APPENDIX

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 bit 0 (TVC) of parameter No. 0000 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	INVALID BREAK POINT OF WORDS	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	1) An unusable G code is specified. 2) The continuous circle motion-based groove cutting option parameter is not effective. 3) The continuous circle motion-based groove cutting enable signal is "0".
0011	FEED ZERO (COMMAND)	1) The cutting feedrate instructed by an F code has been set to 0. 2) 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. 3) During continuous circle motion-based groove cutting, correct Q or F value is not specified or the acceleration clamp value for continuous circle motion in parameter No. 3490 is invalid.
0014	CAN NOT COMMAND G95	A synchronous feed is specified without the option for threading / synchronous feed. Modify the program.
0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either add on the simultaneous axis control extension option, or 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 or involute interpolation. To enable programming of 3 or more axes, the helical interpolation option must be added to each of the relevant axes.
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)	F0 (rapid traverse in inverse feed or feed specified by an F code with 1–digit number) was specified during circular interpolation (G02, G03) or involute interpolation (G02.2, G03.2).

Number	Message	Description
0027	NO AXES COMMANDED IN G43/G44	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.
0029	ILLEGAL OFFSET VALUE	Illegal offset No.
0030	ILLEGAL OFFSET NUMBER	An illegal offset No. was specified. This alarm is also generated when the tool shape offset No. exceeds the maximum number of tool offset sets in the case of tool offset memory B.
0031	ILLEGAL P COMMAND IN G10	The relevant data input or option could not be found for the L No. of G10. No data setting address such as P or R was 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 CUTTER COMPENSATION	The intersection cannot be obtained by the intersection calculation in tool radius/tool nose radius compensation. Modify the program.
0034	NO CIRC ALLOWED IN STUP/EXT BLK	In tool radius/tool nose radius compensation, a startup or cancellation is performed in the G02 or G03 mode. Modify the program.
0035	CAN NOT COMMANDED G31	- 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. - 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. Or, specify the torque limit override by address Q.
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	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in G41 and G42 commands (tool radius/tool nose radius compensation). The program may cause overcutting to occur in chamfering or corner R. Modify the program.
0041	INTERFERENCE IN CUTTER COMPENSATION	In tool radius/tool nose radius compensation, excessive cutting may occur. Modify the program.
0042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in tool radius compensation or three-dimensional cutter compensation. Modify the program.
0043	ILLEGAL T-CODE COMMAND	On a system with a DRILL-MATE ATC installed, M06 is not specified in a block that specifies a T code. Alternatively, a T code beyond the allowable range is specified.
0044	G27-G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 is commanded in canned cycle mode. Modify the program.

Number	Message	Description
0045	ADDRESS Q NOT FOUND (G73/G83)	In a high-speed peck drilling cycle (G73) or peck drilling cycle (G83), the amount of each-time cutting is not specified by address Q, or Q0 is specified. Modify the program.
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.) Although an option for a return to the third or fourth reference position was not set, 3 or 4 was specified in address P.
0047	ILLEGAL AXIS SELECT	Two or more parallel axes (in parallel with a basic axis) have been specified upon start-up of 3-dimensional tool compensation or three-dimensional coordinate conversion.
0048	BASIC 3 AXIS NOT FOUND	Start-up of 3-dimensional tool compensation or three-dimensional coordinate conversion has been attempted, but the three basic axes used when Xp, Yp, or Zp is omitted are not set in parameter No. 1022.
0049	ILLEGAL COMMAND(G68,G69)	When three-dimensional coordinate conversion (G68 or G69) was specified, the tool compensation was not canceled. Or, programs of three-dimensional coordinate conversion (G68, G69) and tool compensation (G43, G44 or G49) were not nested. Or, the three-dimensional coordinate conversion was specified during the tool length compensation and another tool length compensation was specified.
0050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
0052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01 (or vertical line). Modify the program.
0053	TOO MANY ADDRESS COMMANDS	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	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	In chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.
0056	NO END POINT & ANGLE IN CHF/CNR	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	Block end point is not calculated correctly in direct dimension drawing programming. Modify the program.
0058	END POINT NOT FOUND	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. Although input/output of a pot number of tool data or offset input was requested, no tool number was input after power on. 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.

Number	Message	Description
0061	P OR Q COMMAND IS NOT IN THE MULTIPLE REPETIVE CYCLES BLOCK	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	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	The sequence number specified by addresses P and Q in multiple repetitive cycle (G70, G71, G72, or G73) command cannot be searched.
0064	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)	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	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	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	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	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	- The address to be searched was not found. - The program with specified program number was not found in program number search. - In the program restart block number specification, the specified block number could not be found. Check the data.
0072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option) or 1000 (option). 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	ILLEGAL PROGRAM NUMBER	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.

Number	Message	Description
0076	PROGRAM NOT FOUND	The specified program is not found in the subprogram call, macro call or graphic copy. The M, G, T or S codes are called by a P instruction other than that in an M98, G65, G66, G66.1 or interrupt type custom macro, and a program is called by a No. 2 auxiliary function code. 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.
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 3 (ABG0) of parameter No. 2200 is set to “1”. Set bit 3 of parameter No. 2200 to “0” before executing a match.
0080	G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT	- For machining center series 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. - For lathe 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	- For machining center series The tool length measurement function (G37) is specified without specifying an H code. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified without specifying an T code. Correct the program.
0082	G37 SPECIFIED WITH H CODE	- For machining center series The tool length measurement function (G37) is specified together with an H code in the same block. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified together with an T code in the same block. Correct the program.

Number	Message	Description
0083	G37 IMPROPER AXIS COMMAND	- For machining center series 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. - For lathe 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.
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 zero. Reprogram so that the axis returns to zero.
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 Operator'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 offset amount changed.) Perform the correct operation according to the Operator'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 offset amount changed.) Perform the correct operation according to the Operator's Manual.

Number	Message	Description
0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or alarms 0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
0098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
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	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. In the EGB axis skip function (G31.8), a non-existent custom macro variable number is specified. Or, the number of custom macro variables used to store skip positions is not sufficient. Alternatively, the header data in high-speed cycle machining is improper. This alarm is issued in the following cases. 1) The header corresponding to the specified call machining cycle number is absent. 2) The value of cycle connection information falls outside the allowable range (0 to 999). 3) The number of data items in the header falls outside the allowable range (1 to 65535). 4) The storage start data variable number of executable data falls outside the allowable ranges (#20000 to #85535/#200000 to #986431/#2000000 to #3999999). 5) The storage end data variable number of executable data falls outside the allowable ranges (#85535/#986431/#3999999). 6) The storage start data variable number of executable data is the same as the variable number used by the header.
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.

Number	Message	Description
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 can be specified as an argument only in an every-block call (G66.1).
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	The spindle positioning axis and another axis are specified in the same block.
0137	M-CODE & MOVE CMD IN SAME BLK.	The spindle positioning axis and another axis are specified in the same block.
0138	SUPERIMPOSED DATA OVERFLOW	The total distribution amount of the CNC and PMC is too large during superimposed control for PMC axis control.
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.
0141	CAN NOT COMMAND G51 IN 3-D OFFSET	G51 (Scaling ON) is commanded in the 3-dimensional tool compensation mode. Modify the program.
0142	ILLEGAL SCALE RATE	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, coordinate rotation and cylindrical interpolation overflows the data storage. It also is generated during input of the manual intervention amount.
0144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or tool radius·tool nose radius compensation plane must be the same. Modify the program.
0145	ILLEGAL USE OF G12.1/G13.1	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	The modal G code group contains an illegal G code in the polar coordinate interpolation mode or when a mode was canceled. Only the following G codes are allowed: G40, G50, G69.1 An illegal G code was specified while in the polar coordinate interpolation mode. The following C codes are not allowed: G27, G28, G30, G30.1, G31 to G31.4, G37 to G387.3, G52, G92, G53, G17 to G19, G81 to G89, G68 In the 01 group, G codes other than G01, G02, G03, G02.2 and G03.2 cannot be specified.
0148	SETTING ERROR	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters Nos. 1710 to 1714.

Number	Message	Description
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	- For the tool management command H99 or D99 was specified when no tool management data number is assigned to the spindle position. Modify the program. - For the tool life management command 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 the machining program, the T code that is present in the block containing M06 does not correspond to the group currently being used. 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. <1> When different M codes are specified for path 1 and path 2 as waiting M codes without a P command. <2> When the waiting M codes are not identical even though the P commands are identical <3> When the waiting M codes are identical and the P commands are not identical (This occurs when a P command is specified with binary value.) <4> When the number lists in the P commands contain a different number even though the waiting M codes are identical (This occurs when a P command is specified by combining path numbers.) <5> When a waiting M code without a P command (2-path waiting) and a waiting M code with a P command (3-or-more-path waiting) were specified at the same time <6> When a waiting M code without a P command was specified for 3 or more paths.

Number	Message	Description
0161	ILLEGAL P OF WAITING M-CODE	P in a waiting M-code is incorrect. <1> When address P is negative <2> When a P value inappropriate for the system configuration was specified <3> When a waiting M code without a P command (2-path waiting) was specified in the system having 3 or more paths.
0163	ILLEGAL COMMAND IN G68/G69	G68 and G69 are not independently commanded in balance cut. An illegal value is commanded in a balance cut combination (address P).
0169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.
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. For the cylindrical interpolation axis, set not “0” but one of 5, 6 or 7 (parallel axis specification) to parameter No. 1022 to instruct the arc with axis of rotation (bit 1 (ROT) of parameter No. 1006 is set to “1” and parameter No. 1260 is set) ON.
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 mode or code G00 was instructed. Cancel the cylindrical interpolation mode before instructing code G00.
0177	CHECK SUM ERROR (G05)	A checksum error occurred.
0178	ILLEGAL COMMAND G05	The settings of bits 4 to 6 of parameter No.7501 are invalid or G05 was specified in any of the following mode. - Hypothetical axis interpolation (G07) - Cylindrical interpolation (G07.1) - Polar coordinate interpolation (G12.1) - Polar coordinates command (G16) - Spindle speed fluctuation detection (G26) - Tool radius · tool nose radius compensation (G41/G42) - Normal direction control (G41.1/G42.1) - Scaling (G51) - Programmable mirror image (G51.1) - Coordinate system rotation (G68) - Canned cycle (G81 to G89) - Constant surface speed control (G96) - Macro interruption(M96)
0179	PARAM. (NO.7510) SETTING ERROR	The number of controlled axes set by the parameter No. 7510 exceeds the maximum number. Modify the parameter setting value. The distribution of high-speed cycle machining or high-speed binary program operation stopped.
0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 5844.
0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, 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.

Number	Message	Description
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.
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. 2 The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.
0212	ILLEGAL PLANE SELECT	The direct drawing dimensions programming is commanded for the plane other than the Z-X plane. Correct the program.
0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In feed axis synchronization 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 (jog feed or incremental feed) 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. 4) Reference position setting with mechanical stopper was attempted for an axis under axis synchronous control with bit 1 (SFS) of parameter No.7180 = 0. Set bit 1 (SFS) of parameter No.7180 to 1. 5) Reference position setting with mechanical stopper was attempted with the manual handle feed axis select signal selected for the slave axis under axis synchronous control. Select the manual handle feed axis select signal for the master axis under axis synchronous control.
0214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
0217	DUPLICATE G51.2(COMMANDS)	G51.2 is further commanded in the G51.2 mode. Modify the program.

Number	Message	Description
0218	NOT FOUND P/Q COMMAND	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 diagnosis data No. 471.
0219	COMMAND G51.2/G50.2 INDEPENDENTLY	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	Polygon machining synchronous operation and axis 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	A reference 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.
0230	R CODE NOT FOUND	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. Five or more axes were specified as helical axes in the helical interpolation B mode.
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.
0241	ILLEGAL FORMAT IN G02.2/G03.2	The end point of an involute curve on the currently selected plane, or the center coordinate instruction I, J or K of the corresponding basic circle, or basic circle radius R was not specified.
0242	ILLEGAL COMMAND IN G02.2/G03.2	An illegal value was specified in the involute curve. The coordinate instruction I, J or K of the basic circle on the currently selected plane or the basic circle radius R is "0", or the start and end points are not inside the basic circle.
0243	OVER TOLERANCE OF END POINT	The end point is not positioned on the involute curve that passes through the start point, and this error exceeds the permissible error limit (parameter No. 5610).
0245	T-CODE NOT ALLOWED IN THIS BLOCK	One of the G codes, G50, G10, G04, G28, G28.2, G29, G30, and G30.2,G30.1,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.
0253	G05 CAN NOT BE COMMANDED	A binary operation was specified during advanced preview control mode.

Number	Message	Description
0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program. For the T system, one of the following functions is specified during scaling, this alarm is generated. - Finishing cycle (G70 or G72) - Outer surface rough-cutting cycle (G71 or G73) - End side rough-cutting cycle (G72 or G74) - Closed loop cutting cycle (G73 or G75) - End side cutting-off cycle (G74 or G76) - Outer surface or inner surface cutting-off cycle (G75 or G77) - Multiple repetitive threading cycle (G76 or G78) - Face drill cycle (G83 or G83) - Face tap cycle (G84 or G84) - Face boring cycle (G85 or G85) - Side drill cycle (G87 or G87) - Side tap cycle (G88 or G88) - Side boring cycle (G89 or G89) - Outer surface turning cycle or inner surface boring cycle (G77 or G20) - Threading cycle (G78 or G21) - End side turning cycle (G79 or G24) (Specify G codes for systems B and C in that order.)
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. - Since the one-rotation signal of the position detector is not caught, the manual reference position return grid is not established. (Bit 6 of diagnosis data No. 0201 must be set to 1.)
0303	REFERENCE POSITION RETURN IS NOT PERFORMED	When the setting of a reference position at any position was possible in Cs contour control (bit 0 (CRF) of parameter No. 3700 = 1), a G00 command was issued for the Cs contour axis without a return to the reference position after the serial spindle was switched to Cs contour control mode. Perform a reference position return with a G28 command before issuing a G00 command.
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	Although a G28 (automatic return to the reference position), G30 (return to the second, third, or fourth reference position), or G30/1 (return to the floating reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.
0306	MISMATCH AXIS WITH CNR/CHF	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	Reference position setting with mechanical stopper is being attempted for an axis which uses the "reference position setting without dog" function.

Number	Message	Description
0308	G72.1 NESTING ERROR	G72.1 was specified again during G72.1 rotation copying.
0309	G72.2 NESTING ERROR	G72.2 was specified again during G72.2 parallel copying.
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	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 (,) (bit 4 of parameter No. 3405 = 1) was specified for direct input of drawing dimensions, a comma was specified.
0313	ILLEGAL LEAD COMMAND	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.
0314	ILLEGAL SETTING OF POLYGONAL AXIS	An axis was specified invalidly in polygon turning. For polygon turning: A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning between spindles: Valid spindles are not specified. (Parameters Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected. For concurrent use of polygon turning and polygon turning with two spindles: - In the polygon turning mode, the value of parameter No. 7605 (selecting the type of polygon turning) was changed. - An attempt is made to use a spindle used for polygon turning also for polygon turning with two spindles.
0315	ILLEGAL NOSE ANGLE COMMAND IS IN THE THREAD CUTTING CYCLE	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	A 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	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	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	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	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	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	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).

Number	Message	Description
0323	THE FIRST BLOCK OF SHAPE PROGRAM IS A COMMAND OF TYPE II	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). Z (W) command is for G71. X (U) command is for G72.
0324	THE INTERRUPTION TYPE MACRO WAS DONE IN THE MULTIPLE REPETIVE CYCLES	An interruption type macro was issued during the multiple repetitive canned cycle (G70, G71, G72, or G73).
0325	UNAVAILABLE COMMAND IS IN SHAPE PROGRAM	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	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.
0327	MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE	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	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)	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	An axis other than the plane is specified n a canned cycle(G90, G92, or G94).
0331	ILLEGAL AXIS NUMBER IN AX[]	An illegal value is specified for an AX[] axis number.
0332	ILLEGAL AXIS ADDRESS IN AXNUM[]	An illegal value is specified for an AXNUM[] axis address.
0333	TOO MANY SPINDLE COMMANDS	Multiple spindle commands could be found in the same block in using an expansion spindle name. Only one spindle could be commanded in the same block.
0334	INPUT VALUE OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (malfunction prevention function)
0335	PLURAL M CODE	Multiple M codes are commanded simultaneously in a block for a wait function with peripheral devices by an M code.
0336	TOOL COMPENSATION COMMANDED MORE TWO AXES	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)
0340	ILLEGAL RESTART(NANO SMOOTHING)	With manual absolute turned on, an attempt was made to restart the operation in nano smoothing mode after performing the manual interaction.
0341	TOO MANY COMMAND BLOCK (NANO SMOOTHING)	There are more blocks than can be commanded consecutively in nano smoothing mode.
0342	CUSTOM MACRO INTERRUPT ENABLE IN NANO SMOOTHING	A custom macro interrupt was enabled in nano smoothing mode. Or, nano smoothing mode was commanded with a custom macro interrupt enabled.
0343	ILLEGAL COMMAND IN NANO SMOOTHING	G43, G44, or G49 was commanded during a nano smoothing.
0344	CANNOT CONTINUE NANO SMOOTHING	An illegal command or operation by which a nano smoothing could not be continued was performed.
0345	TOOL CHANGE ILLEGAL Z AXIS POS	A tool change position on the Z-axis is incorrect.

Number	Message	Description
0346	TOOL CHANGE ILLEGAL TOOL NUM	A tool change position is not set.
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.	An illegal synchronization control axis number (parameter No. 8180) is set.
0351	BECAUSE THE AXIS IS MOVING, THE SYNC CONTROL IS CAN'T BE USED.	While the axis being subject to synchronization control was moving, an attempt was made to start or cancel the synchronization control by a synchronization control axis selection signal.
0352	SYNCHRONOUS CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform synchronization control for the axis during a synchronization, composition, or superposition. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation. 3) An attempt was made to operate synchronization control although a parent-child-grandchild relation was not set.
0353	THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.	This error occurred when: - For synchronization 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the slave axis. - For composition 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the axis for which bit 7 (MUMx) of parameter No. 8162 is set to 1.
0354	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE	This error occurred when G28 was specified to the master axis being parking during synchronization 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.	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.	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.	This error occurred when an attempt was made to perform composite control for the axis during a synchronization, composition, or superposition.
0359	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE	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.	An illegal superposition control axis number (parameter No. 8186) is specified.
0361	BECAUSE THE AXIS IS MOVING, THE SUPERPOS CONTROL IS CAN'T BE USED.	While the axis being subject to superposition control was moving, an attempt was made to start or cancel the superposition control by a superposition control axis selection signal.

Number	Message	Description
0362	SUPERPOSITION CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform superposition control for the axis during a synchronization, composition, or superposition. 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.	This error occurred when G28 was specified to the superposition control slave axis during superposition control.
0364	THE G53 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G53 was specified to the slave axis being moved during superposition control.
0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The maximum control axis number or maximum control spindle number which could be used within a path was exceeded. (For a loader path, this alarm is generated if the number of axis per path is set to 5 or greater.)
0366	IMPROPER G-CODE IN TURRET METHOD	When the turret change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 0), G43, G43.1, G43.4, G43.5, or G43.7 was commanded.
0367	3-D CONV. WAS COMMANDED IN SYNC MODE AS THE PARAMETER PKUx(NO.8162#2) IS 0.	A three-dimensional coordinate conversion was commanded during synchronization control when the bit 2 (PKUx) of parameter No. 8162 was 0.
0368	OFFSET REMAIN AT OFFSET COMMAND	- When the ATC change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 1) during G43, G43.1, G43.4, or G43.5 mode, G43.7 was commanded. Or, G43, G43.1, G43.4, or G43.5 was commanded during G43.7 mode. - After bit 3 (TCT) of parameter No. 5040 was changed in the state in which a tool offset remained, another tool offset was specified.
0369	G31 FORMAT ERROR	- No axis is specified or tow or more axes are specified in the torque limit switch instruction (G31P98/P99). - The specified torque Q value in the torque limit switch instruction is out of range. The torque Q range is 1 to 254. - The high-speed continuous skip option is not present.
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. 4) <T series > The specified value of address P of G72 or G74 falls outside the 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.
0371	ILLEGAL FORMAT IN G10 OR L50	In a command format for a programmable parameter input, an attempt was made to change the parameter for an encryption (No. 3220), key (No. 3221), or protection range (No.3222 or No.3223) as a "the encryption function for the key and program." Modify the program.

Number	Message	Description
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.
0374	ILLEGAL REGISTRATION OF TOOL MANAGER(G10)	G10L75 or G10L76 data was registered during the following data registration: - From the PMC window. - From the FOCAS2. - By G10L75 or G10L76 in another system. Command G10L75 or G10L76 again after the above operation is completed.
0375	CAN NOT ANGULAR CONTROL(SYNC:MIX:OVL)	Angular axis control is disabled for this axis configuration. 1) When some related axes under angular axis control are not in synchronous control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in synchronous control 2) When some related axes under composite control are not in composite control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in composite control 3) When related axes under angular axis control is switched to superposition control mode1)
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 of parameter No. 1815 is set to “1”.)
0387	ILLEGAL RTM DI/DO VAR	There is no DI/DO variable that has a specified signal address (alphabet, number).
0389	ILLEGAL RTM SIGNAL BIT	Bits other than bits 0 to 7 cannot be specified with a DI/DO signal.
0390	ILLEGAL MACRO VAR	A macro variable which was not supported by the real time custom macro function was used.
0391	RTM BRANCH OVER	The number of branches supported with real time custom macros was exceeded.
0392	TOO MANY SENTENCE CONTROL	Many reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, G65, M99) for RTM control were used in a real time macro command.
0393	NO SENTENCE CONTROL	In a real time macro command, there is no data to be assigned.
0394	ILLEGAL SENTENCE CONTROL	The matching of reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, G65, M99) for RTM control is incorrect.
0395	ILLEGAL NC WORD CONTROL	Control code G65 or M99 for calling a subprogram or returning from a subprogram is not coded correctly.
0396	ILLEGAL RTM SENTENCE CONTROL	In other than a real time macro command, a reserved word (ZONCE, ZEDGE, ZWHILE, ZDO, or ZEND) for RTM control is used.

Number	Message	Description
0397	RTM BUFFER OVER	There is no buffer available for real time macro commands. Too many blocks read in advance are buffered as triggers used by real time macro commands.
0398	'ID OVER IN BUFFER	In blocks read in advance, there are too many real time macro commands with the same ID.
0399	'ID EXECUTION IN SAME TIME	An attempt was made to execute real time macro commands with the same ID by using the same NC statement as a trigger.
0400	ONESHOT CMDOVER	Too many one-shot real time macro commands are specified.
0401	EXEC CMD NUM OVER IN SAME TIME	The number of real time macro commands that can be executed simultaneously was exceeded
0402	ILLEGAL TOKEN FOR RTM	A token, variable, or function that is not supported by the real time custom macro function was detected.
0403	ACCESS TO RTM PROTECT VAR	An attempt was made to access a protected variable.
0404	RTM ERROR	An error related to a real time macro command occurred.
0406	CODE AREA SHORTAGE	The storage size of the real time macro area is insufficient.
0407	DOULE SLASH IN RTM MODE	In the compile mode, an attempt was made to set the compile mode again.
0408	G90 IS NOT PERMITTED	The absolute command cannot be specified.
0409	ILLEGAL AXIS NO	An invalid axis number is specified.
0410	MIDDLE POINT IS NOT ZERO	An intermediate point other than 0 is specified with G28.
0411	SIMULTANEOUSLY AXES OVER	The maximum number of axes that can be controlled simultaneously was exceeded.
0412	ILLEGAL G CODE	An unusable G code was used.
0413	ILLEGAL ADDRESS	An unusable address was used.
0414	ILLEGAL PMC AXIS NO.	An invalid PMC axis number is specified.
0415	GROUP IS IN USE	The group to which the specified axis belongs is already in used.
0416	UNABLE TO USE THE AXIS	The specified axis cannot be used.
0417	AXIS IS UNABLE TO MOVE	The specified axis is placed in the inoperative state.
0418	ILLEGAL FEED SETTING	An incorrect feedrate is set.
0419	ILLEGAL DISTANCE SETTING	A travel distance beyond the specifiable range is specified.
0420	CONSTANT NUMBER P	A subprogram is specified not by using a constant.
0421	ILLEGAL ARGUMENT G54	With G65, an invalid argument, L, is used.
0422	ILLEGAL ARGUMENT G54	With G65, an invalid argument is used.
0423	NO PMC AXIS CONTROL OPTION	The option for PMC axis control is missing.
0424	MULTIPLE AXES IN ONE GROUP	Multiple axes are using one group.
0425	ONE AXIS USE MULTIPLE GROU	One axis is using multiple groups.
0429	ILLEGAL COMMAND IN G10.6	When retract was started in a threading block, a retract command had been issued for the long axis direction of threading.
0430	TOOL LIFE PAIRS ZERO	Tool life management group number parameter No.6813 is 0.
0431	ILLEGAL T/R DATA OF TOOL LIFE	The arbitrary group number (T) or remaining amount setting (R) is invalid.
0432	UNAVAILABLE POSTURE IN TPC	- A tool posture that cannot be assumed under tool posture control was specified. Check the machine configuration and specification. - A command that changes the direction of the tool posture in relation to the interpolation plane was specified in circular interpolation or helical interpolation during tool posture control. Check the machine configuration and the command.

Number	Message	Description
0436	ILLEGAL PARAMETER IN WSC	An incorrect parameter was specified in compensation of workpiece placement error. - The basic three axes are not specified in parameter No.1022.
0437	ILLEGAL COMMAND IN WSC	An invalid command related to compensation of workpiece placement error was specified. - An illegal G code was specified in the workpiece placement error compensation mode. - There is an error in the modal setting used when the compensation of workpiece placement error is started. - G54.4 was not specified solely. - There is not the P command in the block including the G54.4 command. Alternatively, the value following P is out of the range. - Compensation of workpiece placement error was specified redundantly.
0438	ILLEGAL PARAMETER IN TOOL DIRC CMP	If, on a 5-axis machine, either of the two cases below applies, a parameter is illegal. <1> The setting is such that tool direction compensation is performed if workpiece setting error compensation is performed (bit 0 (RCM) of parameter No. 11200 = 1). <2> Tool center point retention type tool axis direction control (G53.6) is performed. - Acc./Dec. before interpolation is disabled. Set parameter No. 1660. - Acc./Dec. before rapid traverse interpolation is disabled. Set bit 1 of parameter No. 1401, bit 5 of parameter No. 1950, and parameter No. 1671. - The parameters Nos.19680 to No.19714 for configuring the machine are incorrect. - The axis set by parameters Nos. 19681 and 19686 is not a rotation axis. - The basic three axes are not set in parameter No. 1022. - In tool length compensation during workpiece setting error compensation, bit 6 (TOS) of parameter No. 5006 is 0 and bit 2 (TOP) of parameter No. 11400 is 0. Set either parameter to 1.
0439	ILLEGAL COMMAND IN TOOL DIRC CMP	When compensation of workpiece placement error was performed in a 5-axis cutting machine (compensation in the tool direction (bit 0 (RCM) of parameter No. 11200 is 0)), an illegal command was issued. - An unspecifiable G code was specified. - There is an error in the modal setting used during startup. - An axis not related to 5-axis machining was specified. - The absolute coordinates of a rotation axis could not be obtained in the startup block of compensation of workpiece placement error or tool center point control.
0441	DUPLICATE PATH TABLE	The same Path Table numbers exist. Example) <AXIS_TABLE_1234_X1> and <TIME_TABLE_1234_X1> exist. <AXIS_TABLE_0001_M> and <TIME_TABLE_0001_M> exist.

Number	Message	Description
0442	PATH TABLE COMMAND EXCES ERR	1. At the start of the Path Table Operation, the difference between the actual axis position and the start command at the Path Table exceeds the parameter No.11101. 2. At the start of the Path Table Operation, the difference between the actual spindle speed and the start command of spindle speed at the Path Table exceeds the parameter No.11102.
0443	PTRDY SIGNAL IS OFF	Even though the Path Table Ready signal PTRDY <Fn519.6> is "0", the Path Table Operation is started. Retry Path Table Operation after Path Table conversion.
0444	ILLEGAL PATH TABLE M-CODE	M/P/Q code for starting the Path Table operation is not correct.
0445	ILLEGAL AXIS OPERATION	The positioning command was issued in the speed control mode. Check the SV speed control mode in-progress signal.
0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	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 live tool axis is incorrectly set. Check the parameter for spindle control with servo motor.
0451	ILLEGAL AUXILIARY FUNCTION TABLE COMMAND	When M code is output, the ladder of the PMC does not execute the completion processing of the previous M code.
0452	ILLEGAL PATH TABLE OPERATION	In Path Table Operation, the following problems occurred. - Skip command is not correct. - The connection of the Path Table is not correct. - Path Table Operation is not correct for other reason. etc. The detail alarm number is read by using the cnc_rdptexedistalm function. The detail alarm number can be read by using the C Language Executor or FOCAS2 cnc_ rdptexedistalm function. For details of cnc_rdptexedistalm function, refer to “CNC/PMC window library” in “C Language Executor Programming Manual (B-63943EN-3).
0455	ILLEGAL COMMAND IN GRINDING	In grinding canned cycles: 1) <M series> The signs of the I, J, and K commands do not match. 2) <M series/T series > The amount of travel of the grinding axis is not specified.
0456	ILLEGAL PARAMETER IN GRINDING	Parameters related to grinding canned cycles are incorrectly set. Probable causes are given below. 1) <M series/T series> The axis number of the grinding axis is incorrectly set (parameters Nos. 5176 to 5179). 2) <M series> The axis number of the dressing axis is incorrectly set (parameters Nos. 5180 to 5183). 3) <M series/T series> The axis numbers of the cut axis, grinding axis, and dressing axis (only for the M series) overlap.
0459	ALL PARALLEL AXES IN PARKING	All the axes specified during automatic operation are parking.
0460	ILLEGAL TORCH AXIS NUMBER	The axis number set in parameter No. 5490 (torch control axis) exceeds the number of control axes.
0461	ILLEGAL SETTING OF ROTATE AXIS FOR TORCH	The parameter setting (bit 0 of parameter No. 1006 = 1) of the rotation axis is not applied to the torch turning axis.
0492	3DCHK FIG. ILLEGAL: [Target name]	The figure data of [Target name] specified for the built-in 3D interference check is invalid.
0493	3DCHK AXIS ILLEGAL: [Target name]	The move axis data of [Target name] specified for the built-in 3D interference check is invalid.

Number	Message	Description
0494	3DCHK FUNCTION INVALID	The 3D interference check function is disabled by bit 0 (ICE) of parameter No. 10930.
0495	3DCHK TOO MANY FIGURE	The total number of shapes included in all interference check targets except the tool exceeds 23.
0496	ILLEGAL P,Q COMMAND IN G22.2	As for G22.2 command, parameter P or Q is out of range or not specified. Please correct G22.2 command.
0497	CANNOT MAKE TOOL FIGURE BY TOOL MANAGEMENT	According to bit 2 (ICT) of parameter No.10930, though the tool figure should be automatically made by using the tool management function, the option for the tool management function is not available.
0501	THE COMMANDED M-CODE CAN NOT BE EXECUTED	The M code specified in parameter No. 11631 to 11646 was specified in other than an execution macro, macro interrupt, macro call using a G or M code, or subprogram call using a T, S, or second auxiliary function code.
0502	ILLEGAL G-CODE	1) A G code unavailable in the inter-path flexible synchronous mode was specified. 2) A G code unavailable in the advanced superimposition state was specified.
0503	ILLEGAL MODAL IN SUPERIMPOSED MODE	Modal state of a G code that cannot be superimposed.
0507	ILLEGAL PARAMETER(NO.7526)	The address of the R signal for the high-speed cycle machining operation information output function is invalid. 1) The specified R signal address is invalid. 2) The start address is not a multiple of 4 (0, 4, 8, ...). 3) A 36-byte area is not allocated.
0508	G code to need G90(PAC)	In parallel axis control, a G code requiring an absolute command (G90) in the block immediately before was specified.
0509	TOOL OFFSET COMMAND IS NOT AVAILABLE	- Tool offset (for the lathe system) was specified in the thread cutting block. - Tool offset was specified (lathe system) in the scaling mode, coordinate system rotation mode, or programmable mirror image mode.
0511	CS HI-SPEED SWITCHING FORMAT ERROR	The format of Cs contour control high speed switching is invalid.
0512	IMPOSSIBLE COMMAND FOR CS HI-SPEED SWITCHING	The following commands cannot be specified in Cs contour control high speed switching: - Move command not for high-speed cycle machining - Synchronous/composite control, superimposed control - Simple spindle synchronous control - Simple spindle electronic gear box - Manual reference position return
0513	CS HI-SPEED SWITCHING SETTING ERROR	The setting for Cs contour control high speed switching is invalid. Possible causes are: - An M code value for Cs contour control high speed switching is used for multiple Cs contour control axes. - FIN is returned for the M code for high-speed switching of Cs contour control when the Cs contour control high speed switching completion signal CSMCx does not become 1. - The spindle software does not support the spindle control switching function for high-speed cycle machining.

Number	Message	Description
0514	ILLEGAL COMMAND IN FLEXIBLE PATH AXIS ASSIGNMENT	1) An assignment command in flexible path axis assignment was issued for an axis yet to be removed. 2) The P, Q, R, I, J, K, or L value specified by G52.1, G52.2, or G52.3 is invalid. 3) The value of the parameter No. 11560 is duplicated. 4) An attempt was made to execute a removal command (G52.1) for an axis already removed. 5) An attempt was made to exchange axes having different settings of bit 1 (FAN) of parameter No. 11562. 6) An attempt was made to perform flexible path axis assignment without canceling the offset.
0515	ILLEGAL FORMAT IN SMOOTH TCP(G43.4L1)	An illegal command was specified in smooth TCP. • An illegal command was specified in a smooth TCP start block. - An invalid value was specified with address "L". A value other than 0 and 1 was specified with address "L". - G10.8 was specified at the same time.
0516	ILLEGAL PARAMETER IN SMOOTH TCP(G43.4L1)	A parameter related to smooth TCP is illegal. • On a machine whose axis configuration is table rotation type or composite type, when the setting was such that the workpiece coordinate system was used as the programming coordinate system (bit 5 (WKP) of parameter No.19696 = 1), smooth TCP was specified • On a machine whose axis configuration is table rotation type or composite type, address "L" was omitted in a TCP start block when the setting was such that smooth TCP would start if address "L" was omitted in the TCP start block (bit 0 (STC) of parameter No. 10485 = 1), but the setting is such that the workpiece coordinate system is used as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 = 1).
0517	SETTING ERROR AMOUNT IS OUT OF RANGE	An attempt was made to start workpiece setting error compensation when a rotation direction setting error was outside the range set in the corresponding parameter No. 11753 to 11758.
0520	ILLEGAL FORMAT IN G10.8L1	An illegal command was specified to change a tolerance of smooth TCP. • A negative value was specified as a tolerance. - Specify positive values as addresses "α" and "β". • An invalid P value was specified. - Specify either 0 or 1 as address "P". • Address P is specified together with addresses "α" and "β". - Specify only either address "P" or "addresses "α" and "β". • An invalid address was specified. - In G10.8L1, only L, P, α, β, O, N, and M can be specified. • Another G code was specified at the same time. - Specify G10.8L1 alone. • G10.8 was specified in smooth TCP mode, but address "L" is not specified or the value of address "L" is not 1. - In smooth TCP mode, only G10.8L1 can be specified.

Number	Message	Description
0521	ILLEGAL USAGE OF G10.8L1	Modal information used when specifying G10.8L1 contains an error. • The system is not in smooth TCP mode. - G10.8L1 can be specified in smooth TCP mode only. • The system is in smooth TCP mode, but the command is not linear interpolation (G01). - G10.8L1 can be specified only during linear interpolation (G01).
1001	AXIS CONTROL MODE ILLEGAL	Axis control mode is illegal.
1013	ILLEGAL POS. OF PROGRAM NO.	Address O or N is specified in an illegal location (e.g. after a macro statement).
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.
1059	COMMAND IN BUFFERING MODE	The manual intervention compensation request signal MIGET became "1" when a advanced block was found during automatic operation. To input the manual intervention compensation during automatic operation, a sequence for manipulating the manual intervention compensation request signal MIGET is required in an M code instruction without buffering.
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.
1090	PROGRAM FORMAT ERROR	A lowercase alphabetic character is found in other than an NC program statement comment section, program name, or folder name.
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. [#_OFSxx] does not match the tool offset memory option configuration.
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.

Number	Message	Description
1100	CANCEL WITHOUT MODAL CALL	Call mode cancel (G67) was specified even though macro continuous–state call mode (G66) was not in effect.
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 99999999).
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	The G10 L No. contains no relevant data input or corresponding option. 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.
1145	G10.1 TIME OUT	The response to a G10.1 instruction was not received from the PMC within the specified time limit.
1146	G10.1 FORMAT ERROR	The G10.1 instruction format is in error.
1152	G31.9/G31.8 FORMAT ERROR	The format of the G31.9 or G31.8 block is erroneous in the following cases: - The axis was not specified in the G31.9 or G31.8 block. - Multiple axes were specified in the G31.9 or G31.8 block. - The P code was specified in the G31.9 or G31.8 block.

Number	Message	Description
1153	CANNOT USE G31.9	G31.9 cannot be specified in this modal state. This alarm is also generated when G31.9 is specified when a group 07 G code (e.g. tool radius compensation) is not canceled.
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	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. If the zero point of the drilling axis is not specified or parallel axes are specified in a block containing a G code in a canned cycle, simultaneously specify the parallel axes for the drilling axis.
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. 1841 before the deceleration limit switch is left (deceleration signal *DEC returns to “1”).
1202	NO F COMMAND AT G93	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	1) An attempt was made to execute an instruction that uses the spindle although the spindle to be controlled has not been set correctly. 2) Interpolation type rigid tapping was specified in a path in which the Cs contour control function is not enabled.
1282	ILLEGAL COMMAND IN 3-D OFFSET	An illegal G code was specified in the 3-dimensional tool compensation mode.
1283	ILLEGAL IJK IN 3-D OFFSET	When bit 0 (ONI) of parameter No. 6029 is set to 1, I, J, and K commands are specified without the decimal point in 3-dimensional tool compensation mode.
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. An invalid address R value is specified in a pattern program for each machining purpose on the high–speed high–precision setting screen. This alarm is also generated when illegal word values are found.

Number	Message	Description
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.
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.
1360	PARAMETER OUT OF RANGE (TLAC)	Illegal parameter setting. (Set value is out of range.)
1361	PARAMTER SETTING ERROR 1 (TLAC)	Illegal parameter setting. (axis of rotation setting)
1362	PARAMETER SETTING ERROR 2 (TLAC)	Illegal parameter setting (tool axis setting)
1370	PARAMETER SETTING ERROR (DM3H-1)	Out–of–range data was set during setting of the three–dimensional handle feed parameter.
1371	PARAMETER SETTING ERROR (DM3H-2)	An illegal axis of rotation was set during setting of the three–dimensional handle feed parameter.
1372	PARAMETAR SETTING ERROR (DM3H-3)	An illegal master axis was set during setting of the three–dimensional handle feed parameter.
1373	PARAMETER SETTING ERROR (DM3H-4)	An illegal parallel axis or twin table was set during setting of the three–dimensional handle feed parameter.
1470	G40.1 –G42.1 PARAMETER MISS	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 (bit 0 (ROTx) of parameter No. 1006) = 1 and No.1022=0).

Number	Message	Description
1471	ILLEGAL COMMAND IN G40.1 –G42.1	A G code which cannot be specified in the normal direction control mode was specified.
1508	DUPLICATE M-CODE (INDEX TABLE REVERSING)	A function to which the same code as this M code is set exists. (index table indexing)
1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	A function to which the same code as this M code is set exists. (spindle positioning, orientation)
1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	A function to which the same code as this M code is set exists. (spindle positioning, positioning)
1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
1531	ILLEGAL USE OF DECIMAL POINT (F-CODE)	When the feedrate instruction contains valid data below the decimal point, the alarm is set and the F code contains valid data below the decimal point.
1532	ILLEGAL USE OF DECIMAL POINT (E-CODE)	When the feedrate instruction contains valid data below the decimal point, the alarm is set and the E code contains valid data below the decimal point.
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 (G95).
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 (G95).
1535	ADDRESS E UNDERFLOW (G95)	The feedrate for the hole drilling axis calculated from the E and S codes is too slow in the feed per single rotation mode (G95).
1536	ADDRESS E OVERFLOW (G95)	The feedrate for the hole drilling axis calculated from the E and S codes is too fast in the feed per single rotation mode (G95).
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.
1539	ADDRESS E UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the E instruction is too slow.
1540	ADDRESS E OVERFLOW (OVERRIDE)	The speed obtained by applying override to the E instruction is too fast.
1541	S-CODE ZERO	“0” has been instructed as the S code.
1542	FEED ZERO (E-CODE)	“0” has been instructed as the feedrate (E code).
1543	ILLEGAL GEAR SETTING	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 axis/Cs contour control axis was specified during switching of the controlled axis mode.
1561	ILLEGAL INDEXING ANGLE	The specified angle of rotation is not an integer multiple of the minimum indexing angle.
1564	INDEX TABLE AXIS – OTHER AXIS SAME TIME	The index table indexing axis and another axis have been specified in the same block.
1567	INDEX TABLE AXIS DUPLICATE AXIS COMMAND	Index table indexing was specified during axis movement or on an axis for which the index table indexing sequence was not completed.

Number	Message	Description
1580	ENCODE ALARM (PSWD&KEY)	When an attempt was made to read a program, the specified password did not match the password on the tape and the password on tape was not equal to 0. When an attempt was made to punch an encrypted tape, the password was not in the range 0 to 99999999. The password parameter is No. 2210.
1581	ENCODE ALARM (PARAMETER)	When an attempt was made to punch an encrypted tape, the punch code parameter was set to EIA. Set bit 1 (ISO) of parameter No. 0000 to “0”. An incorrect instruction was specified for program encryption or protection. This alarm is generated if an attempt is made to perform program editing, deletion, or range-specified punch-out in the protected range in the lock state. Or, a program outside the protected range is specified in rage specification punch-out in the unlock state. The protected range is defined from the program No. preset by parameter No. 3222 up to the program No. preset to parameter No. 3223. When both parameters are set to “0”, the protected range becomes O9000 to O9999.
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	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) or (No. 7782 or 7783) is not set.) (4) For a hobbing-machine-compatible command, parameter No. 7710 is not specified. (5) The slave axis commanded by the G81 is the slave axis of simple spindle EGB. (6) No signal-based EGB synchronization ratio (parameters Nos. 7784 and 7785) has been set.
1594	EGB FORMAT ERROR	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. (4) In the G81.5 block, there is no command for the master or slave axis. (5) In the G81.5 block, data out of the specified range is specified for the master or slave axis.

Number	Message	Description
1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30,G30.1, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc. (3) Synchronization start command using G81 or G81.5 when bit 3 (ECN) of parameter No. 7731 is 0 (4) For the slave axis of the spindle EGB, the Cs contour control mode is not selected.
1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.
1597	EGB AUTO PHASE FORMAT ERROR	Format error in the G80 or G81 block in EGB automatic phase synchronization (1) R is outside the permissible range. (2) In spindle EGB, before the G81R2 command, the reference point return is not executed on the master spindle.
1598	EGB AUTO PHASE PARAMETER SETTING ERROR	Error in the setting of a parameter related to EGB automatic phase synchronization (1) The acceleration/deceleration parameter is not correct. (2) The automatic phase synchronization parameter is not correct.
1618	ILLEGAL P-DATA(WHEEL WEAR COMPENSATION)	There is an error in P-data in compensation selection of grinding wheel wear compensation. Alternatively, the P command is not present.
1619	ILLEGAL AXIS(WHEEL WEAR COMPENSATION)	The compensation axis was switched in the grinding wheel wear compensation mode or compensation vector hold mode. Alternatively, parameters Nos. 5071 and 5072, which determine the axis to be subjected to grinding wheel wear compensation, are incorrectly set.
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 Zero Return] The P address Nos. for instructing No. 2 to No. 4 zero return are each out of the range 2 to 4. [Single Rotation Dwell] The specified spindle rotation is “0” when single rotation dwell is specified. [3-dimensional tool compensation] A G code that cannot be specified was specified in the 3-dimensional tool compensation mode. Scaling instruction G51, skip cutting G31 and automatic tool length measurement G37 were 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 I/O interface option that has not yet been added on was specified. The external I/O device and baud rate, stop bit and protocol selection settings are erroneous.
1808	DEVICE DOUBLE OPENED	An attempt was made to open a device that is being accessed.

Number	Message	Description
1809	ILLEGAL COMMAND IN G41/G42	Specified direction tool length compensation parameters are incorrect. A move instruction for a axis of rotation was specified in the specified direction tool length compensation mode.
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 an M code for performing preset with an each axis workpiece coordinate system preset signal was specified, the each axis workpiece coordinate system preset signal was not turned “1”. 3. The auxiliary function lock is enabled. 4. When bit 6 (PGS) of parameter No. 3001 was set to 0 (M, S, T, and B codes are not output in the high speed program check mode), an M code for turning “1” an each axis workpiece coordinate system preset signal in the high speed program check mode was specified.
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.
1889	ILLEGAL COMMAND IN G54.3	An illegal command was issued in G54.3 block. (1) An attempt was made to command G54.3 in a mode in which it cannot be accepted. (2) The command was not issued in a single block.
1892	ILLEGAL PARAMETER IN G43.3	A parameter related to nutating rotary head tool length compensation is incorrect.
1893	ILLEGAL PARAMETER IN G44.9	A parameter related to spindle unit compensation is incorrect.
1898	ILLEGAL PARAMETER IN G54.2	An illegal parameter (Nos. 6068 to 6076) was specified for fixture offset.
1912	V-DEVICE DRIVER ERROR (OPEN)	An error occurred during device driver control.
1919	FATAL ERROR(USB MEMORY)	A fatal error occurred in the USB file system. To restore the file system, turn the power off.
1924	UNEXPECTED ERROR(USB MEMORY)	An unexpected error occurred in the USB file system.
1925	ILLEGAL PATH/FILE(USB MEMORY)	An invalid path or file name was specified.
1926	ACCESS DENIED(USB MEMORY)	The USB memory could not be accessed.
1927	DEVICE IN FORMATTING(USB MEMORY)	The USB memory is being formatted.
1928	DEVICE NOT FOUND(USB MEMORY)	No USB memory is inserted. Check the connection.

Number	Message	Description
1930	ILLEGAL COMMAND AFTER RESTART	The restart block does not satisfy either of the following conditions: (1) An absolute command is specified in the block. (2) The G00 or G01 command is specified in the block. Select a block satisfying conditions (1) and (2) as the restart block.
1931	ILLEGAL MODE AFTER RESTART	Suppress motion is specified in a mode in which suppress motion is not available. Select a block in a mode in which suppress motion is available as the restart block.
1932	DEVICE IS FULL(USB MEMORY)	The capacity of the USB memory is insufficient.
1937	RECOGNITION ERROR(USB MEMORY)	The format of the USB memory is invalid. Format the USB memory in FAT or FAT32 format. If the alarm is still issued, replace the USB memory.
1938	END OF FILE FOUND(USB MEMORY)	The end of file was detected before EOR(%) was read. The file may be damaged.
1939	UNDEFINED ERROR(USB MEMORY)	An undefined error occurred.
1951	DEVICE IS BUSY(USB MEMORY)	The USB memory is busy.
1952	TOO MANY FILES(USB MEMORY)	The maximum number of files that can be opened concurrently is exceeded.
1953	REMOVED IN ACCESSING(USB MEMORY)	The USB memory was removed while being accessed.
1954	PATH/FILE EXIST(USB MEMORY)	The specified path or file already exists.
1955	PATH/FILE NOT FOUND(USB MEMORY)	The specified path or file is not found.
1956	DEVICE OVERCURRENT(USB MEMORY)	Overcurrent was detected in the USB memory. Replace the USB memory.
1957	PARITY ERROR(USB MEMORY)	A parity error occurred in the USB memory. Turn the power to the CNC off.
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.
1990	SPL:ILLEGAL AXIS COMMAND	The axis specified by the smooth interpolation (G5.1Q2) is illegal.

Number	Message	Description
1993	SPL:CAN'T MAKE VECTOR	The end point and the 2 previous point are the same in generation of the 3-dimensional tool compensation vector by the end point for smooth interpolation.
1995	ILLEGAL PARAMETER IN G41.2/G42.2	The parameter settings (parameters Nos. 6080 to 6089) for determining the relationship between the axis of rotation and the rotation plane are incorrect.
1999	ILLEGAL PARAMETER IN G41.3	The parameter settings (parameters Nos. 6080 to 6089) for determining the relationship between the axis of rotation and the rotation plane are incorrect.
2002	NO KNOT COMMAND (NURBS)	Knot has not been specified, or a block not related to NURBS interpolation was specified in the NURBS interpolation mode.
2003	ILLEGAL AXIS COMMAND (NURBS)	An axis not specified as a control point was specified in the No. 1 block.
2004	ILLEGAL KNOT	There is an insufficient number of knot individual blocks.
2005	ILLEGAL CANCEL (NURBS)	The NURBS interpolation mode was turned OFF even though NURBS interpolation was not completed.
2006	ILLEGAL MODE (NURBS)	A mode that cannot be paired with the NURBS interpolation mode was specified.
2007	ILLEGAL MULTI-KNOT	Nested knots for each level can be specified for the start and end points.
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.
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	THE NUMBER OF #30000 IS UNMATCH	An attempt was made to enter a P-CODE-only variable not existing in the system.
2054	THE NUMBER OF #40000 IS UNMATCH	An attempt was made to enter an extended P-CODE-only variable not existing in the system.
2060	ILLEGAL PARAMETER IN G43.4/G43.5	The parameter for the pivot tool length compensation is incorrect.
2061	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I, J, or K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02 mode. - G43.4 or G43.5 was specified in the tool center point control mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane.
2070	G02.1/ G03.1 FORMAT ERROR	- The format is invalid. - The specified arc exceeds the interpolation enable range.
4010	ILLEGAL REAL VALUE OF OBUF :	The real value for a output buffer is in error.

Number	Message	Description
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 rate parameter No. 1410 or the parameter for the maximum cutting feedrate for each axis is 0. The parameter for the maximum cutting feedrate for each axis is No. 1432 if acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause acceleration/deceleration before interpolation include AI contour control, tool center point control, and workpiece setting error compensation.
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 setting of the parameter for the maximum cutting feedrate is 0. The parameter is No. 1432 if acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause acceleration/deceleration before interpolation include AI contour control, tool center point control, and workpiece setting error compensation.
5014	TRACE DATA NOT FOUND	A transfer could not be made because of no trace data.
5015	NO ROTATION AXIS	No rotation axis was found in a handle feed in the tool axis direction or in the tool axis right angle direction.
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	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 diagnosis data No. 0471.
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.
5043	TOO MANY G68 NESTING	Three-dimensional coordinate conversion has been specified three or more times. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.

Number	Message	Description
5044	G68 FORMAT ERROR	Errors for three-dimensional coordinate conversion command are: (1) No I, J, or K command was issued in three-dimensional coordinate conversion command block. (without coordinate rotation option) (2) All of I, J, or K command were 0 in three-dimensional coordinate conversion command block. (3) No rotation angle R was not commanded in three-dimensional coordinate conversion command block.
5046	ILLEGAL PARAMETER (S-COMP)	The setting of a parameter related to straightness compensation contains an error. Possible causes include: - A non-existent axis number is set in a moving or compensation axis parameter. - More than 128 pitch error compensation points are set between the furthest points in the negative and position regions. - The straightness compensation point numbers do not have correct magnitude relationships. - No straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. - The compensation per compensation point is either too large or too small.
5050	ILL-COMMAND IN G81.1 MODE	During chopping, a move command has been issued for the chopping axis.
5058	G35/G36 FORMAT ERROR	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5060	ILLEGAL PARAMETER IN G02.3/G03.3	The axis parameter setting to perform an exponential interpolation is in error. Parameter No. 5641: A liner axis number for performing an exponential interpolation Parameter No. 5642: A rotation axis number for performing an exponential interpolation The settable value is 1 to the number of control axes, but it must not be duplicated.
5061	ILLEGAL FORMAT IN G02.3/G03.3	The exponential interpolation command (G02.3/G03.3) has a format error. The command range for address I or J is -89.0 to -1.0 or +1.0 to +89.0. No I or J is specified or out-of -range value is specified. No address R, or 0 is specified.
5062	ILLEGAL COMMAND IN G02.3/G03.3	The value specified in an exponential interpolation command (G02.3/03.3) is illegal. A value that does not allow exponential interpolation is specified. (For example, the value for In is 0 or negative.)
5064	DIFFERRENT AXIS UNIT	Circular interpolation has been specified on a plane consisting of axes having different increment systems.
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.

Number	Message	Description
5066	RESTART ILLEGAL SEQUENCE NUMBER	A sequence number from 7000 to 7999 was read during the search for the next number in a restart program for the back or restart function.
5068	FORMAT ERROR IN G31P90	No travel axis was specified. Two or more travel axes were specified.
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.
5085	SMOOTH IPL ERROR 1	A block for specifying smooth interpolation contains a syntax error.
5110	IMPROPER G-CODE (AICC MODE)	An unspecifiable G code was specified in the AI contour control mode.
5115	ILLEGAL ORDER (NURBS)	There is an error in the specification of the rank.
5116	ILLEGAL KNOT VALUE (NURBS)	Monotone increasing of knots is not observed.
5117	ILLEGAL 1ST CONTROL POINT (NURBS)	The first control point is incorrect. Or, it does not provide a continuity from the previous block.
5118	ILLEGAL RESTART (NURBS)	After manual intervention with manual absolute mode set to on, NURBS interpolation was restarted.
5122	ILLEGAL COMMAND IN SPIRAL	A spiral interpolation or conical interpolation command has an error. Specifically, this error is caused by one of the following: 1) L = 0 is specified. 2) Q = 0 is specified. 3) R/, R/, C is specified. 4) Zero is specified as height increment. 5) Zero is specified as height difference. 6) Three or more axes are specified as the height axes. 7) A height increment is specified when there are two height axes. 8) Q is specified when radius difference = 0. 9) Q < 0 is specified when radius difference > 0. 10) Q > 0 is specified when radius difference < 0. 11) A height increment is specified when no height axis is specified.
5123	OVER TOLERANCE OF END POINT IN SPIRAL	The difference between a specified end point and the calculated end point exceeds the allowable range (parameter 3471).
5124	CAN NOT COMMAND SPIRAL	A spiral interpolation or conical interpolation was specified in any of the following modes: 1) Scaling 2) Polar coordinate interpolation 3) In tool radius·tool nose radius compensation mode, the center is set as the end point.
5130	NC AND SUPERIMPOSE AXIS CONFLICT	In the PMC superposition axis control, the NC command and The PMC axis control command were conflicted. Modify the program and the ladder.
5131	NC COMMAND IS NOT COMPATIBLE	The PMC axis control and three-dimensional coordinate conversion or a polar coordinate interpolation were specified simultaneously.
5132	CANNOT CHANGE SUPERIMPOSED AXIS	The superposition axis was selected for the axis for which the PMC superposition axis is being controlled.
5155	NOT RESTART PROGRAM BY G05	When learning control/preview repetitive control was enabled, an attempt was made to use feed hold or interlock to stop high-speed cycle machining/high-speed binary operation. Neither feed hold nor interlock can be used in such a case.

Number	Message	Description
5195	DIRECTION CAN NOT BE JUDGED	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. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. 2. The tool is moving along the two axes (X-axis and Y-axis). [For the movement direction discrimination specification] 1. The recorded pulse direction is not constant. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. 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.
5196	ILLEGAL AXIS OPERATION	During HPCC or during the execution of a 5-axis-related function, an unavailable function was used.
5199	ILLEGAL FINE TORQUE SENSING PARAMETER	A parameter for fine torque sensing is incorrectly set. - The control axis number of the target axis is invalid.
5211	ILLEGAL AXIS OPERATION	In servo spindle synchronization mode, a servo axis command was executed from the CNC. Correct the program.
5219	CAN NOT RETURN	Manual intervention and return cannot be performed during execution of three-dimensional coordinate system conversion, tilted working plane command, tool center point control, or work setting error compensation.
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.
5242	ILLEGAL AXIS NUMBER	A master axis number or a slave axis number was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation. In inter-path flexible synchronous control, this alarm is issued in either of the following cases. (The alarm is issued at the start of inter-path flexible synchronous control.) 1. The axis number of the master or slave axis is incorrect. 2. The master and slave axis settings make a loop.
5243	DATA OUTRANGE	A gear ratio was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation.
5244	TOO MANY DI ON	• When an attempt was made to change the flexible synchronous control status, the select signal was not turned on or off after the execution of the M code. • An attempt was made to turn flexible synchronous control on or off without stopping the tool along all axes. (Except when automatic phase synchronization for flexible synchronous control is used) • Flexible synchronous control was turned off in any of the following function modes: - Tool center point control - Tilted working plane command - 3-dimensional cutter compensation - Workpiece setting error compensation

Number	Message	Description
5245	OTHERAXIS ARE COMMANDED	- For a flexible synchronization control group for which a PMC axis was a master axis, an attempt was made to turn on the synchronous mode during time other than automatic operation. - An attempt was made to turn on a synchronization group for which an PMC axis was a master axis when there existed a flexible synchronization control group for which a non-PMC, normal axis was a master axis. - The master and slave axes as synchronous axes overlap the EGB dummy axis. - The master and slave axes as synchronous axes overlap the chopping axis. - The master and slave axes as synchronous axes overlap the axis related to angular axis control. - The master and slave axes as synchronous axes overlap the axis related to composite control. - The master and slave axes as synchronous axes overlap the axis related to superposition control. - The slave axis as a synchronous axis overlaps the axis related to synchronization control. - The reference position return mode is turned on (was turned on). - Over travel alarm occurs on slave axis. - A servo alarm occurred in a path in inter-path flexible synchronous control. - An emergency stop was applied in another path in inter-path flexible synchronous control. - When an attempt was made to execute flexible synchronization between different paths during automatic operation, the inter-path flexible synchronous mode was not enabled.
5255	G12.4/G13.4 FORMAT ERROR	The specified P, I, and K are incorrect or I is less than K.
5256	G12.4/G13.4 EXECUTION ERROR	1) In continuous circle motion-based groove cutting mode, a command other than G01, G02, G03, G04, G90, G91, and auxiliary functions is specified. 2) In a mode that cannot be used, the continuous circle motion-based groove cutting command is specified.
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) Parameter No.3781 is not specified to the spindle to be selected. 3) An illegal G code which cannot be commanded with an S_P_; command is specified. 4) A multi spindle cannot be used because the 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 prohibited command for a spindle was issued (parameter No. 11090). 7) An invalid value is set in parameter No. 11090.
5312	ILLEGAL COMMAND IN G10 L75/76/77	One of formats in G10L75, G10L76, or G10L77 to G11 commands is in error, or the command value is out of data range. Modify the program.
5316	TOOL TYPE NUMBER NOT FOUND	A tool with the specified tool-type number could not be found. Modify the program or register the tool.
5317	ALL TOOL LIFE IS OVER	The lives of all tools with the specified tool-type number have expired. Replace the tool.
5320	DIA./RAD. MODE CAN'T BE SWITCHED .	In any of the following states, diameter/radius specification was switched: 1) When a buffered program is being executed 2) When a movement is being made on the axis
5324	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed during three-dimensional coordinate conversion, execution of the tilted working plane command, or workpiece setting error compensation.
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.	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 Cs-axis is subjected to synchronous control or superposition control 5. When the emergency stop state is entered during coordinate establishment 6. When an attempt is made to release composite control for the Cs axis being subjected to coordinate establishment 7. When an attempt is made to start synchronous, composite, or superposition control for the Cs axis being subjected to coordinate establishment.

Number	Message	Description
5360	TOOL INTERFERENCE CHECK ERROR	This alarm is issued when interference with another tool is caused by a data modification based on G10 data input or file reading or when an attempt is made to modify the tool figure data of a tool registered in the cartridge.
5361	ILLEGAL MAGAZINE DATA	Tools stored in the cartridge are interfering with each other. Reregister the tools in the cartridge, or modify the tool management data or tool figure data. If this alarm is issued, no tool interference check is made when tools are registered in the cartridge management table. Moreover, empty pot search operation does not operate normally. If this alarm is issued, the power must be turned off before operation is continued.
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.
5364	ILLEGAL COMMAND IN PROGRAM CHECK	(1) An unspecifiable G code was specified in the high speed program check mode. (2) The angular axis control option or customer's board option is enabled. (3) One of the following operations was performed. - Chopping in the high speed program check mode - Starting the high speed program check mode during chopping - High speed cycle machining in the high speed program check mode - Reference position return of an axis for which the reference position is not established, in the high speed program check mode (4) Switching of PMC axis selection signal EAX*<G0136> was performed. (5) G10 was specified for bit 3 (PGR) of parameter No. 3454 in the high speed program check mode. (6) G10 was specified for bit 6 (PGS) of parameter No. 3001 in the high speed program check mode.
5365	NOT CHANGE OF PROGRAM CHECK MODE	(1) Switching of high speed program check input signal PGCK<Gn290.5> was performed during execution of the program.
5372	IMPROPER MODAL G-CODE (G53.2)	In a block in which G53.2 is specified, a G code in group 01 other than G00 and G01 is specified. Or, G53.2 is specified when the modal G code in group 01 is in a state other than the G00 and G01 states.
5373	ARGUMENT CONVERSION ERROR	For outputting a target MDI program for program restart, a macro call argument cannot be converted to a 9-digit number.
5374	FSC MODE MISMATCH IN RESTART	The current flexible synchronous mode differs from the flexible synchronous mode specified in a programmed command in the program restart block.
5375	FSC MODE CAN NOT CHANGED	The flexible synchronous mode was changed during the execution of program restart.
5376	FSC SLAVE AXIS CAN NOT COMMANDED	In the flexible synchronous mode, a command was specified for the slave axis.
5377	INVALID COMMAND AFTER FSC OFF	After the flexible synchronous mode was canceled, an incremental command was specified before an absolute command for the axis specified as the slave axis.
5378	INVALID RESTART BLOCK	The block specified as the restart block after the flexible synchronous mode was canceled was not a block after an absolute command for the axis specified as the slave axis.

Number	Message	Description
5379	WRITE PROTECTED TO SLAVE AXIS	It is not possible to directly set the parameters for the slave axis under axis synchronous control.
5381	INVALID COMMAND IN FSC MODE	An attempt was made to issue the following commands: 1 When the reference position for the master axis under flexible synchronization control has not been established, G28 command for the master axis. 2 G27/G28/G29/G30/G30.1/G53 command for a slave axis.
5384	RETRACT FOR RIGID CANNOT BE CMD.	In retraction for rigid tapping by the G30 command, coordinate mode used when rigid tapping is stopped and that used for retraction for rigid tapping are different.
5391	CAN NOT USE G92	Workpiece coordinate system setting G92 (or G50 for the lathe system G-code system A) 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.
5406	G41.3/G40 FORMAT ERROR	(1) The G41.3 or G40 block contains a move command. (2) The G41.3 block contains a G or M code that suppresses buffering.
5407	ILLEGAL COMMAND IN G41.3	(1) In the G41.3 mode, a G code of group 01 other than G00 and G01 is specified. (2) In the G41.3 mode, an offset command (a G code of group 07) is specified. (3) The block next to G41.3 (startup) specifies no movement.
5408	G41.3 ILLEGAL START_UP	(1) In a mode of group 01 other than G00 and G01, G41.3 (startup) is specified. (2) The included angle between the tool vector and move vector is 0 or 180 degrees at the time of startup.
5420	ILLEGAL PARAMETER IN G43.4/G43.5	A parameter related to tool center point control is illegal. - Acceleration/deceleration before interpolation is disabled. Set parameter No. 1660. - Rapid traverse acceleration/deceleration before interpolation is disabled. Set bit 1 of parameter No. 1401, bit 5 of parameter No. 19501, parameter No. 1671, and parameter No. 1672. - The AI contour control I or AI contour control II option is absent. Set bit 2 (AAI) of parameter No. 11260 to 0.

Number	Message	Description
5421	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I,J,K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02/G03 mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane. - A G code not specifiable during the tool center point control mode was specified. - The modal code used to specify tool center point control is incorrect. - If, in tool center point control mode, any of the following conditions is met, an axis not related to tool center point control (non 5-axis machining control axis) is specified: (1) The option, the expansion of axis move command in tool center point control, is not provided. (2) The number of non 5-axis machining control axes exceeds the maximum number of axes that can be specified. (3) Nano smoothing or NURBS interpolation is performed. - When bit 0 (RCM) of parameter No. 11200 is set to 0 to disable tool direction compensation, tool center point control is specified during the workpiece setting error compensation/tilted working plane command mode. - When tool posture control is enabled under tool center point control (type 2), a command is specified to set a tool posture near a singular point. (This alarm may be suppressed with bit 3 (NPC) of parameter No. 19696.) Check the machine configuration and specification. - When tool posture control is enabled under tool center point control (type 1), a rotary axis angular displacement that disables tool posture control is specified. Check the machine configuration and specification. - During tool center point control (type 2) or tool posture control, nano smoothing or NURBS interpolation is specified. Check the specification. - For nano smoothing in tool center point control (type 1), only linear axes are specified as axes for nano smoothing. Specify rotation axes. - In a state in which the shift of a mirror image remains, tool center point control, tool posture control, or the cutting point command is specified.
5422	EXCESS VELOCITY IN G43.4/G43.5	An attempt was made to make a movement at an axis feedrate exceeding the maximum cutting feedrate by tool center point control.
5424	ILLEGAL TOOL DIRECTION	The rotation axis position for specifying the tool axis direction is not ±90° × n (n = 0, 1, 2,･･･).
5425	ILLEGAL OFFSET VALUE	The offset number is incorrect.