) The SRN signal is 1.
DS0023	ILLEGAL PARAMETER (I-COMP VAL)	The setting of the inclination compensation parameter is incorrect. The compensation per compensation point is too large or too small.
DS0024	UINT SIGNAL WAS ILLEGALLY INPUT	An interruption custom macro was started during movement to the machining restart position at the dry run speed.
DS0025	G60 CANNOT BE EXECUTED	M The state of a mirror image is different between the time when look-ahead of a block for unidirectional positioning was performed and the time when execution of the block was started, so unidirectional positioning cannot be performed. Modify the program.
DS0026	MISMATCH OF ANGULAR AXIS(D.C.S)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.
DS0027	MISMATCH OF SYNCHRONOUS AXIS(D.C.S)	Master/slave axes of axis synchronous control, one of them is the linear scale with distance-coded reference marks, and the other of them is not the linear scale with distance-coded reference marks. Please establish reference position with the input signal SYNCn<G138>, SYNCJn<G140> or parameter setting to 0.

Number	Message	Description
DS0059	SPECIFIED NUMBER NOT FOUND	[External data I/O] The No. specified for a program No. or sequence No. search could not be found. There was an I/O request issued for offset (tool data), but either no tool numbers have been input since power ON or there is no data for the entered tool No. [External workpiece No. search] The program corresponding to the specified workpiece No. could not be found.
DS0131	TOO MANY MESSAGE	An attempt was made to display an external operator message or external alarm message, but five or more displays were required simultaneously.
DS0132	MESSAGE NUMBER NOT FOUND	An attempt to cancel an external operator message or external alarm message failed because the specified message number was not found.
DS0133	TOO LARGE NUMBER	A value other than 0 to 4095 was specified as the external operator message or the external alarm message number.
DS0300	APC ALARM: NEED REF RETURN	A setting to zero position for the absolute position detector (association with reference position and the counter value of the absolute position detector) is required. Perform the return to the reference position. This alarm may occur with other alarms simultaneously. In this case, other alarms must be handled first.
DS0306	APC ALARM: BATTERY VOLTAGE 0	The battery voltage of the absolute position detector has dropped to a level at which data can no longer be held. Or, the power was supplied to the Pulsecoder for the first time. If this problem recurs after the power is turned off and then back on, the battery or cable may be defective. Replace the battery with the machine turned on.
DS0307	APC ALARM: BATTERY LOW 1	The battery voltage of the absolute position detector has dropped to a level at which a replacement is required. Replace the battery with the machine turned on.
DS0308	APC ALARM: BATTERY LOW 2	The battery voltage of the absolute position detector dropped to a level at which a replacement was required in the past. (including during power off) Replace the battery with the machine turned on.
DS0309	APC ALARM: REF RETURN IMPOSSIBLE	An attempt was made to set the zero point for the absolute position detector by MDI operation when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
DS0405	ZERO RETURN END NOT ON REF	The axis specified in automatic zero return was not at the correct zero point when positioning was completed. Perform zero return from a point whose distance from the zero return start position to the zero point is 2 or more revolutions of the motor. Other probable causes are: - The positional deviation after triggering the deceleration dog is less than 128. - Insufficient voltage or malfunctioning Pulsecoder.
DS1120	UNASSIGNED ADDRESS (HIGH)	The upper 4 bits (EIA4 to EIA7) of an external data I/O interface address signal are set to an undefined address (high bits).

Number	Message	Description
DS1121	UNASSIGNED ADDRESS (LOW)	The lower 4 bits (EIA0 to EIA3) of an external data I/O interface address signal are set to an undefined address (low bits).
DS1124	OUTPUT REQUEST ERROR	OUTPUT REQUEST ERROR An output request was issued during external data output, or an output request was issued for an address that has no output data.
DS1128	DI.EIDLL OUT OF RANGE	The numerical value input by external data input signals ED0 to ED31 has exceeded the permissible range.
DS1130	SEARCH REQUEST NOT ACCEPTED	No requests can be accepted for a program No. or a sequence No. search as the system is not in the memory mode or the reset state.
DS1131	EXT-DATA ERROR (OTHER)	[External Data I/O] An attempt was made to input tool data for tool offset by a tool No. during loading by the G10 code.
DS1150	A/D CONVERT ALARM	A/D converter malfunction
DS1184	PARAMETER ERROR IN TORQUE	An invalid parameter was set for torque control. The torque constant parameter is set to “0”.
DS1448	ILLEGAL PARAMETER (D.C.S.)	The setting value of parameter for reference marks is satisfied the following any conditions. 1) A setting is made to use the absolute position detector (bit 5 (APC) of parameter No. 1815 is 1). 2) Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0. 3) Parameters 1821 and 1882 have identical settings. 4) The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting. 5) The setting value of parameters 1883 and 1884 are over the valid data range.
DS1449	REFERENCE MARK ARE DIFFERENT FROM PARAMETER	In case of distance coded linear scale I/F, the actual interval of reference marks is different from parameter (No.1821,1882) setting value.
DS1450	ZERO RETURN NOT FINISHED	When bit 0 (ZRN) of parameter No. 1005 is 0, if a manual reference position return has never been performed after power-up, the 1st reference position return (07h) is specified.
DS1451	IMPROPER PMC AXIS COMMAND	The PMC axes cannot be controlled in this state.
DS1512	EXCESS VELOCITY	T The feedrate of the linear axis during polar coordinate interpolation exceeded the maximum cutting feedrate.
DS1933	NEED REF RETURN(SYNC:MIX:OVL)	T The relation between a machine coordinate of an axis in synchronization, composite, or superimposed control, and the absolute, or relative coordinate was displaced. Perform the manual return to the reference position.
DS2003	ILLEGAL PARAMETER SETTING FOR LIVE TOOL AXIS (PMC AX-CTRL)	The spindle controlled with the servo motor is set as a PMC-controlled axis.
DS2005	NOW GAIN TUNING	Automatic operation cannot be started during automatic speed gain adjustment. Start automatic operation after confirming the completion of automatic adjustment.
DS5340	PARAMETER CHECK SUM ERROR	The parameter check sum does not match the reference check sum because of a change in the parameters. Restore the parameters or set the reference check sum again.

(12) Malfunction prevention function alarms (IE alarm)

Number	Message	Description
IE0001	+ OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the positive side was exceeded.
IE0002	- OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the negative side was exceeded.
IE0003	+ OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the positive side was exceeded.
IE0004	- OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the negative side was exceeded.
IE0005	+ OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the positive side was exceeded.
IE0006	- OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the negative side was exceeded.
IE0007	EXCESS MAXIMUM REV. DATA	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
IE0008	ILLEGAL ACC/DEC	The malfunction prevention function detected the acceleration/deceleration error.
IE0009	ILLEGAL MCN COODINATE	The malfunction prevention function detected the displacement of a machine coordinate in the check point.

A.2 ALARM LIST (PMC)

A.2.1 Messages That May Be Displayed on the PMC Alarm Screen

The following table lists the PMC alarm messages that may be displayed on the PMC alarm screen.

Alarm number	Faulty location/corrective action	Contents
ER01 PROGRAM DATA ERROR	<1> Enter the sequence program again. <2> If this error recurs even after you have entered the sequence program again, the error may be due to a hardware fault. In that case, contact us.	The sequence program is invalid.
ER02 PROGRAM SIZE OVER	<1> Reduce the size of the sequence program. <2> Contact us, and specify a ladder step count option that allows you to set a larger program size.	The sequence program is too large. The sequence program is invalid.
ER03 PROGRAM SIZE ERROR(OPTION)	<1> Reduce the size of the sequence program. <2> Contact us, and specify a ladder step count option that allows you to set a larger program size.	The sequence program exceeds the size specified by the ladder step count option.
ER04 PMC TYPE UNMATCH	Change the sequence program so that it specifies the adequate PMC type, by using the programmer.	The PMC type specified in the sequence program does not match the type of the PMC actually in use.
ER07 NO OPTION(LADDER STEP)	<1> Restore the backup CNC parameter data. <2> Contact us, and specify a ladder step count option that allows you to set a larger program size.	No ladder step count option is found.
ER08 OBJECT UNMATCH	Contact us.	An unsupported function is used in the sequence program.

Alarm number	Faulty location/corrective action	Contents
ER09 PMC LABEL CHECK ERROR. PLEASE TURN ON POWER AGAIN WITH PRESSING 'O'&'Z'. (CLEAR PMC SRAM)	<1> Turn on the power of the CNC again, by holding down the 'O' and 'Z' keys at the same time. <2> Replace the backup batteries.	The nonvolatile memory of the PMC system needs to be initialized in such cases as when you have changed the PMC model.
ER17 PROGRAM PARITY	<1> Enter the sequence program again. <2> If this error recurs even after you have entered the sequence program again, the error may be due to a hardware fault. In that case, contact us.	The parity of the sequence program is invalid.
ER18 PROGRAM DATA ERROR BY I/O	Enter the sequence program again.	An interrupt was specified while the sequence program was being read.
ER19 LADDER DATA ERROR	Display the LADDER DIAGRAM EDITOR screen again, and terminate the editing operation by pressing the [EXIT] soft key.	A function key was pressed during the editing of the ladder program, causing a switch to the CNC screen.
ER22 NO PROGRAM	Enter the sequence program again.	The sequence program is empty.
ER27 LADDER FUNC. PRM IS OUT OF RANGE	Correct the sequence program; change the parameter number specified in a functional instruction to a value that is within the allowable range.	An out-of-range parameter number is specified in the TMR, TMRB, CTR, CTRB, DIFU, or DIFD functional instruction.
ER28 NO OPTION(I/O LINK CHx)	Contact us; specify the I/O Link point count expansion option for the indicated channel.	The I/O Link point count expansion option is not specified for CHx.
ER31 NO OPTION (EXTENDED PMC LADDER INSTRUCTION FUNCTION)	Change the sequence program so that it specifies the adequate PMC model by using the programmer.	An attempt was made to execute a ladder program including an extended ladder diagram. Alternatively, the model set in the sequence program does not match the actual model.
ER32 NO I/O DEVICE	<1> Check whether the power of each I/O device is on. <2> Check whether the power of each I/O device has been turned on before the CNC. <3> Check cable connections.	None of the I/O devices, such as the I/O Link, connection unit, and Power Mate, is connected.
ER33 I/O LINK ERROR or ER33 I/O LINK ERROR(CHn)	Contact us; replace the faulty hardware.	The LSI for the I/O Link is faulty.
ER34 I/O LINK ERROR(xx) or ER34 I/O LINK ERROR(CHn xx)	<1> Check the cable connections to the devices of group xx. <2> Check whether the power of each I/O device has been turned on before the CNC. <3> Replace any device of group xx in which the PMC control module is embedded.	An I/O device communication error occurred on the slave side of group xx.
ER35 TOO MUCH OUTPUT DATA IN GROUP(xx) or ER35 TOO MUCH OUTPUT DATA IN GROUP(CHn xx)	Reduce the output data count of group xx.	The output data count of I/O Link group xx exceeds the upper limit (33 bytes). The superfluous data is regarded as invalid.
ER36 TOO MUCH INPUT DATA IN GROUP(xx) or ER36 TOO MUCH INPUT DATA IN GROUP(CHn xx)	Reduce the input data count of group xx.	The input data count of I/O Link group xx exceeds the upper limit (33 bytes). The superfluous data is regarded as invalid.

Alarm number	Faulty location/corrective action	Contents
ER37 TOO MUCH SLOT IN BASE or ER37 TOO MUCH SLOT IN BASE(CHn)	Correct the slot number to a value of 10 or less.	The slot number for the I/O Link exceed the upper limit (10). The slot number larger than 11 is regarded as invalid.
ER38 MAX SETTING OUTPUT DATA OVER(xx) or ER38 MAX SETTING OUTPUT DATA OVER(CHn xx)	Reduce the total amount of output data of all groups to 128 bytes or less.	The I/O area for the I/O Link is insufficient. (The area allocated to the group xx and later on the output side is regarded as invalid.)
ER39 MAX SETTING INPUT DATA OVER(xx) or ER39 MAX SETTING INPUT DATA OVER(CHn xx)	Reduce the total amount of input data of all groups to 128 bytes or less.	The I/O area for the I/O Link is insufficient. (The area allocated to the group xx and later on the input side is regarded as invalid.)
ER43 PROGRAM DATA ERROR(PT/NT)	<1> Input the sequence program recompiled by FANUC LADDER-III. <2> If the error persists even after inputting the sequence program again, contact FANUC.	The sequence program is invalid.
ER56 TOTAL PROGRAM SIZE OVER (OPTION)	<1> Contact FANUC and set the ladder number option again.	The total ladder step option is too large for a multipath PMC.
ER97 IO LINK FAILURE(CHx yyGROUP)	If this alarm is issued on a machine that once operated normally, possible causes are given below. (1) When communication with an I/O unit in the yy group or later is disabled - The communication cable between the (yy-1) group and the yy group is broken or in poor contact. - An I/O unit in the yy group or later is powered off or is not powered on yet. - An I/O unit in the yy group is defective or an I/O unit in the (yy-1) group is defective. (2) When the CNC is powered off and back on again, an I/O unit remains powered on. - When the CNC is powered off and back on again, all I/O units must be powered off once. If this alarm is issued when the sequence program is debugged, any of the following settings may be incorrect in addition to the above causes. (1) I/O module allocation setting (2) Setting of the parameter of the I/O Link allocation data select function I/O Link channel 2-path allocation setting	This alarm is issued when the number of I/O units is insufficient. This alarm is issued when the number of I/O units set by the I/O module allocation and I/O Link allocation select function parameters does not match the number of I/O units actually connected to the CNC. None of the I/O units connected to the channel on which this alarm was issued is linked. The ladder program runs regardless of this alarm.
WN02 OPERATE PANEL ADDRESS ERROR	Correct the Series 0 operator's panel address that is set in the PMC system parameter.	The Series 0 operator's panel address that is set in the PMC system parameter is invalid.

Alarm number	Faulty location/corrective action	Contents
WN03 ABORT NC-WINDOW/EXIN	<1> Check the ladder program to verify that it is free from errors, and then restart the ladder program (press the RUN key). <2> Turn on the power of the CNC again.	The ladder program was stopped while communication was in progress between CNC and PMC. This alarm may cause the WINDR, WINDW, EXIN, and DISPB functional instructions to malfunction.
WN07 LADDER SP ERROR(STACK)	Correct the sequence program so that the subprogram has eight or fewer levels of nesting.	There are too many levels of nesting (levels more than 8) for the CALL or CALLU functional instruction to call the subprogram.
WN09 SEQUENCE PROGRAM IS NOT WRITTEN TO FLASH ROM	If you want to use a changed sequence program again next time you power on the system, write the sequence program to flash ROM. If you have made any unwanted change to the sequence program by mistake, read the original sequence program from flash ROM.	You have changed the sequence program using the LADDER DIAGRAM EDITOR screen or DATA I/O screen, but you have not yet written the changed sequence program to flash ROM. If you shut down the system without writing the changed sequence program to flash ROM, the changes you have made will be nowhere next time you turn on the power.
WN10 NO OPTION(STEP SEQUENCE)	<1> Add the step sequence option. <2> Arrange so that the step sequence subprogram will not be called.	No step sequence option was found when the system attempted to execute a step sequence.
WN11 INCOMPATIBLE FUNCTION	Recompile the program using FANUC LADDER-III or ladder editing package.	A function command does not comply with this PMC.
WN57 OVERRIDE FUNCTION IS ACTIVE	Disable the override function before shipment because it is used to debug a ladder program.	The override function is enabled.
WN58 UNSUPPORTED FUNCTION	Correct the program using the internal ladder editing function.	There is an unsupported function command. The command was not processed.
WN59 MESSAGE FILE SYMBOL UNDEFINED	Correct the error in the multi-language display message file.	The multi-language display message file has a symbol not present in the ladder program.
WN60 MESSAGE FILE SYMBOL INVALID	Correct the error in the multi-language display message file.	A symbol other than the A address is defined in the multi-language display message file.
WN61 MESSAGE FILE ADDRESS DUPLICATE	Correct the error in the multi-language display message file.	An A address is redundantly defined between symbols and addresses or between symbols.
WN62 MESSAGE FILE NUMBER ERROR	Correct the error in the multi-language display message file.	Different message numbers are used for the same A address between the ladder program and the multi-language display message file.
WN63 MESSAGE FILE IS NOT WRITTEN TO FLASH ROM	When using the modified multi-language display message file, write the message file into a flash ROM.	Multi-language display message data was modified on the data input/output screen, but the modified multi-language display message data is not written to the flash ROM. The modified multi-language display message data is lost during power-on unless it is written to the flash ROM.

Alarm number	Faulty location/corrective action	Contents
WN64 MESSAGE FILE SIZE OVER	Reduce the size of the multi-language display message file. Contact FANUC and specify an option for a lager size.	The size of the multi-language display message file is larger than the program storage area. The multi-language display message file is invalid.
WN65 MESSAGE FILE MISMATCH	Contact FANUC.	An unsupported function is used in the multi-language display message file.
WN66 MESSAGE FILE PARITY	Enter the multi-language display message file again. If the error persists even after entering the message file, contact FANUC because a hardware failure is suspected.	The parity of the multi-language display message file is invalid.
WN67 MESSAGE FILE ERROR BY I/O	Enter the multi-language display message file again.	Interruption was specified during reading of the multi-language display message file.

NOTE
The group number displayed in message of ER34, ER35, ER36, ER38, ER39, or ER97 is a group number for wiring I/O units.

A.2.2 PMC System Alarm Messages

Alarm number	Faulty location/corrective action	Contents
PC004 CPU ERR xxxxxxxx:yyyyyyyy PC006 CPU ERR xxxxxxxx:yyyyyyyy PC009 CPU ERR xxxxxxxx:yyyyyyyy PC010 CPU ERR xxxxxxxx:yyyyyyyy PC012 CPU ERR xxxxxxxx:yyyyyyyy	This alarm may be due to a hardware fault; contact us with information on the circumstances under which the alarm occurred (displayed message, system configuration, operation suspected of causing the alarm, timing of alarm occurrence, frequency of occurrence, etc.) as well as the displayed internal error codes.	A CPU error occurred in the PMC system. xxxxxxxx and yyyyyyyy are internal error codes.
PC030 RAM PARI xxxxxxxx:yyyyyyyy	This alarm may be due to a hardware fault; contact us with information on the circumstances under which the alarm occurred (displayed message, system configuration, operation suspected of causing the alarm, timing of alarm occurrence, frequency of occurrence, etc.) as well as the displayed internal error codes.	A RAM parity error occurred in the PMC system. xxxxxxxx and yyyyyyyy are internal error codes.

Alarm number	Faulty location/corrective action	Contents
PC050 IOLINK ER1 CHz:GRyy:xx	(1) Check whether instantaneous power interruption or changes in voltage occurred in the slave unit of group number yy (0 to 15) connected to the I/O link line of channel number z (1 to 3). (2) Check whether there is no failure or no poor contact on the cable between JD1A of group number yy-1 (0 to 15) and JD1B of group number yy (0 to 15), which are connected to the I/O link line of channel number z (1 to 3). (3) Check whether the slave unit of group number yy (0 to 15) connected to the I/O Link line of channel number z (1 to 3) is not defective.	A communication error occurred in I/O Link. The symbol z indicates a channel number (1 to 3). The symbol yy indicates the group number (0 to 15) of the slave unit in which a problem may be caused. The symbol xx indicates an internal error code. This alarm is issued if communication with the slave unit of channel number z and group number yy is interrupted. The interruption may be caused by: (1) Instantaneous power interruption, a change in voltage, or a defective power cable in the slave unit (2) Defective communication cable or poor contact (3) Failure in the slave unit Group number yy indicated in this alarm may be incorrect depending on the state in which the alarm occurred, so the defective position may not always be identified.

Alarm number	Faulty location/corrective action	Contents
PC051 IOLINK ER2 CHz:yy:xx:ww:vv	(1) When using the I/O Unit-MODEL A, even though the base expansion has been allocated, the base is not connected. Check whether the units actually connected follow the I/O Link allocation. (2) When the Power Mate or Servo Motor β series I/O Link option was connected as an I/O slave unit, check whether a system alarm occurred first in any of these units. (3) Check whether noise is generated on the communication line. Check the grounding state of I/O Link slave units and the shielding state of connected cables. (4) Check whether the DO output of I/O units is not short-circuited. (5) Check whether instantaneous power interruption or changes in voltage occurred in the power for I/O Link master and slave units. (6) Check that there is no failure in cable connection. (7) Check that there is no failure in cables. (8) Check that the ground terminals of I/O units and the shield wires of communication cables are properly grounded. (9) Check that the I/O Link slave units are not defective. (10) Check that the PMC modules are not defective.	A communication error occurred in I/O Link. The symbol z indicates a channel number (1 to 3). The symbols yy, xx, ww, and vv indicate internal error codes. A communication error occurred on channel z of I/O Link. This alarm is issued by various factors related to I/O Link.
PC060 FBUS xxxxxxxx:yyyyyyyy PC061 FL-R xxxxxxxx:yyyyyyyy	This alarm may be due to a hardware fault; contact us with information on the circumstances under which the alarm occurred (displayed message, system configuration, operation suspected of causing the alarm, timing of alarm occurrence, frequency of occurrence, etc.) as well as the displayed internal error codes.	A bus error occurred in the PMC system.
PC070 LADDER SPE (PMCn)	Check the correspondence between the CALL or CALLU instruction and the SPE instruction.	A stack error occurred in function command SPE in the ladder program of the n-th path PMC.

Alarm number	Faulty location/corrective action	Contents
PC097 LADDER PARITY ERR (PMCn) PC098 CODE PARITY ERR	This alarm may be due to a hardware fault; contact us with information on the circumstances under which the alarm occurred (displayed message, system configuration, operation suspected of causing the alarm, timing of alarm occurrence, frequency of occurrence, etc.) as well as the displayed internal error codes.	A RAM check error occurred.
PC501 NC/PMC INTERFACE ERR PATH_	Contact us with information on the circumstances under which the alarm occurred (displayed message, system configuration, operation suspected of causing the alarm, timing of alarm occurrence, frequency of occurrence, etc.).	The read or write operation between CNC and PMC failed.
PC502 LADDER SUBaaa (PMCn)	Correct the sequence program so that the SUBaaa functional instruction will not be used.	Command SUBaaa used in the n-th path PMC is unsupported.

NOTE
The group number displayed in message of PC050 is a group number for wiring I/O units.

A.2.3 Operation Errors

Error messages that may be displayed on the PMC LADDER DIAGRAM VIEWER screen

Alarm number	Faulty location/corrective action	Contents
INPUT INVALID	Input a valid address or numeric value.	The input address or numeric value is invalid.
PROGRAM IS PROTECTED BY PASSWORD	Enter the password.	The screen cannot be displayed because the program is protected by the password.
ILLEGAL SUBPROGRAM NAME	Input a existent subprogram number or symbol.	A nonexistent subprogram number or symbol is specified.
SYMBOL UNDEFINED	Input a defined symbol or bit address.	An undefined symbol character string is specified.
THE NET IS NOT FOUND		The specified net is not found.
THE ADDRESS IS NOT FOUND		The specified address is not found.
THE FUNCTIONAL INSTRUCTION IS NOT FOUND		The specified functional instruction is not found.
WRITE COIL NEEDS BIT ADDRESS	Specify a bit address for the write coil search.	You entered a byte address when specifying an address used for the write coil search.
SOME NETS ARE DISCARDED	The system cannot pick up all the nets. Choose the nets to pick up, by using the LADDER DIAGRAM VIEWER display screen, and then perform the net pickup operation manually.	The system failed to pick up all the nets because there were 128 nets or more to be picked up.

Alarm number	Faulty location/corrective action	Contents
PROGRAM IS BEING MODIFIED	Disconnect the online communication with FANUC LADDER-III. Stop other applications from accessing the ladder data.	The ladder data cannot be displayed because online communication with FANUC LADDER-III is in progress or another application is accessing the ladder data.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.

Error messages that may be displayed on the PMC LADDER DIAGRAM EDITOR screen

Alarm number	Faulty location/corrective action	Contents
THIS NET IS PROTECTED		When you are editing data on a per-subprogram basis, you cannot edit the subprogram frame nets (END1, END2, END3, SP, and SPE).
TOO LARGE DATA TO COPY	Reduce the range of data to copy. Perform the copy operation several times, copying a smaller range of data at a time.	The selected range of data exceeds the size of the copy buffer.
TOO LARGE DATA TO PASTE	Reduce the size of data to paste.	An attempt was made to paste data whose size exceeded the free space of the sequence program.
BIT ADDRESS IS REQUIRED	Make sure that the address types match for the alteration operation.	An attempt was made to alter a bit address to a byte address.
BYTE ADDRESS IS REQUIRED	Make sure that the address types match for the alteration operation.	An attempt was made to alter a byte address to a bit address.
ILLEGAL PMC ADDRESS	Check the address to be input, and then enter it correctly.	• A character string was entered that was unacceptable as a PMC address. • A wildcard (*) was specified in an inappropriate manner. • Ether "OLD ADDRESS" or "NEW ADDRESS" was not entered.
THE ADDRESS IS READ-ONLY	Enter a write-permitted address.	• An attempt was made to alter a write coil address to a write-prohibited bit address. • An attempt was made to alter an address set in an output parameter of a functional instruction to a write-prohibited bit address.
THE ADDRESS TYPE ARE MISMATCHED	Check the types of the address in "OLD ADDRESS" and "NEW ADDRESS" and, if necessary, enter the correct address or addresses.	The type of the addresses in "OLD ADDRESS" does not match that in "NEW ADDRESS".
***** DOSE NOT HAVE SYMBOL	Define symbol data in "OLD ADDRESS".	No symbol data is defined in "OLD ADDRESS".
***** ALREADY HAS SYMBOL	Make sure that the address types match for the alteration operation.	Symbol data is already defined in "NEW ADDRESS".

Error messages that may be displayed on the PMC LADDER DIAGRAM EDITOR screen (when updating)

Alarm number	Faulty location/corrective action	Contents
OVERLAPPED COM	If COME is missing, add it in proper position. If the COM is unnecessary, remove it.	There is no COME that corresponds to this COM.
END IN COM END1 IN COM END2 IN COM	If COME is missing, add it in proper position. If COM is unnecessary, remove it.	END,END1,END2, or END3 is found between COM and COME.
JMPE IN COM	JMPE and corresponding JMP must have same COM/COME status. Review JMP range and COM range, to adjust not to overlap with each other: it is possible that one range includes the other completely.	JMPE is found between COM and COME, and JMP and corresponding JMPE have different COM/COME status.
SP/SPE IN COM	If COME is missing, add it in proper position. If the COM is unnecessary, remove it.	SP or SPE is found between COM and COME.
COME WITHOUT COM	If COM is missing, add it in proper position. If the COME is unnecessary, remove it.	There is no COM that corresponds to this COME.
DUPLICATE CTR NUMBER (WARNING)	If some of them are unnecessary, remove them. If all of them are necessary, assign other number to parameter of them to make them unique. (If two or more instructions with same parameter number will never be active simultaneously at one time, the Ladder program has a possibility to work correctly, however, it is recommended from safety and maintenance points of view, that all these instructions should have different parameter number with each other.)	Plural CTRs have the same number as their parameter. (This is warning.)
ILLEGAL CTR NUMBER	If unnecessary, remove it. Assign correct number not to exceed the maximum number defined by each PMC model.	CTR has parameter number that is out of range.
DUPLICATE DIFU/DIFD NUMBER (WARNING)	If some of them are unnecessary, remove them. If all of them are necessary, assign other number to parameter of them to make them unique. (If two or more instructions with same parameter number will never be active simultaneously at one time, the Ladder program has a possibility to work correctly, however, it is recommended from safety and maintenance points of view, that all these instructions should have different parameter number with each other.)	Plural DIFUs or DIFDs have the same number as their parameter. (This is warning.)
ILLEGAL DIFU/DIFD NUMBER	If unnecessary, remove it. Assign correct number not to exceed the maximum number defined by each PMC model.	DIFU or DIFD has parameter number that is out of range.

Alarm number	Faulty location/corrective action	Contents
NO END NO END1 NO END2 NO END3	Add END, END1, END2 or END3 in proper position.	END, END1, END2 or END3 is not found.
DUPLICATE END1 DUPLICATE END2 DUPLICATE END3	Remove extra END1, END2 or END3.	Multiple END1, END2 or END3 are found.
GARBAGE AFTER END GARBAGE AFTER END2 GARBAGE AFTER END3	Remove unnecessary nets, and move necessary nets to proper position so that they will be executed.	There are some nets after END, END2 or END3, which will not be executed.
OVERLAPPED JMP	If JMPE is missing, add it in proper position. If the JMP is unnecessary, remove it.	There is no JMPE that corresponds to this JMP.
JMP/JMPE TO BAD COM LEVEL	JMP and corresponding JMPE must have same COM/COME status. Review JMP range and COM range, to adjust not to overlap with each other: it is possible that one range includes the other completely.	JMP and corresponding JMPE have different COM/COME status.
COME IN JMP	COME and corresponding COM must have same JMP/JMPE status. Review COM range and JMP range, to adjust not to overlap with each other: it is possible that one range includes the other completely.	COME is found between JMP and JMPE, and COM and corresponding COME have different JMP/JMPE status.
END IN JMP END1 IN JMP END2 IN JMP END3 IN JMP	If JMPE is missing, add it in proper position. If JMP is unnecessary, remove it.	END,END1,END2, or END3 is found between JMP and JMPE.
SP/SPE IN JMP	If JMPE is missing, add it in proper position. If the JMP is unnecessary, remove it.	SP or SPE is found between JMP and JMPE.
JMPB OVER COM BORDER	JMPB and its destination must have same COM/COME status. Review range of JMPB and COM range, to adjust not to overlap with each other: it is possible that one range includes the other completely.	JMPB and its destination differ in COM/COME status.
JMPB OVER LEVEL	JMPB can only jump to the same program level, or within a subprogram. If the JMPB is unnecessary, remove it. If LBL for the JMPB is missing, add it in proper position. If it should be JMPC, correct it.	JMPB jumps to different program level.
LBL FOR JMPB NOT FOUND	If JMPB is unnecessary, remove it. If LBL is missing, add it in proper position.	Can not find proper LBL for JMPB.
JMPC IN BAD LEVEL	JMPC is used to jump from a subprogram to level 2. If the JMPC is unnecessary, remove it. If it should be JMPB or JMP, correct it.	JMPC is used in other than subprogram.
LBL FOR JMPC NOT FOUND	If JMPC is unnecessary, remove it. If LBL is missing, add it in proper position: JMPC jumps into level 2.	Can not find proper LBL for JMPC.

Alarm number	Faulty location/corrective action	Contents
LBL FOR JMPC IN BAD LEVEL	JMPC is used to jump from a subprogram to level 2. If the JMPC is unnecessary, remove it. If another LBL of same L-address that the JMPC is intended to jump exists in the subprogram, assign different L-address to these two LBLs. If it should be JMPB or JMP, correct it.	Destination of JMPC is not level 2.
JMPC INTO COM	LBL for JMPC must be located out of any COM and COME pair. If the JMPC is unnecessary, remove it. If the LBL is located wrong, move it to correct position. If the L-address of JMPC is wrong, correct it.	JMPC jumps to LBL between COM and COME.
JMPE WITHOUT JMP	If JMP is missing, add it in proper position. If the JMPE is unnecessary, remove it.	There is no JMP that corresponds to this JMPE.
TOO MANY LBL	Remove unnecessary LBLs. If this error still occurs, adjust the construction of program to use less LBLs.	There are too many LBLs.
DUPLICATE LBL	If some of these LBLs are unnecessary, remove them. If all of these LBLs is necessary, assign other L-addresses to them to make all LBLs unique.	Same L-address is used in plural LBLs.
OVERLAPPED SP	If SPE is missing, add it in proper position. If the SP is unnecessary, remove it.	There is no SPE that corresponds to this SP.
SPE WITHOUT SP	If SP is missing, add it in proper position. If the SPE is unnecessary, remove it.	There is no SP that corresponds to this SPE.
END IN SP	If SPE is missing, add it in proper position. If END is in wrong place, move it to proper position.	END is found between SP and SPE.
DUPLICATE P ADDRESS	If some of these SPs are unnecessary, remove them. If all of these SPs is necessary, assign other P-addresses to them to make all SPs unique.	Same P-address is used in plural SPs.
DUPLICATE TMRB NUMBER (WARNING)	If some of them are unnecessary, remove them. If all of them are necessary, assign other number to parameter of them to make them unique. (If two or more instructions with same parameter number will never be active simultaneously at one time, the Ladder program has a possibility to work correctly, however, it is recommended from safety and maintenance points of view, that all these instructions should have different parameter number with each other.)	Plural TMRBs have the same number as their parameter. (This is warning.)
ILLEGAL TMRB NUMBER	If unnecessary, remove it. Assign correct number not to exceed the maximum number defined by each PMC model.	TMRB has parameter number that is out of range.

Alarm number	Faulty location/corrective action	Contents
DUPLICATE TMR NUMBER (WARNING)	If some of them are unnecessary, remove them. If all of them are necessary, assign other number to parameter of them to make them unique. (If two or more instructions with same parameter number will never be active simultaneously at one time, the Ladder program has a possibility to work correctly, however, it is recommended from safety and maintenance points of view, that all these instructions should have different parameter number with each other.)	Plural TMRs have the same number as their parameter. (This is warning.)
ILLEGAL TMR NUMBER	If unnecessary, remove it. Assign correct number not to exceed the maximum number defined by each PMC model.	TMR has parameter number that is out of range.
NO SUCH SUBPROGRAM	If it calls wrong subprogram, correct it. If the subprogram is missing, create it.	Subprogram that is called by CALL/CALLU is not found.
UNAVAILABLE INSTRUCTION	Confirm that this ladder program is correct one. If this program is correct one, all these unsupported instructions have to be removed.	Unsupported instruction for this PMC model is found.
SP IN BAD LEVEL	SP can be used at top of a subprogram. Correct it so that no SP exists in other place.	SP is found in wrong place.
LADDER PROGRAM IS BROKEN	This ladder program must be all cleared once, and remake ladder program.	Ladder program may be broken by some reason.
NO WRITE COIL	Add proper write coil.	Write coil is necessary, but is not found.
CALL/CALLU IN BAD LEVEL	CALL/CALLU must be used in Level 2 or in subprograms. Do not use any other places.	CALL/CALLU is used in wrong place.
SP IN LEVEL3	If END3 is located wrong, move it to correct position. If the SP is unnecessary, remove it.	SP is found in level 3.

Error messages that may be displayed on the PMC NET EDITOR screen

Alarm number	Faulty location/corrective action	Contents
ILLEGAL FUNCTIONAL INSTRUCTION NAME	Specify the name of an available functional instruction.	The entered name of functional instruction is invalid.
TOO MANY FUNCTIONAL INSTRUCTIONS IN ONE NET	Only one functional instruction is allowed to constitute a net. If necessary, divide the net into plural nets.	Too many functional instructions are in one net.
TOO LARGE NET	Divide the net into plural nets so that step number in a net may become small.	Net is too large. When a net is converted into the object, the net exceeds 256 steps.
NO INPUT FOR OPERATION	Coil without input, or coil connected to output of functional instruction that has no output, causes this error. If coil is not necessary, remove it. If necessary, connect it to meaningful input.	No signal is provided for logical operation.

OPERATION AFTER FUNCTION IS FORBIDDEN	Output of functional instruction can not be connected to a contact, nor to conjunction with other signal that will be implemented by logical-or operation.	No logical operation with functional instruction output is permitted, except write coils.
WRITE COIL IS EXPECTED	Write coil is not found even if it is expected. Add proper write coil to the net.	Write coil is expected, but not found.
BAD COIL LOCATION	Coil can be located only at rightmost column. Any coil located at other place must be erased once, and place necessary coils in correct place.	Coil is located in bad position.
SHORT CIRCUIT	Find contact with terminals connected by short circuit, and correct connections.	Some contacts are connected with short circuit. CTR has a parameter number that is out of the range.
FUNCTION AFTER DIVERGENCE IS FORBIDDEN	Functional instruction can not be used in output section of net. If necessary, divide the net into plural nets.	Functional instruction is used in output section of net.
ALL COIL MUST HAVE SAME INPUT	Left terminals of all coils in a net must be connected to same input point.	When a net contains more than one coil, the coils should not have any contact beside them affects only of the coils.
BAD CONDITION INPUT	Check the connection of all condition inputs of the functional instruction. Especially for functional instruction that has more than one condition input, check if connections to condition inputs interfere with each other.	Some condition input of functional instruction is not connected correctly.
NO CONNECTION	Find gap that is expected to be connected, and correct the connection.	There is signal connected to nowhere.
NET IS TOO COMPLICATED	Examine every connection, and find unnecessarily bending connection, or coils that are connected to different point.	Net is too complicated to analyze.
PARAMETER IS NOT SUPPLIED	Enter all of the relay addresses, and parameters of functional instructions.	Relay with blank address, or blank parameter of functional instruction, is found.

Error messages that may be displayed on the TITLE DATA EDITOR screen

Alarm number	Faulty location/corrective action	Contents
TOO MANY CHARACTERS	Make sure that the entered character string is within the allowable input length.	The number of characters in the entered character string exceeds the allowable input length. Some of the characters are discarded.
PROGRAM IS BEING MODIFIED	Disconnect the online communication with FANUC LADDER-III. Stop other applications from accessing the title data.	The title data cannot be displayed because online communication with FANUC LADDER-III is in progress or another application is accessing the title data.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.

Error messages that may be displayed on the SYMBOL & COMMENT DATA EDITOR screen

Alarm number	Faulty location/corrective action	Contents
TOO MANY CHARACTERS	Make sure that the entered address is within the allowable input length.	The number of characters in the entered address exceeds the allowable address input length.
ADDRESS IS REQUIRED	Enter an address correctly.	No address was entered during the batch input of address, symbol, and comment data using the SYMBOL & COMMENT EDITOR screen.
ILLEGAL PMC ADDRESS	Enter an address correctly.	The specified address is invalid, or the entered address character string contains a space or spaces.
THE ADDRESS ALREADY HAS AN ENTRY	Specify another address.	An already registered address was entered.
THE SYMBOL NAME IS ALREADY USED	Specify another symbol.	An already registered symbol was entered.
PMC ADDRESS MUST BE ENTERED	Enter a PMC address in the ADDRESS field.	No PMC address was entered when new symbol/comment data is registered.
TOO LONG SYMBOL NAME	Make sure that the symbol consists of 16 characters or less.	The entered symbol exceeds the specified number of characters.
TOO LONG COMMENT STRING	Make sure that the comment consists of 30 characters or less.	The entered comment exceeds the specified number of characters.
BAD SYMBOL NAME	Define a symbol that contains no space.	The entered symbol contains a space or spaces.
THE STRING IS NOT FOUND	Specify another character string for the search.	The search was done for the specified character string but did not find it.
OUT OF SPACE	Create free space for the sequence program, by deleting unnecessary ladder or message data.	The symbol/comment editing area has no free space.
PROGRAM IS BEING MODIFIED	Disconnect the online communication with FANUC LADDER-III. Stop other applications from accessing the symbol/comment data.	The symbol/comment data cannot be displayed because online communication with FANUC LADDER-III is in progress or another application is accessing the symbol/comment data.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.
BAD SYMBOL NAME	Change the symbol name.	The symbol name is invalid.
CANNOT EDIT ADDRESS AUTO ASSIGNED SYMBOL	Use FANUC LADDER-III to change the symbol.	The symbol whose PMC address is assigned automatically by compiling FANUC LADDER-III, can not edit.
ILLEGAL DATA TYPE	Enter a correct data type.	The specified data type is invalid.
ILLEGAL PROGRAM NAME	Enter a correct program name.	The specified program is invalid.
LINE FEED IS NOT AVAILABLE IN THIS DATA	Line feed code can be entered in comment data only. Do not enter it in other data.	Line feed code can not be entered in this data.
LINE FEED IS NOT AVAILABLE IN THIS MODE	Enter Line feed code in the insert or overwrite mode.	Line feed code can not be entered in this mode.
NO SYMBOL. PROGRAM SETTING IS IGNORED	Symbol name is required for local symbol.	The specified program is ignored because no symbol is specified.
NOTHING TO PASTE	You need to copy or cut character strings before you paste them.	You try to paste character strings without copying or cutting ones.

Alarm number	Faulty location/corrective action	Contents
TOO LARGE DATA TO PASTE	Shorten the character string to copy or cut.	The character strings is too long to copy or cut.
UNAVAILABLE CHARACTERS WAS OMITTED.	Do not copy or cut characters which can not be used at pasted position.	The characters which can not be used at pasted position, were omitted.

Error messages that may be displayed on the MESSAGE DATA EDITOR screen

Alarm number	Faulty location/corrective action	Contents
INPUT INVALID	Enter ";" in the 5th digit position in the batch message input process.	The delimiter code - semicolon (;) - was not entered in the batch message input process.
ILLEGAL NUMBER	Enter a four-digit number as the message number.	The entered message number contains any nonnumeric character, or a number shorter than four digits was entered.
THE NUMBER IS OUT OF RANGE	Make sure that the entered message number is in the range between 1000 and 9999.	The entered message number is out of the 1000-9999 range.
CLOSING "@" IS NOT FOUND	When entering kana or other Japanese characters, make sure that they are enclosed within a pair of @ signs.	One of the @ sign pair is missing.
BAD NUMBER OF CHARACTERS IN "@-@"	Enter a character string correctly between a pair of @ signs.	The number of characters entered between the pair of @ signs is not even.
ILLEGAL CHARACTER IN "@-@"	Enter a character string correctly between a pair of @ signs.	One or more invalid character codes exist between the pair of @ signs.
BAD NUMBER OF CHARACTERS FOR 2-BYTE CODE	Enter a two-byte code correctly between @02 and 01@.	The number of characters in the two-byte code (characters entered between @02 and 01@) is not a multiple of four.
ILLEGAL 2-BYTE CODE	Enter a two-byte code correctly between @02 and 01@.	The two-byte code (characters entered between @02 and 01@) contains one or more characters other than the JIS codes.
CLOSING CONTROL CODE "01" IS NOT FOUND	Enter the closing control code.	The two-byte code (characters entered between @02 and 01@) lacks the closing control code (01).
CONTROL CODE "XX" IS REPEATED	Remove any repeated control code.	The starting control code (02), closing control code (01), and/or umlaut code (0D) is repeated.
CLOSING "]" IS NOT FOUND	Make sure that the "[" and "]" codes are entered in pairs.	The delimiter codes for numerical data are not entered in pairs.
BAD NUMERICAL DATA FORMAT	Specify the numerical data correctly.	The format of the numerical data is invalid.
BAD PMC ADDRESS FOR NUMERIAL DATA	Enter an available address.	The address section of the numerical data is invalid.
PROGRAM IS BEING MODIFIED	Disconnect the online communication with FANUC LADDER-III. Stop other applications from accessing the message data.	The message data cannot be displayed because online communication with FANUC LADDER-III is in progress or another application is accessing the message data.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.

Error messages that may be displayed on the I/O MODULE EDITOR screen

Alarm number	Faulty location/corrective action	Contents
GROUP NUMBER IS TOO LARGE	Specify 15 or a smaller value as the group number.	The entered group number is too large.
BASE NUMBER IS TOO LARGE	Specify base number 0 for I/O Unit-B (##, #1 - #10).	The entered base number is too large.
SLOT NUMBER IS TOO LARGE	Specify 30 or a smaller value as the slot number for I/O Unit-B (##, #1 - #10). For other I/O units, specify 10 or a smaller value.	The entered slot number is too large.
SLOT NUMBER IS TOO SMALL	Specify 0 or a large value as the slot number for I/O Unit-B (##, #1 - #10). For other I/O units, specify 1 or a larger value.	The entered slot number is too small.
I/O UNIT NAME MISMATCH	Check the I/O unit name or address.	The input I/O unit is assigned to the Y address, or the output I/O unit is assigned to the X address.
ILLEGAL I/O UNIT NAME	Enter a correct I/O unit name.	The entered I/O unit name is invalid.
NOT ENOUGH SPACE	Enter the data again after creating free space by deleting the data allocated behind the current cursor position or by other adequate means.	There is not enough free address space for the size of the I/O unit you are going to assign. This error also occurs if you attempt to assign the I/O unit to an already allocated address space.
PROGRAM IS BEING MODIFIED	Disconnect the online communication with FANUC LADDER-III. Stop other applications from accessing the I/O module data.	The I/O module data cannot be displayed because online communication with FANUC LADDER-III is in progress or another application is accessing the I/O module data.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.

Error messages that may be displayed on the SYSTEM PARAMETER screen

Alarm number	Faulty location/corrective action	Contents
INPUT INVALID	Enter a correct numerical value.	The entered numerical value or its input format is invalid.
SYMBOL UNDEFINED	Enter a defined symbol or bit address.	An undefined symbol character string was entered.
PROGRAM IS BEING MODIFIED	Disconnect the online communication with FANUC LADDER-III. Stop other applications from accessing the system parameter data.	The system parameter data cannot be displayed because online communication with FANUC LADDER-III is in progress or another application is accessing the system parameter data.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.

Error messages that may be displayed on the SIGNAL STATUS screen

Alarm number	Faulty location/corrective action	Contents
INPUT INVALID	Enter a correct numerical value.	The entered numerical value or its input format is invalid.
SYMBOL UNDEFINED	Enter a defined symbol or bit address.	An undefined symbol character string was entered.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.

Error messages that may be displayed on the PMC PARAM screen

Alarm number	Faulty location/corrective action	Contents
INPUT INVALID	Enter a correct numerical value.	The entered numerical value or its input format is invalid.
MUST BE IN EMERGENCY STOP OR IN MDI MODE	Change to MDI mode or emergency stop mode.	There is not in MDI mode or emergency stop mode.
PWE MUST BE ON	Set “PWD” to 1 in NC setting screen.	“PWD” is 0 in NC setting screen.
EITHER PWE OR KEY4 MUST BE ON	Set “PWD” to 1 in NC setting screen or set the program protect signal “KEY4” to 1	“PWD” is 0 in NC setting screen and the program protect signal “KEY4” is 0.
THIS FUNCTION IS PROTECTED	Cancel the protection by the programmer protection function or 8-level protection function.	This function is protected by the programmer protection function or 8-level protection function.

Error messages that may be displayed on the SIGNAL TRACE screen

Alarm number	Faulty location/corrective action	Contents
TRACE FUNCTION IS ALREADY IN USE	Wait until FANUC LADDER-III or some other application finishes using the trace function before executing it.	FANUC LADDER-III or some other application is currently using the trace function.
NO SAMPLING ADDRESS	Specify a bit address as a sampling address in the trace parameter.	No sampling address is specified in the trace parameter.
NO STOP TRIGGER ADDRESS	Specify a bit address as the stop trigger address in the trace parameter.	The stop trigger address is not specified in the trace parameter.
NO SAMPLING TRIGGER ADDRESS	Specify a bit address as the sampling trigger address in the trace parameter.	The sampling trigger address is not specified in the trace parameter.

Error messages that may be displayed on the trace setting screen

Alarm number	Faulty location/corrective action	Contents
INPUT INVALID	Enter a numerical value that is within the specified data range of the relevant trace parameter.	A nonnumeric value or an out-of-range parameter value was entered.
SYMBOL UNDEFINED	Enter a defined symbol or bit address.	An undefined symbol character string was entered.
BIT ADDRESS IS REQUIRED	Specify a bit address as the stop or sampling trigger address.	A byte address was specified as the stop or sampling trigger address.
INVALID STOP TRIGGER ADDRESS	Enter a PMC signal address that can be used as the stop trigger address.	The bit address entered as the stop trigger address is invalid.

Alarm number	Faulty location/corrective action	Contents
INVALID SAMPLING TRIGGER ADDRESS	Enter a PMC signal address that can be used as the sampling trigger address.	The bit address entered as the sampling trigger address is invalid.

Error messages that may be displayed on the I/O diagnosis screen

Alarm number	Faulty location/corrective action	Contents
ENTER STRING TO SEARCH.	Enter a string before starting a search.	No search string is specified.
I/O DIAGNOSIS FUNCTION IS NOT SUPPORTED	To use the I/O diagnosis function, update the PMC system software.	The I/O diagnosis function cannot be used because the PMC system software is an older version.
INPUT INVALID	Check the entered string.	The entered string is invalid.
LADDER PROGRAM IS BROKEN	Check the information displayed on the PMC alarm screen and reload the program.	The program is corrupted.
NO GROUP FORMAT.	Use [GROUP] on the I/O diagnosis (setting) screen.	No group display is set.
PROGRAM IS BEING MODIFIED.	Retry after completing the function that is using the program.	The program cannot be referenced because it is being used by another function.
REACHED TO THE END OF SYMBOL DATA.	To make another search, specify a string again.	The search has been completed until the end of the data has been reached.
SYMBOL ORDER IS NOT AVAILABLE.	Use the FANUC LADDER-III to convert the program to one with an expanded function.	The format of this program does not allow sorting and display in symbol order.
THE GROUP IS NOT FOUND	Check the specified group.	The specified group is not found.
FORCING IS PROTECTED ON THIS PATH.	Disable the programmer protection function.	The forced input/output function is currently protected on the selected PMC path.
THE STRING IS NOT FOUND	Check the specified string.	The specified string is not found.

A.2.4 I/O Communication Error Messages

The error messages that may appear on the I/O screen and their meanings and actions are listed below.

Error messages displayed during memory card I/O operation

Alarm number	Faulty location/corrective action	Contents
MEMORY CARD IS NOT READY	Check whether a memory card is installed.	No memory card is installed.
MEMORYCARD IS FULL	Delete files to create available space.	There is no available space in the memory card.
MEMORYCARD IS WRITE PROTECTED	Release the write protection of the memory card.	The memory card is write-protected.
MEMORYCARD IS NOT FORMATTED	Format the memory card in FAT16. (It cannot be recognized if formatted in FAT32.)	The memory card cannot be recognized.
TOO MANY FILES IN MEMORYCARD	Delete unnecessary files to reduce the number of files.	There are too many files.
FILE NOT FOUND	On the list screen, check the file name or file number.	The specified file cannot be found.
FILE IS READ-ONLY	Check the attributes of the file.	Write to the specified file is not permitted.
FILE NAME IS INVALID	Specify the file name in MS-DOS form.	The file name is illegal.

Alarm number	Faulty location/corrective action	Contents
COULD NOT FORMAT MEMORY CARD	The NC cannot format this memory card. Format the memory card in FAT16 using another PC. (It cannot be recognized if formatted in FAT32.)	The memory card cannot be formatted.
UNSUPPORTED MEMORYCARD	Replace the memory card with another one.	This memory card is not supported.
CAN NOT DELETE FILE	Check the attributes of the file.	An error occurred when a file was deleted from the memory card.
MEMORYCARD BATTERY ALARM	Replace the battery of the memory card.	The battery of the memory card has become weak.
THIS FILE NAME IS ALREADY USED	Change the file name to another one.	The file name is already used.
MEMORYCARD ACCESS ERROR	Replace the memory card with another one.	The memory card cannot be accessed.
DIFFERENCE FOUND		File comparison detected a mismatch.
MEMORY CARD IS LOCKED BY OTHER FUNCTION	Wait until the PMC user completes processing, then retry.	Another PMC user is using the memory card.
MEMORY CARD HEADER ROM DATA ID IS ILLEGAL	This file cannot be read. Check the type of the file.	An attempt was made to read a file, but its ROM data ID was illegal.
FILE NUMBER CAN NOT SELECTED	If the file does not exist, the key entry is invalid. If this error occurs even when the cursor is placed at a file name, contact the FANUC service center.	The file number cannot be selected.
THE FILE NUMBER DOES NOT EXIST	Check the total number of files on the list screen.	The entered file number is not present. The entered number exceeds the total number of files.
FILE NUMBER IS RESTRICTED TO "128"	Enter a numeric value not exceeding 128.	A value up to 128 can be entered as the file number.
MEMORY CARD IS USED BY OTHER FUNCTION	Retry after terminating the other function that is currently using the memory card.	Some other function is currently using the memory card.
MEMORY CARD IS WRITE PROTECTED	Cancel the write protection of the memory card, or use another memory card that is not write protected.	The memory card is write protected.
UNSUPPORTED MEMORY CARD	Use another memory card.	This is an unsupported type of memory card.
COULD NOT DELETE FILE	Check the read/write permission attribute of the file.	The file cannot be deleted.
TRACE FILE NUMBER IS OVER	Delete unnecessary old trace result file or files.	No more trace result file can be created because the maximum trace result file number (file extension) has been reached.
INTERNAL ERROR (xxxxxxxxxx)	Contact the FANUC service center, and report the displayed message correctly.	An error due to an internal factor occurred. Details on the error are displayed in parentheses.

Error messages displayed during flash ROM I/O operation

Alarm number	Faulty location/corrective action	Contents
NOT IN EMG STOP MODE	Place the system in the emergency stop state.	The system is not in the emergency stop state.
INVALID LADDER PROGRAM	Check the program.	The transfer program is illegal.
DIFFERENCE FOUND		A file comparison detected a mismatch.

Alarm number	Faulty location/corrective action	Contents
FLASH ROM IS LOCKED BY OTHER FUNCTION	Wait until the PMC user completes processing, then retry.	Another PMC user is using the flash ROM.
FLASH ROM HEADER ROM DATA ID IS ILLEGAL	This file cannot be read. Check the type of the file.	An attempt was made to read a file, but its ROM data ID was illegal.
FLASH ROM IS USED BY OTHER FUNCTION	This file cannot be read. Check the type of the file.	The ROM data ID of the file you attempted to read is invalid.
INTERNAL ERROR (xxxxxxxxxx)	Contact the FANUC service center, and report the displayed message correctly.	An error due to an internal factor occurred. Details on the error are displayed in parentheses.

Error messages displayed during FLOPPY or other input/output device I/O operation

Alarm number	Faulty location/corrective action	Contents
ILLEGAL PMC PARAMETER FORMAT	Specify a file of the PMC parameter format. Also, check the specified file to see whether its content is not disrupted.	The specified file is not of the PMC parameter format.
ILLEGAL HANDY FILE FORMAT	Specify a file of the handy file format. Also, check the specified file to see whether its content is not disrupted.	The specified file is not of the handy file format.
UNKNOWN FILE FORMAT	Specify file of recognizable format such as PMC parameter format, or check the contents of the file.	Can not recognize the format of specified file.
FILE NAME OR FILE NUMBER IS REQUIRED	Specify file name or file number for the operation.	Need file name or file number to identify file to read, compare, or delete.
COMMUNICATION TIMEOUT	Check the communication parameters such as baud rate, and retry to communicate.	Communication with the I/O device has been timeout.
I/O DEVICE IS NOT ATTACHED OR IN ERROR STATUS	Check the power of I/O device is ON. Check the I/O device is connected. Check the cable that connects I/O device with PMC is correct one. If some error has occurred in I/O device, solve it.	Any I/O device is not connected, or some error has occurred in it.
RECEIVED BAD DATA: CHECK THE COMMUNICATION PARAMETERS	Check the PMC's communication parameters such as baud rate match the ones of I/O device.	Invalid data has been received.
RECEIVED DATA HAS OVERRUN	Check the communication parameters about flow control.	Too many data have received at once.
OTHERS FUNCTION IS USING THIS CHANNEL	Use the other channel, or stop the function.	Others function is using this channel.
BAD COMMUNICATION PARAMETER	Check the communication parameters such as baud rate.	Setting parameters of communication are not correct.
OTHER FUNCTION IS USING I/O FUNCTION	Wait until function that using I/O function do finish, or stop the function.	Another function such as FANUC LADDER-III is using I/O function.
UNKNOWN HANDY FILE FORMAT DATA	Check the file.	The received data is not a program of the PMC system or is a program of some other incompatible type.
ILLEGAL BAUD RATE SETTING	Set a valid baud rate.	The set baud rate is invalid.
ILLEGAL CHANNEL NUMBER	Set a valid channel number.	The set channel number is invalid.
ILLEGAL PARITY BIT SETTING	Set a valid parity bit.	The set parity bit is invalid.
ILLEGAL STOP BIT SETTING	Set a valid stop bit.	The set stop bit is invalid.

Alarm number	Faulty location/corrective action	Contents
ILLEGAL WRITE CODE SETTING	Set a valid output code.	The set output code is invalid.
SEQUENCE PROGRAM IS IN USE BY ONLINE FUNCTION	Wait until On-line function, do finish the using I/O function. In general, both of I/O function and On-line function should not be used at the same time.	Can not input/output of sequence program, because On-line function is using sequence program.

Common error messages that may be displayed on individual devices during the I/O operations

Alarm number	Faulty location/corrective action	Contents
ERROR OCCURS IN LADDER PROGRAM	Check the PMC alarm screen and correct the indicated program error accordingly.	Data cannot be output because there is an error in the ladder program.
UNKNOWN DATA TYPE	Check the file.	The PMC type of the input data is unknown.
MUST BE IN EMERGENCY STOP	Set the NC to the emergency stop state.	The NC is not in the emergency stop state when the PMC parameter is read.
PWE MUST BE ON	Set PWE to 1 on the NC setting screen.	PWE on the NC setting screen is 0 during reading from the PMC parameter.
MUST BE IN EDIT MODE	Set the NC to the EDIT mode.	The NC is not in the EDIT mode during writing to the PMC parameter.
THIS FUNCTION IS NOT ALLOWED	Release the protection by the programmer protection function or 8-level protection function.	Protection is made by the programmer protection function or 8-level protection function.
PMC PARAMETER IS LOCKED BY OTHER FUNCTION	Retry after terminating the other function that is currently using the PMC parameter.	The PMC parameter is currently used by some other function and cannot be referenced by this function.
THIS DEVICE IS USED BY OTHER FUNCTION	Retry after terminating the other function that is currently using the specified device.	The specified device is currently used by some other function and cannot be used by this function.
PMC PARAMETER IS PROTECTED BY OTHER FUNCTION	Retry after terminating the other function that is currently using the PMC parameter.	The PMC parameter is currently used by some other function and cannot be changed by this function.
LADDER TYPE UNMATCH	Specify a program of a valid type.	The specified program is of a different type and cannot be read.
TOO LARGE LADDER PROGRAM	Check the file. Or, change to a step number option that allows you to set a larger program size.	The ladder program is too large to read.
LADDER PROGRAM IS USED BY OTHER FUNCTION	Retry after terminating the other function that is currently displaying the ladder program.	The ladder program is currently used by some other function and cannot be referenced by this function.

A.3 ALARM LIST (SERIAL SPINDLE)

When a serial spindle alarm occurs, the following number is displayed on the CNC.

NOTE

*1 Note that the meanings of the Spindle Amplifier indications differ depending on which LED, the red or yellow LED, is on. When the red LED is on, the Spindle Amplifier indicates a 2-digit alarm number. When the yellow LED is on, the Spindle Amplifier indicates an error number that designates a sequence problem (for example, when a rotation command is entered with the emergency stop state not released).
See "Error Codes (Serial Spindle)."

*2 For serial spindle alarms with a number not listed, refer to the following documents depending on the spindle motor to which a connection is actually made.
- FANUC AC SPINDLE MOTOR αi series Maintenance Manual (B-65285EN)
- Technical Report etc.

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9001	SSPA:01 MOTOR OVERHEAT	01	1 Check and correct the peripheral temperature and load status. 2 If the cooling fan stops, replace it.	The internal temperature of the motor exceeds the specified level. The motor is used in excess of the continuous rating, or the cooling component is abnormal.
SP9002	SSPA:02 EX DEVIATION SPEED	02	1 Check and correct the cutting conditions to decrease the load. 2 Correct parameter No. 4082.	The motor speed cannot follow a specified speed. An excessive motor load torque is detected. The acceleration/deceleration time in parameter No. 4082 is insufficient.
SP9003	SSPA:03 DC-LINK FUSE IS BROKEN	03	1 Replace the Spindle Amplifier. 2 Check the motor insulation status.	The Power Supply (PS) becomes ready (“00” is indicated), but the DC link voltage is too low in the Spindle Amplifier. The fuse in the DC link section in the Spindle Amplifier is blown. (The power device is damaged or the motor is ground-fault.)
SP9004	SSPA:04 POWER SUPPLY ERROR	04	Check the voltage of the power input to the Power Supply (PS) and the connection status.	The Power Supply (PS) found a missing power supply phase. (Power Supply (PS) alarm E)
SP9006	THERMAL SENSOR DISCONNECT	06	1 Check and correct the parameter. 2 Replace the feedback cable.	The temperature sensor of the motor is disconnected.

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9007	SSPA:07 OVER SPEED	07	Check for a sequence error. (For example, check whether spindle synchronization was specified when the spindle could not be turned.)	The motor speed has exceeded 115% of its rated speed. When the spindle axis was in position control mode, positional deviations were accumulated excessively (SFR and SRV were turned off during spindle synchronization.)
SP9009	SSPA:09 OVERHEAT MAIN CIRCUIT	09	1 Improve the heat sink cooling status. 2 If the external radiator cooling fan stops, replace the Spindle Amplifier.	The temperature in the power semiconductor cooling radiator is abnormally high.
SP9010	SSPA:10 LOW VOLT INPUT POWER	10	1 The input power voltage in the Power Supply (PS) is too low. 2 The power cable between amplifiers is abnormal. 3 The spindle amplifier is abnormal.	A drop in the input power voltage in the spindle amplifier is detected.
SP9011	SSPA:11 OVERVOLT POWER CIRCUIT	11	1 Check the selected Power Supply (PS). 2 Check the input power voltage and change in power during motor deceleration. If the voltage exceeds 253 VAC (for the 200-V system) or 530 VAC (for the 400-V system), improve the power supply impedance.	Overvoltage of the DC link section of the Power Supply (PS) was detected. (Power Supply (PS) alarm indication: 7) Power Supply (PS) selection error. (The maximum output specification of the Power Supply (PS) is exceeded.)
SP9012	SSPA:12 OVERCURRENT POWER CIRCUIT	12	1 Check the motor insulation status. 2 Check the spindle parameters. 3 Replace the Spindle Amplifier.	The motor current is abnormally high. A motor-specific parameter does not match the motor model. Poor motor insulation
SP9013	SSPA:13 CPU DATA MEMORY FAULT	13	Replace the Spindle Amplifier control printed circuit board.	Abnormality in an Spindle Amplifier control circuit component is detected. (Internal RAM is abnormal.)
SP9014	SERIAL SPINDLE ALARM	14	Upgrade the spindle software program.	A spindle amplifier that is not registered in the spindle software program is used.
SP9015	SSPA:15 SPINDLE SWITCHING FAULT	15	1 Check and correct the ladder sequence. 2 Replace the switching MC.	The switch sequence in spindle switch/output switch operation is abnormal. The switching MC contact status check signal and command do not match.
SP9016	SSPA:16 RAM ERROR	16	Replace the Spindle Amplifier control printed circuit board.	Abnormality in an Spindle Amplifier control circuit component is detected. (RAM for external data is abnormal.)

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9017	SERIAL SPINDLE ALARM	17	Replace the Spindle Amplifier.	Abnormality in spindle amplifier ID data is detected.
SP9018	SSPA:18 SUMCHECK ERROR PROGRAM ROM	18	Replace the Spindle Amplifier control printed circuit board.	Abnormality in an Spindle Amplifier control circuit component is detected. (Program ROM data is abnormal.)
SP9019	SSPA:19 EXCESS OFFSET CURRENT U	19	Replace the Spindle Amplifier.	Abnormality in an Spindle Amplifier component is detected. (The initial value for the U phase current detection circuit is abnormal.)
SP9020	SSPA:20 EXCESS OFFSET CURRENT V	20	Replace the Spindle Amplifier.	Abnormality in an Spindle Amplifier component is detected. (The initial value of the V phase current detection circuit is abnormal.)
SP9021	POS SENSOR POLARITY ERROR	21	Check and correct the parameters. (Parameter No. 4000#0, 4001#4)	The polarity parameter setting of the position sensor is wrong.
SP9022	SERIAL SPINDLE ALARM	22	1 Review operation conditions (acceleration/ deceleration and cutting) to reduce the load. 2 Check and correct the parameters.	A spindle amplifier overload current was detected.
SP9024	SSPA:24 SERIAL TRANSFER ERROR	24	1 Place the CNC-to-spindle cable away from the power cable. 2 Replace the cable.	The CNC power is turned off (normal power-off or broken cable). An error is detected in communication data transferred to the CNC.
SP9027	SSPA:27 DISCONNECT POSITION CODER	27	Replace the cable.	The spindle position coder (connector JYA3) signal is abnormal.
SP9029	SSPA:29 OVERLOAD	29	Check and correct the load status.	Excessive load has been applied continuously for a certain period of time. (This alarm is issued also when the motor shaft has been locked in the excitation state.)
SP9030	SSPA:30 OVERCURRENT INPUT CIRCUIT	30	Check and correct the power supply voltage.	Overcurrent is detected in Power Supply (PS) main circuit input. (Power Supply (PS) alarm indication: 1) Unbalanced power supply. Power Supply (PS) selection error (The maximum Power Supply (PS) output specification is exceeded.)
SP9031	SSPA:31 MOTOR LOCK OR DISCONNECT DETECTOR	31	1 Check and correct the load status. 2 Replace the motor sensor cable (connector JYA2).	The motor cannot rotate at a specified speed. (A level not exceeding the SST level for the rotation command has existed continuously.)
SP9032	SSPA:32 SIC-LSI RAM FAULT	32	Replace the Spindle Amplifier control printed circuit board.	Abnormality in an Spindle Amplifier control circuit component is detected. (The LSI device for serial transfer is abnormal.)

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9033	SSPA:33 SHORTAGE POWER CHARGE	33	1 Check and correct the power supply voltage. 2 Replace the Power Supply (PS).	The DC power voltage in the power circuit is insufficient when the magnetic contactor is turned on. (Power Supply (PS) alarm indication: 5) (Such as open phase and defective charging resistor).
SP9034	SSPA:34 ILLEGAL PARAMETER	34	Correct a parameter value according to the FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN). If the parameter number is unknown, connect the spindle check board, and check the indicated parameter.	Parameter data exceeding the allowable limit is set.
SP9036	SSPA:36 OVERFLOW ERROR COUNTER	36	Check whether the position gain value is too large, and correct the value.	An error counter overflow occurred.
SP9037	SSPA:37 ILLEGAL SETTING VELOCITY DETECTOR	37	Correct the value according to the FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN).	The setting of the parameter for the number of pulses in the speed detector is incorrect.
SP9041	SSPA:41 ILLEGAL 1REV SIGN OF POSITION CODER	41	1 Check and correct the parameter. 2 Replace the cable.	1 The 1-rotation signal of the spindle position coder (connector JYA3) is abnormal. 2 Parameter setting error
SP9042	SSPA:42 NO 1REV SIGN OF POSITION CODER	42	Replace the cable.	The 1-rotation signal of the spindle position coder (connector JYA3) is disconnected.
SP9043	SSPA:43 DISCONNECT POSITION CODER DEF. SPEED	43	Replace the cable.	The differential speed position coder signal (connector JYA3S) in the submodule SW is abnormal.
SP9046	SSPA:46 ILLEGAL 1REV SIGN OF SCREW CUT	46	1 Check and correct the parameter. 2 Replace the cable. 3 Readjust the BZ sensor signal.	The 1-rotation signal in threading is abnormal.
SP9047	SSPA:47 ILLEGAL SIGNAL OF POSITION CODER	47	1 Replace the cable. 2 Correct the cable layout (vicinity of the power line).	The A/B phase signal of the spindle position coder (connector JYA3) is abnormal. The relationship between the A/B phase and 1-rotation signal is incorrect (Pulse interval mismatch).
SP9049	SSPA:49 DEF. SPEED IS OVER VALUE	49	Check whether the calculated differential speed value exceeds the maximum motor speed.	In differential speed mode, the speed of the other spindle converted to the speed of the local spindle has exceeded the allowable limit (the differential speed is calculated by multiplying the speed of the other spindle by the gear ratio).

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9050	SSPA:50 SYNCRONOUS VALUE IS OVER SPEED	50	Check whether the calculated value exceeds the maximum motor speed.	In spindle synchronization, the speed command calculation value exceeded the allowable limit (the motor speed is calculated by multiplying the specified spindle speed by the gear ratio).
SP9051	SSPA:51 LOW VOLT POWER CIRCUIT	51	1 Check and correct the power supply voltage. 2 Replace the MC.	Input voltage drop was detected. (Power Supply (PS) alarm indication: 4) (Momentary power failure or poor MC contact)
SP9052	SSPA:52 ITP FAULT 1	52	1 Replace the Spindle Amplifier control printed circuit board. 2 Replace the main board or sub CPU board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier (the ITP signal stopped).
SP9053	SSPA:53 ITP FAULT 2	53	1 Replace the Spindle Amplifier control printed circuit board. 2 Replace the main board or sub CPU board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier the ITP signal stopped).
SP9054	SSPA:54 OVERCURRENT	54	Review the load state.	An overload current was detected.
SP9055	SSPA:55 ILLEGAL POWER LINE	55	1 Replace the magnetic contactor. 2 Check and correct the sequence.	The power line state signal of the magnetic contactor for selecting a spindle or output is abnormal.
SP9056	COOLING FAN FAILURE	56	Replace the internal cooling fan.	The internal cooling fan stopped.
SP9057	CONV. EX. DECELERATION POW.	57	1 Decrease the acceleration/deceleration duty. 2 Check the cooling condition (peripheral temperature). 3 If the cooling fan stops, replace the resistor. 4 If the resistance is abnormal, replace the resistor.	An overload was detected in the regenerative resistance. (Power Supply (PS) alarm indication: H) Thermostat operation or short-time overload was detected. The regenerative resistor was disconnected, or an abnormal resistance was detected.
SP9058	CNV. OVERLOAD	58	1 Check the Power Supply (PS) cooling status. 2 Replace the Power Supply (PS).	The temperature of the radiator of the Power Supply (PS) has increased abnormally. (Power Supply (PS) alarm indication: 3)
SP9059	CNV. COOLING FAN FAILURE	59	Replace the Power Supply (PS).	The internal cooling fan for the Power Supply (PS) stopped. (Power Supply (PS) alarm indication: 2)
SP9061	SERIAL SPINDLE ALARM	61	Check parameter settings.	The error between the semi-closed and full-closed sides when the dual position feedback function is used is too large.

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9065	SERIAL SPINDLE ALARM	65	1 Check parameter settings. 2 Check sensor connections and signals. 3 Check power line connections.	The move distance is too long when the magnetic pole is confirmed (synchronization spindle)
SP9066	COM. ERROR BETWEEN SP AMPS	66	1 Replace the cable. 2 Check and correct the connection.	An error was found in communication between spindle amplifiers (connector JX4).
SP9069	SAFETY SPEED OVER	69	1 Check federate. 2 Check parameter settings. 3 Check sequence.	In the state in which safety speed monitoring was enabled, the system detected that the motor speed exceeded the safety speed or detected an error during a free-run stop.
SP9070	ILLEGAL AXIS DATA	70	Setting the spindle amplifier side to conform the connection state.	The axis data is abnormal. An error was detected in an axis number check.
SP9071	SAFETY PARAMETER ERROR	71	1 Inputting the safety parameter again. 2 Replace the spindle amplifier control printed-circuit board.	The safety parameter is abnormal. An abnormality is detected in the safety parameter.
SP9072	MISMATCH RESULT OF MOTOR SPEED CHECK	72	1 Replace the spindle amplifier control printed-circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	A mismatch was detected between the safety speed check results of the spindle amplifier and those of the CNC.
SP9073	MOTOR SENSOR DISCONNECTED	73	1 Replace the feedback cable. 2 Check the shield processing. 3 Check and correct the connection. 4 Adjust the sensor.	The motor sensor feedback signal is not present (connector JYA2).
SP9074	CPU TEST ERROR	74	Replace the spindle amplifier control printed-circuit board.	An error was detected in a CPU test.
SP9075	CRC ERROR	75	Replace the spindle amplifier control printed-circuit board.	An error occurred in a spindle amplifier CRC test.
SP9076	INEXECUTION OF SAFETY FUNCTIONS	76	Replace the spindle amplifier control printed-circuit board.	The spindle amplifier detected that safety functions were not executed.
SP9077	MISMATCH RESULT OF AXIS NUMBER CHECK	77	1 Replace the spindle amplifier control printed-circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	A mismatch was detected between the axis number check results of the spindle amplifier and those of the CNC.
SP9078	MISMATCH RESULT OF SAFETY PARAMETER CHECK	78	1 Replace the spindle amplifier control printed-circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	A mismatch was detected between the safety parameter check results of the spindle amplifier and those of CNC.

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9079	INITIAL TEST ERROR	79	Replace the spindle amplifier control printed-circuit board.	An error was detected in a spindle amplifier initial test.
SP9080	ALARM AT THE OTHER SP AMP.	80	Remove the cause of the alarm of the remote Spindle Amplifier.	During inter-Spindle Amplifier communication, an alarm was generated on the remote Spindle Amplifier.
SP9081	1-ROT MOTOR SENSOR ERROR	81	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the motor sensor cannot be correctly detected(connector JYA2).
SP9082	NO 1-ROT MOTOR SENSOR	82	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the motor sensor is not generated(connector JYA2).
SP9083	MOTOR SENSOR SIGNAL ERROR	83	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a motor sensor feedback signal(connector JYA2).
SP9084	SPNDL SENSOR DISCONNECTED	84	1 Replace the feedback cable. 2 Check the shield processing. 3 Check and correct the connection. 4 Check and correct the parameter. 5 Adjust the sensor.	The spindle sensor feedback signal is not present(connector JYA4).
SP9085	1-ROT SPNDL SENSOR ERROR	85	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the spindle sensor cannot be correctly detected(connector JYA4).
SP9086	NO 1-ROT SPNDL SENSOR	86	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the spindle sensor is not generated(connector JYA4).
SP9087	SPNDL SENSOR SIGNAL ERROR	87	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a spindle sensor feedback signal(connector JYA4).
SP9088	COOLING RADI FAN FAILURE	88	Replace the Spindle Amplifier external radiator cooling fan.	The external radiator cooling fan stopped.
SP9089	SERIAL SPINDLE ALARM	89	1 Check the connection between the Spindle Amplifier and the submodule SM (SSM). 2 Replace the submodule SM(SSM). 3 Replace the Spindle Amplifier control printed-circuit board.	Submodule SM (SSM) error (synchronous spindle)
SP9090	SERIAL SPINDLE ALARM	90	1 Check magnetic pole detection operation. 2 Check whether the rotor and sensor are aligned correctly.	Abnormal rotation of the synchronous spindle motor was detected.

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9091	SERIAL SPINDLE ALARM	91	Replace the motor sensor cable.	Count error of the magnetic pole position of the synchronous spindle motor
SP9092	SERIAL SPINDLE ALARM	92	Check the sequence (whether SFR or SRV is turned on and off in the position control mode).	The motor speed exceeds the acceleration level corresponding to the velocity command.
SP9110	AMP COMMUNICATION ERROR	b0	1 Replace the communication cable between Spindle Amplifier and Power Supply (PS). 2 Replace the Spindle Amplifier or Power Supply (PS) control printed circuit board.	Communication error between Spindle Amplifier and Power Supply (PS).
SP9111	CONV. LOW VOLT CONTROL	b1	Replace the Power Supply (PS) control printed circuit board.	Low Power Supply (PS) control power supply voltage (Power Supply (PS) indication = 6)
SP9112	CONV. EX. DISCHARGE POW.	b2	1 Check the regenerative resistance. 2 Check the motor selection. 3 Replace the Power Supply (PS)	Excessive Power Supply (PS) regenerative power (Power Supply (PS) indication = 8)
SP9113	CONV. COOLING FAN FAILURE	b3	Replace the external radiator cooling fan for Power Supply (PS).	Stopped the external radiator cooling fan for Power Supply (PS) (Power Supply (PS) indication = A)
SP9120	COMMUNICATION DATA ERROR	C0	1 Replace the communication cable between CNC and Spindle Amplifier. 2 Replace the Spindle Amplifier control printed circuit board. 3 Replace the CNC side main board or sub CPU board.	Communication data alarm
SP9121	COMMUNICATION DATA ERROR	C1	1 Replace the communication cable between CNC and Spindle Amplifier. 2 Replace the Spindle Amplifier control printed circuit board. 3 Replace the CNC side main board or sub CPU board.	Communication data alarm
SP9122	COMMUNICATION DATA ERROR	C2	1 Replace the communication cable between CNC and Spindle Amplifier. 2 Replace the Spindle Amplifier control printed circuit board. 3 Replace the CNC side main board or sub CPU board.	Communication data alarm
SP9123	SERIAL SPINDLE ALARM	C3	Replace the submodule SW(SSW).	Submodule SW (SSW) error (spindle switching)
SP9131	SERIAL SPINDLE ALARM	d1	Check the message displayed by SERVO GUIDE.	Spindle adjustment function alarm

Number	Message	Amplifier indication (*1)	Faulty location and remedy	Description
SP9132	SERIAL SPINDLE ALARM	d2	Replace the sensor.	Serial data error
SP9133	SERIAL SPINDLE ALARM	d3	Replace the sensor.	Data transfer error
SP9134	SERIAL SPINDLE ALARM	d4	1 Check and correct the sensor parameter setting. 2 Take action against noise. 3 Replace the sensor.	Soft phase
SP9135	SAFETY SPEED ZERO ERROR(SP)	d5	Perform operation within the safety speed zero range.	The motor position exceeded the safety speed zero monitoring width.
SP9136	MISMATCH RESULT OF SAFETY SPEED ZERO CHECK(SP)	d6	Replace the spindle amplifier control printed circuit board.	The spindle amplifier speed zero determination result did not match the CNC speed zero determination result.
SP9137	SERIAL SPINDLE ALARM	d7	Replace the spindle amplifier control printed circuit board.	Device communication error
SP9139	SERIAL SPINDLE ALARM	d9	Replace the sensor.	Pulse error
SP9140	SERIAL SPINDLE ALARM	E0	1 Take action against noise. 2 Replace the sensor.	Count error
SP9141	SERIAL SPINDLE ALARM	E1	1 Check and correct the sensor parameter setting. 2 Replace the sensor.	Serial sensor one-rotation signal undetected

A.4 ERROR CODES (SERIAL SPINDLE)

NOTE

*1 Note that the meanings of the Spindle Amplifier indications differ depending on which LED, the red or yellow LED, is on. When the yellow LED is on, an error code is indicated with a 2-digit number. When the red LED is on, the Spindle Amplifier indicates the number of an alarm generated in the serial spindle.
→ See Appendix A.3, “ALARM LIST (SERIAL SPINDLE).”

Diagnosis indication (*1)	Description	Remedy
01	Although neither *ESP (emergency stop signal; there are two types of signals including the input signal and contact signal of Power Supply (PS)) nor MRDY (machine ready signal) is input, SFR (forward rotation signal)/SRF (reverse rotation signal)/ORCM (orientation command) is input.	Check the *ESP and MRDY sequence. For MRDY, pay attention to the parameter setting regarding the use of the MRDY signal (parameter No. 4001#0).
03	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No. 4002 = 0, 0, 0, 0), but a Cs contour control command is input. In this case, the motor is not excited.	Check the parameter settings.

Diagnosis indication (*1)	Description	Remedy
04	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No.4002 = 0, 0, 0,), but a servo mode (rigid tapping, spindle positioning, etc.) or spindle synchronization command is input. In this case, the motor is not excited.	Check the parameter settings.
05	The orientation function option parameter is not specified, but ORCM (orientation command) is input.	Check the orientation function parameter settings.
06	The output switching control function option parameter is not specified, but low-speed characteristic winding is selected (RCH = 1).	Check the output switching control function parameter settings and the power line state check signal (RCH).
07	A Cs contour control command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
08	A servo mode (rigid tapping, spindle positioning, etc.) control command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
09	A spindle synchronization command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
10	A Cs contour control command is input, but another mode (servo mode, spindle synchronization, or orientation) is specified.	Do not switch to another mode during a Cs contour control command. Before moving to another mode, cancel the Cs contour control command.
11	A servo mode (rigid tapping, spindle positioning, etc.) command is input, but another mode (Cs contour control, spindle synchronization, or orientation) is specified.	Do not switch to another mode during a servo mode command. Before moving to another mode, cancel the servo mode command.
12	A spindle synchronization command is input, but another mode (Cs contour control, servo mode, or orientation) is specified.	Do not switch to another mode during a spindle synchronization command. Before moving to another mode, cancel the spindle synchronization command.
13	An orientation command is input, but another mode (Cs contour control, servo mode, or spindle synchronization control) is specified.	Do not switch to another mode during an orientation command. Before moving to another mode, cancel the orientation command.
14	Both SFR (clockwise rotation command) and SRV (counterclockwise rotation command) are input at the same time.	Issue either of them.
16	The parameter settings are such that the differential speed control function is not used (bit 5 of parameter No. 4000 = 0), but DEFMD (differential speed mode command) is input.	Check the parameter settings and differential speed mode command.
17	The speed detector parameter settings (bits 2, 1, and 0 of parameter No. 4011) are not valid. There is no corresponding speed detector.	Check the parameter settings.

Diagnosis indication (*1)	Description	Remedy
18	The parameter settings are such that a position sensor is not used (position control not performed (bits 3, 2, 1, and 0 of parameter No. 4002), but position coder system orientation is issued.	Check the parameter settings and the input signal.
19	The magnetic sensor orientation command is input, but another mode (Cs contour control, servo mode, or spindle synchronization control) is specified.	Do not switch to another mode during an orientation command. Before moving to another mode, cancel the orientation command.
21	The tandem operation command was input in the spindle synchronization control enable state.	Input the tandem operation command when spindle synchronization control is canceled.
22	Spindle synchronization control was specified in the tandem operation enable state.	Specify spindle synchronization control when torque tandem operation is canceled.
23	The tandem operation command is input without the required option.	Torque tandem control requires a CNC software option. Check the option.
24	If index is performed continuously in position coder system orientation, an incremental operation is performed first (INCMD = 1), then an absolute position command (INCMD = 0) is input.	Check INCMD (incremental command). If an absolute position command is to follow, be sure to perform absolute position command orientation first.
26	The parameter settings are such that both spindle switch and three-stage output switch are used.	Check the parameter settings and the input signal.
29	The parameter settings are such that the shortest-time orientation function is used (bit 6 of parameter No. 4018 = 0, Nos. 4320 to 4323 ≠0).	In the αi series spindle amplifier, the shortest-time orientation function cannot be used. The use of the optimum orientation function is recommended.
30	The magnetic pole has not been detected, but a command is input.	In the magnetic pole undetected state (EPFIXA = 0), the motor cannot be driven even when a command is input. Input a command in the magnetic pole detected state (EPFIXA = 1). When EPFSTR is set to 1, any command is ignored and this error is displayed even in the magnetic pole detected state. After the completion of magnetic pole detection, set EPFSTR to 0.
32	S0 is not specified as the velocity mode, but the disturbance input function is enabled (bit 7 of parameter No. 4395 is set to 1).	Specify S0 as the velocity mode before enabling the disturbance input function (bit 7 of parameter No. 4395 to 1).
34	Both the spindle FAD function and spindle EGB function are enabled. In this case, the motor is not excited.	These functions cannot be used simultaneously. Enable only either of these functions.
35	Spindle amplifier ID information cannot be obtained.	Replace the spindle amplifier with one with correct ID information.
36	The submodule SM (SSM) is abnormal. 1) The interface signal between the spindle amplifier and the SSM is disconnected. 2) SSM failure	For action to be taken for this status error, refer to Section 1.4, “SUBMODULE SM,” in Part IV in the FANUC AC SPINDLE MOTOR $\alpha i/\beta i$ series Parameter Manual (B-65280EN).
37	The current loop setting (No. 4012) has been changed.	Check the setting of parameter No. 4012, and turn the power off, then on again.

Diagnosis indication (*1)	Description	Remedy
38	A parameter related to communication between spindle amplifiers is specified incorrectly. Alternatively, a function unavailable with the torque tandem function is set.	Check the parameters.
39	Although SFR (forward rotation command), SRV (reverse rotation command), or ORCM (orientation command) is input, DSCN (disconnection detection disable signal) is input.	Check the sequence. Do not input DSCN (disconnection detection disable signal) during the input of a command which excites the motor.
43	A setting which does not support the αiCZ sensor (serial) is used.	Check the parameter settings.
44	The spindle amplifier does not support the control period setting.	Check the setting of parameter No. 4012.