# B.1 LIST OF ALARM CODES

## (1) Program errors /Alarms on program and operation (P/S alarm)

| Number | Message | Contents |
|---|---|---|
| 000 | PLEASE TURN OFF POWER | A parameter which requires the power off was input, turn off power. |
| 001 | TH PARITY ALARM | TH alarm (A character with incorrect parity was input).<br>Correct the tape. |
| 002 | TV PARITY ALARM | TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective. |
| 003 | TOO MANY DIGITS | Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.) |
| 004 | ADDRESS NOT FOUND | A numeral or the sign “ – ” was input without an address at the beginning of a block. Modify the program . |
| 005 | NO DATA AFTER ADDRESS | The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program. |
| 006 | ILLEGAL USE OF NEGATIVE SIGN | Sign “ – ” input error (Sign “ – ” was input after an address with which it cannot be used. Or two or more “ – ” signs were input.)<br>Modify the program. |
| 007 | ILLEGAL USE OF DECIMAL POINT | Decimal point “ . ” input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.)<br>Modify the program. |
| 009 | ILLEGAL ADDRESS INPUT | Unusable character was input in significant area.<br>Modify the program. |
| 010 | IMPROPER G–CODE | An unusable G code or G code corresponding to the function not provided is specified. Modify the program. |
| 011 | NO FEEDRATE COMMANDED | Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program. |
| 014 | ILLEGAL LEAD COMMAND<br>(T series) | In variable lead threading, the lead incremental and decremental outputted by address K exceed the maximum command value or a command such that the lead becomes a negative value is given.<br>Modify the program. |
| | CAN NOT COMMAND G95<br>(M series) | A synchronous feed is specified without the option for threading / synchronous feed. |
| 015 | TOO MANY AXES COMMANDED | An attempt was made to move the machine along the axes, but the number of the axes exceeded the specified number of axes controlled simultaneously. Alternatively, in a block where where the skip function activated by the torque–limit reached signal (G31 P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis. |
| | TOO MANY AXES COMMANDED<br>(T series) | An attempt has been made to move the tool along more than the maximum number of simultaneously controlled axes. Alternatively, no axis movement command or an axis movement command for two or more axes has been specified in the block containing the command for skip using the torque limit signal (G31 P99/98). The command must be accompanied with an axis movement command for a single axis, in the same block. |
| 020 | OVER TOLERANCE OF RADIUS | In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410. |

| Number | Message | Contents |
|---|---|---|
| 021 | ILLEGAL PLANE AXIS COMMANDED | An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program. |
| 022 | NO CIRCLE RADIUS | 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. |
| 023 | ILLEGAL RADIUS COMMAND (T series) | In circular interpolation by radius designation, negative value was commanded for address R. Modify the program. |
| 025 | CANNOT COMMAND F0 IN G02/G03 (M series) | F0 (fast feed) was instructed by F1 –digit column feed in circular interpolation. Modify the program. |
| 027 | NO AXES COMMANDED IN G43/G44 (M series) | No axis is specified in G43 and G44 blocks for the tool length offset type C.<br>Offset is not canceled but another axis is offset for the tool length offset type C. Modify the program. |
| 028 | ILLEGAL PLANE SELECT | In the plane selection command, two or more axes in the same direction are commanded.<br>Modify the program. |
| 029 | ILLEGAL OFFSET VALUE (M series) | The offset values specified by H code is too large.<br>Modify the program. |
| | ILLEGAL OFFSET VALUE (T series) | The offset values specified by T code is too large.<br>Modify the program. |
| 030 | ILLEGAL OFFSET NUMBER (M series) | The offset number specified by D/H code for tool length offset or cutter compensation is too large. Modify the program. |
| | ILLEGAL OFFSET NUMBER (T series) | The offset number in T function specified for tool offset is tool large.<br>Modify the program. |
| 031 | ILLEGAL P COMMAND IN G10 | In setting an offset amount by G10, the offset number following address P was excessive or it was not specified.<br>Modify the program. |
| 032 | 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. |
| 033 | NO SOLUTION AT CRC (M series) | A point of intersection cannot be determined for cutter compensation.<br>Modify the program. |
| | NO SOLUTION AT CRC (T series) | A point of intersection cannot be determined for tool nose radius compensation. Modify the program. |
| 034 | NO CIRC ALLOWED IN ST–UP /EXT BLK (M series) | The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. Modify the program. |
| | NO CIRC ALLOWED IN ST–UP /EXT BLK (T series) | The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program. |
| 035 | CAN NOT COMMANDED G39 (M series) | G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. Modify the program. |
| | CAN NOT COMMANDED G31 (T series) | Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program. |
| 036 | CAN NOT COMMANDED G31 (M series) | Skip cutting (G31) was specified in cutter compensation mode.<br>Modify the program. |
| 037 | CAN NOT CHANGE PLANE IN CRC (M seires) | G40 is commanded on the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program. |
| | CAN NOT CHANGE PLANE IN NRC (T seires) | The offset plane is switched in tool nose radius compensation.<br>Modify the program. |

| Number | Message | Contents |
|---|---|---|
| 038 | INTERFERENCE IN CIRCULAR BLOCK (M seires) | Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program. |
| | INTERFERENCE IN CIRCULAR BLOCK (T series) | Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program. |
| 039 | CHF/CNR NOT ALLOWED IN NRC (T series) | Chamfering or corner R was specified with a start–up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program. |
| 040 | INTERFERENCE IN G90/G94 BLOCK (T series) | Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program. |
| 041 | INTERFERENCE IN CRC (M seires) | Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program. |
| | INTERFERENCE IN NRC (T seires) | Overcutting will occur in tool nose radius compensation. Modify the program. |
| 042 | G45/G48 NOT ALLOWED IN CRC (M series) | Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program. |
| 043 | ILLEGAL T–CODE COMMAND (M series) | In a system using the DRILL–MATE with an ATC, a T code was not specified together with the M06 code in a block. Alternatively, the Tcode was out of range. |
| 044 | G27–G30 NOT ALLOWED IN FIXED CYC (M sries) | One of G27 to G30 is commanded in canned cycle mode. Modify the program. |
| 046 | ILLEGAL REFERENCE RETURN COMMAND | Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command. |
| 047 | ILLEGAL AXIS SELECT (M series) | Two or more parallel axes (in parallel with a basic axis) have been specified upon start–up of three–dimensional tool compensation or three–dimensional coordinate conversion. |
| 048 | BASIC 3 AXIS NOT FOUND (M series) | Start–up of three–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. |
| 050 | CHF/CNR NOT ALLOWED IN THRD BLK (M series) | Optional chamfering or corner R is commanded in the thread cutting block. Modify the program. |
| | CHF/CNR NOT ALLOWED IN THRD BLK(T series) | Chamfering or corner R is commanded in the thread cutting block. Modify the program. |
| 051 | MISSING MOVE AFTER CHF/CNR (M series) | Improper movement or the move distance was specified in the block next to the optional chamfering or corner R block. Modify the program. |
| | MISSING MOVE AFTER CHF/CNR (T series) | Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program. |
| 052 | CODE IS NOT G01 AFTER CHF/CNR (M series) | The block next to the chamfering or corner R block is not G01,G02 or G03. Modify the program. |
| | CODE IS NOT G01 AFTER CHF/CNR (T series) | The block next to the chamfering or corner R block is not G01. Modify the program. |

| Number | Message | Contents |
|---|---|---|
| 053 | TOO MANY ADDRESS COMMANDS (M series) | For systems without the arbitary angle chamfering or corner R cutting, a comma was specified. For systems with this feature, a comma was followed by something other than R or C  Correct the program. |
| | TOO MANY ADDRESS COMMANDS (T seires) | In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(”,”) is not C or R in direct drawing dimensions programming. Modify the program. |
| 054 | NO TAPER ALLOWED AFTER CHF/ CNR (T series) | A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program. |
| 055 | MISSING MOVE VALUE IN CHF/CNR (M series) | In the arbitrary angle chamfering or corner R block, the move distance is less than chamfer or corner R amount. |
| | MISSING MOVE VALUE IN CHF/CNR (T series) | In chamfering or corner R block, the move distance is less than chamfer or corner R amount. |
| 056 | NO END POINT & ANGLE IN CHF/ CNR (T series) | Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering comman, I(K) is commanded for the X(Z) axis. |
| 057 | NO SOLUTION OF BLOCK END (T series) | Block end point is not calculated correctly in direct dimension drawing programming. |
| 058 | END POINT NOT FOUND (M series) | In a arbitrary angle chamfering or corner R cutting block, a specified axis is not in the selected plane. Correct the program. |
| | END POINT NOT FOUND (T series) | Block end point is not found in direct dimension drawing programming. |
| 059 | PROGRAM NUMBER NOT FOUND | In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background eiting. |
| 060 | SEQUENCE NUMBER NOT FOUND | Commanded sequence number was not found in the sequence number search. Check the sequence number. |
| 061 | ADDRESS P/Q NOT FOUND IN G70–G73 (T series) | Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program. |
| 062 | ILLEGAL COMMAND IN G71–G76 (T series) | 1. The depth of cut in G71 or G72 is zero or negative value.<br>2. The repetitive count in G73 is zero or negative value.<br>3. the negative value is specified to $\Delta$i or $\Delta$k is zero in G74 or G75.<br>4. A value other than zero is specified to address U or W though $\Delta$i or $\Delta$k is zero in G74 or G75.<br>5. A negative value is specified to $\Delta$d, thoughthe relief direction in G74 or G75 is determined.<br>6. Zero or a negative value is specified to the height of thread or depth of cut of first time in G76.<br>7. The specified minimum depth of cut in G76 is greater than the height of thread.<br>8. An unusable angle of tool tip is specified in G76.<br>Modify the program. |
| 063 | SEQUENCE NUMBER NOT FOUND (T series) | The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program. |
| 064 | SHAPE PROGRAM NOT MONOTONOUSLY (T series) | A target shape which cannot be made by monotonic machining was specified in a repetitive canned cycle (G71 or G72). |
| 065 | ILLEGAL COMMAND IN G71–G73 (T series) | 1. G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command.<br>2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively.<br>Modify the program. |

| Number | Message | Contents |
|---|---|---|
| 066 | IMPROPER G–CODE IN G71–G73 (T series) | An unallowable G code was commanded beween two blocks specified by address P in G71, G72, or G73. Modify the program. |
| 067 | CAN NOT ERROR IN MDI MODE (T series) | G70, G71, G72, or G73 command with address P and Q. Modify the program. |
| 069 | FORMAT ERROR IN G70–G73 (T series) | The final move command in the blocks specified by P and Q of G70, G71, G72, and G73 ended with chamfering or corner R. Modify the program. |
| 070 | NO PROGRAM SPACE IN MEMORY | The memory area is insufficient. Delete any unnecessary programs, then retry. |
| 071 | DATA NOT FOUND | The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data. |
| 072 | 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 registeration again. |
| 073 | PROGRAM NUMBER ALREADY IN USE | The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registeration again. |
| 074 | ILLEGAL PROGRAM NUMBER | The program number is other than 1 to 9999. Modify the program number. |
| 075 | PROTECT | An attempt was made to register a program whose number was protected. |
| 076 | ADDRESS P NOT DEFINED | Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program. |
| 077 | SUB PROGRAM NESTING ERROR | The subprogram was called in five folds. Modify the program. |
| 078 | NUMBER NOT FOUND | A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing. |
| 079 | PROGRAM VERIFY ERROR | In memory or program collation,a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device. |
| 080 | G37 ARRIVAL SIGNAL NOT ASSERTED (M series) | In the automatic tool length measurement function (G37), the measurement position reach signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter 6254 6255 (value ε). This is due to a setting or operator error. |
| | G37 ARRIVAL SIGNAL NOT ASSERTED (T series) | In the automatic tool compensation function (G36, G37), the measurement position reach signal (XAE or ZAE) is not turned on within an area specified in parameter 6254 (value ε). This is due to a setting or operator error. |
| 081 | OFFSET NUMBER NOT FOUND IN G37 (M series) | Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program. |
| | OFFSET NUMBER NOT FOUND IN G37 (T series) | Automatic tool compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program. |
| 082 | H–CODE NOT ALLOWED IN G37 (M series) | H code and automatic tool compensation (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program. |
| | T–CODE NOT ALLOWED IN G37 (T series) | T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program. |

| Number | Message | Contents |
|---|---|---|
| 083 | ILLEGAL AXIS COMMAND IN G37 (M series) | In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program. |
| | ILLEGAL AXIS COMMAND IN G37 (T series) | In automatic tool compensation (G36, G37), an invalid axis was specified or the command is incremental. Modify the program. |
| 085 | COMMUNICATION ERROR | When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect. |
| 086 | DR SIGNAL OFF | When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off.<br>Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective. |
| 087 | BUFFER OVERFLOW | When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective. |
| 088 | LAN FILE TRANS ERROR (CHANNEL–1) | File data transfer via OSI–ETHERNET has been stopped due to a transfer error. |
| 089 | LAN FILE TRANS ERROR (CHANNEL–2) | File data transfer via OSI–ETHERNET has been stopped due to a transfer error. |
| 090 | REFERENCE RETURN INCOMPLETE | 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. Check the program contents. |
| 091 | REFERENCE RETURN INCOMPLETE | Manual reference position return cannot be performed when automatic operation is halted. |
| 092 | AXES NOT ON THE REFERENCE POINT | The commanded axis by G27 (Reference position return check) did not return to the reference position. |
| 094 | 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.)<br>Perform the correct operation according to th operator's manual. |
| 095 | 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.)<br>Perform the correct operation according to th operator's manual. |
| 096 | 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.)<br>Perform the correct operation according to the operator's manual. |
| 097 | P TYPE NOT ALLOWED (AUTO EXEC) | P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.) Perform automatic operation. |
| 098 | 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.<br>Perform the reference position return. |
| 099 | MDI EXEC NOT ALLOWED AFT. SEARCH | After completion of search in program restart, a move command is given with MDI. Move axis before a move command or don't interrupt MDI operation. |
| 100 | PARAMETER WRITE ENABLE | On the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system. |

| Number | Message | Contents |
|---|---|---|
| 101 | PLEASE CLEAR MEMORY | The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted.<br>Register the deleted program. |
| 109 | P/S ALARM | A value other than 0 or 1 was specified after P in the G08 code, or no value was specified. |
| 110 | DATA OVERFLOW | The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program. |
| 111 | CALCULATED DATA OVERFLOW | The result of calculation turns out to be invalid, an alarm No.111 is issued.<br>$-10^{47}$ to $-10^{-29}$, 0, $10^{-29}$ to $10^{47}$<br>Modify the program. |
| 112 | DIVIDED BY ZERO | Division by zero was specified. (including tan 90°)<br>Modify the program. |
| 113 | IMPROPER COMMAND | A function which cannot be used in custom macro is commanded.<br>Modify the program. |
| 114 | FORMAT ERROR IN MACRO | There is an error in other formats than <Formula>.<br>Modify the program. |
| 115 | ILLEGAL VARIABLE NUMBER | A value not defined as a variable number is designated in the custom macro or in high–speed cycle machining.<br>The header contents are improper. This alarm is given in the following cases:<br>High speed cycle machining<br>1. The header corresponding to the specified machining cycle number called is not found.<br>2. The cycle connection data value is out of the allowable range (0 – 999).<br>3. The number of data in the header is out of the allowable range (0 – 32767).<br>4. The start data variable number of executable format data is out of the allowable range (#20000 – #85535).<br>5. The last storing data variable number of executable format data is out of the allowable range (#85535).<br>6. The storing start data variable number of executable format data is overlapped with the variable number used in the header.<br>Modify the program. |
| 116 | WRITE PROTECTED VARIABLE | The left side of substitution statement is a variable whose substitution is inhibited. Modify the program. |
| 118 | PARENTHESIS NESTING ERROR | The nesting of bracket exceeds the upper limit (quintuple).<br>Modify the program. |
| 119 | ILLEGAL ARGUMENT | The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument.<br>Modify the program. |
| 122 | FOUR FOLD MACRO MODAL–CALL | The macro modal call is specified four fold.<br>Modify the program. |
| 123 | CAN NOT USE MACRO COMMAND IN DNC | Macro control command is used during DNC operation.<br>Modify the program. |
| 124 | MISSING END STATEMENT | DO – END does not correspond to 1 : 1. Modify the program. |
| 125 | FORMAT ERROR IN MACRO | <Formula> format is erroneous. Modify the program. |
| 126 | ILLEGAL LOOP NUMBER | In DOn, $1 \leqq n \leqq 3$ is not established. Modify the program. |

| Number | Message | Contents |
|---|---|---|
| 127 | NC, MACRO STATEMENT IN SAME BLOCK | NC and custom macro commands coexist.<br>Modify the program. |
| 128 | ILLEGAL MACRO SEQUENCE NUMBER | The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program. |
| 129 | ILLEGAL ARGUMENT ADDRESS | An address which is not allowed in <Argument Designation > is used. Modify the program. |
| 130 | ILLEGAL AXIS OPERATION | An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program. |
| 131 | TOO MANY EXTERNAL ALARM MESSAGES | Five or more alarms have generated in external alarm message.<br>Consult the PMC ladder diagram to find the cause. |
| 132 | ALARM NUMBER NOT FOUND | No alarm No. concerned exists in external alarm message clear.<br>Check the PMC ladder diagram. |
| 133 | ILLEGAL DATA IN EXT. ALARM MSG | Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram. |
| 135 | ILLEGAL ANGLE COMMAND<br>(M series) | The index table indexing positioning angle was instructed in other than an integral multiple of the value of the minimum angle.<br>Modify the program. |
| | SPINDLE ORIENTATION PLEASE<br>(T series) | Without any spindle orientation , an attept was made for spindle indexing. Perform spindle orientation. |
| 136 | ILLEGAL AXIS COMMAND<br>(M series) | In index table indexing.Another control axis was instructed together with the B axis.<br>Modify the program. |
| | C/H–CODE & MOVE CMD IN SAME BLK. (T series) | A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program. |
| 137 | M–CODE & MOVE CMD IN SAME BLK. | A move command of other axes was specified to the same block as M–code related to spindle indexing. Modify the program. |
| 138 | SUPERIMPOSED DATA OVERFLOW | The total distribution amount of the CNC and PMC is too large during superimposed control of the extended functions for PMC axis control. |
| 139 | CAN NOT CHANGE PMC CONTROL AXIS | An axis is selected in commanding by PMC axis control.<br>Modify the program. |
| 141 | CAN NOT COMMAND G51 IN CRC<br>(M series) | G51 (Scaling ON) is commanded in the tool offset mode.<br>Modify the program. |
| 142 | ILLEGAL SCALE RATE<br>(M series) | Scaling magnification is commanded in other than 1 – 999999.<br>Correct the scaling magnification setting (G51 $P_p$ .....................<br>or parameter 5411 or 5421). |
| 143 | SCALED MOTION DATA OVERFLOW<br>(M series) | The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling mangification. |
| 144 | ILLEGAL PLANE SELECTED<br>(M series) | The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program. |
| 145 | ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION | The conditions are incorrect when the polar coordinate interpolation starts or it is canceled.<br>1) In modes other than G40, G12.1/G13.1 was specified.<br>2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified.<br>Modify the value of program or parameter. |
| 146 | IMPROPER G CODE | G codes which cannot be specified in the polar coordinate interpolation mode was specified. See section II–4.4 and modify the program. |
| 148 | ILLEGAL SETTING DATA<br>(M series) | Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714) |

| Number | Message | Contents |
|---|---|---|
| 149 | FORMAT ERROR IN G10L3 (M series) | A code other than Q1,Q2,P1 or P2 was specified as the life count type in the extended tool life management. |
| 150 | ILLEGAL TOOL GROUP NUMBER | Tool Group No. exceeds the maximum allowable value. Modify the program. |
| 151 | TOOL GROUP NUMBER NOT FOUND | The tool group commanded in the machining program is not set. Modify the value of program or parameter. |
| 152 | NO SPACE FOR TOOL ENTRY | The number of tools within one group exceeds the maximum value registerable. Modify the number of tools. |
| 153 | T–CODE NOT FOUND | In tool life data registration, a T code was not specified where one should be. Correct the program. |
| 154 | NOT USING TOOL IN LIFE GROUP (M series) | When the group is not commanded, H99 or D99 was commanded. Correct the program. |
| 155 | ILLEGAL T–CODE IN M06 (M series) | In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program. |
| | ILLEGAL T–CODE IN M06 (T series) | Group No.ΔΔ which is specified with TΔΔ 88 of the machining program do not included in the tool group in use. Correct the program. |
| 156 | P/L COMMAND NOT FOUND | P and L commands are missing at the head of program in which the tool group is set. Correct the program. |
| 157 | TOO MANY TOOL GROUPS | The number of tool groups to be set exceeds the maximum allowable value. (See parameter No. 6800 bit 0 and 1) Modify the program. |
| 158 | ILLEGAL TOOL LIFE DATA | The tool life to be set is too excessive. Modify the setting value. |
| 159 | TOOL DATA SETTING INCOMPLETE | During executing a life data setting program, power was turned off. Set again. |
| 160 | MISMATCH WATING M–CODE T series (At two–path) | Diffrent M code is commanded in heads 1 and 2 as waiting M code. Modify the program. |
| | G72.1 NESTING ERROR (M series) | A subprogram which performs rotational copy with G72.1 contains another G72.1 command. |
| 161 | G72.1 NESTING ERROR (M series) | A subprogram which performs parallel copy with G72.2 contains another G72.2 command. |
| 163 | COMMAND G68/G69 INDEPENDENTLY T series (At two–path) | G68 and G69 are not independently commanded in balance cut. Modify the program. |
| 169 | ILLEGAL TOOL GEOMETRY DATA T series (At two–path) | Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data. |
| 175 | ILLEGAL G107 COMMAND | Conditions when performing circular interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of "G07.1 rotation–axis name radius of cylinder." |
| 176 | IMPROPER G–CODE IN G107 (M series) | Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified.<br>1) G codes for positioning: G28,, G73, G74, G76, G81 – G89, including the codes specifying the rapid traverse cycle<br>2) G codes for setting a coordinate system: G52,G92,<br>3) G code for selecting coordinate system: G53 G54–G59<br>Modify the program. |
| | IMPROPER G–CODE IN G107 (T series) | Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified.<br>1) G codes for positioning: G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle<br>2) G codes for setting a coordinate system: G50, G52<br>3) G code for selecting coordinate system: G53 G54–G59<br>Modify the program. |

| Number | Message | Contents |
|---|---|---|
| 177 | CHECK SUM ERROR (G05 MODE) | Check sum error<br>Modify the program. |
| 178 | G05 COMMANDED IN G41/G42 MODE | G05 was commanded in the G41/G42 mode.<br>Correct the program. |
| 179 | PARAM. (NO. 7510) SETTING ERROR | The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value. |
| 180 | COMMUNICATION ERROR (REMOTE BUF) | Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device. |
| 181 | FORMAT ERROR IN G81 BLOCK (M series) | G81 block format error (hobbing machine)<br>1) T (number of teeth) has not been instructed.<br>2) Data outside the command range was instructed by either T, L, Q or P.<br>Modify the program. |
| 182 | G81 NOT COMMANDED (M series) | G83 (C axis servo lag quantity offset) was instructed though synchronization by G81 has not been instructed. Correct the program. (hobbing machine) |
| 183 | DUPLICATE G83 (COMMANDS) (M series) | G83 was instructed before canceled by G82 after compensating for the C axis servo lag quantity by G83. (hobbing machine) |
| 184 | ILLEGAL COMMAND IN G81 (M series) | A command not to be instructed during synchronization by G81 was instructed. (hobbing machine)<br>1) A C axis command by G00, G27, G28, G29, G30, etc. was instructed.<br>2) Inch/Metric switching by G20, G21 was instructed. |
| 185 | RETURN TO REFERENCE POINT (M series) | G81 was instructed without performing reference position return after power on or emergency stop. (hobbing machine) Perform reference position return. |
| 186 | PARAMETER SETTING ERROR (M series) | Parameter error regarding G81 (hobbing machine)<br>1) The C axis has not been set to be a rotary axis.<br>2) A hob axis and position coder gear ratio setting error<br>Modify the parameter. |
| 190 | ILLEGAL AXIS SELECT (M series) | In the constant surface speed control, the axis specification is wrong. (See parameter No. 3770.) The specified axis command (P) contains an illegal value.<br>Correct the program. |
| 194 | SPINDLE COMMAND IN SYNCHRO–MODE | A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance. |
| 195 | MODE CHANGE ERROR | Switching command to contouring mode, Cs axis control or rigid tap mode or switching to spindle command mode is not correctly completed.<br>(This occurs when the response to switch to the spindle control unit side with regard to the switching command from the NC is incorrect.<br>This alarm is not for the purposes of warning against mistakes in operation, but because continuing operation in this condition can be dangerous it is a P/S alarm.) |
| 197 | C–AXIS COMMANDED IN SPINDLE MODE | The program specified a movement along the Cs–axis when the signal CON(DGN=G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on. |
| 199 | MACRO WORD UNDEFINED | Undefined macro word was used. Modify the custom macro. |

| Number | Message | Contents |
|---|---|---|
| 200 | ILLEGAL S CODE COMMAND | In the rigid tap, an S value is out of the range or is not specified. Modify the program. |
| 201 | FEEDRATE NOT FOUND IN RIGID TAP | In the rigid tap, no F value is specified.<br>Correct the program. |
| 202 | POSITION LSI OVERFLOW | In the rigid tap, spindle distribution value is too large. (System error) |
| 203 | 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. |
| 204 | ILLEGAL AXIS OPERATION | In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 or G74 for M series (G84 or G88 for T series) block. Modify the program. |
| 205 | RIGID MODE DI SIGNAL OFF | Rigid mode DI signal is not ON when G84 or G74 for M series (G84 or G88 for T series) is executed though the rigid M code (M29) is specified.Consult the PMC ladder diagram to find the reason the DI signal (DGNG061.1) is not turned on. |
| 206 | CAN NOT CHANGE PLANE (RIGID TAP) (M series) | Plane changeover was instructed in the rigid mode.<br>Correct the program. |
| 210 | CAN NOT COMAND M198/M199 | M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.<br>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. Correct the program.<br>2) The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle. |
| 211 | G31 (HIGH) NOT ALLOWED IN G99 | G31 is commanded in the per revolution command when the high–speed skip option is provided. Modify the program. |
| 212 | ILLEGAL PLANE SELECT (M series) | The arbitrary angle chamfering or a corner R is commanded or the plane including an additional axis. Correct the program. |
| | ILLEGAL PLANE SELECT (T series) | The direct drawing dimensions programming is commanded for the plane other than the Z–X plane. Correct the program. |
| 213 | ILLEGAL COMMAND IN SYNCHRO–MODE | Movement is commanded for the axis to be synchronously controlled. Any of the following alarms occurred in the operation with the simple synchronization control.<br>1) The program issued the move command to the slave axis.<br>2) The program issued the manual continuous feed/manual handle feed/incremental feed command to the slave axis.<br>3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.<br>4) The difference between the position error amount of the master and slave axes exceeded the value specified in parameter NO.8313. |
| | ILLEGAL COMMAND IN SYNCHRO–MODE (T series) | A move command has been specified for an axis subject to synchronous control. |
| 214 | 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. |
| 217 | DUPLICATE G51.2 (COMMANDS) (T series) | G51.2/G251 is further commanded in the G51.2/G251 mode. Modify the program. |
| 218 | NOT FOUND P/Q COMMAND IN G251 (T series) | P or Q is not commanded in the G251 block, or the command value is out of the range. Modify the program. |
| 219 | COMMAND G250/G251 INDEPENDENTLY (T series) | G251 and G250 are not independent blocks. |

| Number | Message | Contents |
|---|---|---|
| 220 | ILLEGAL COMMAND IN SYNCHR–MODE (T series) | In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. |
| 221 | ILLEGAL COMMAND IN SYNCHR–MODE (T series) | Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program. |
| 222 | DNC OP. NOT ALLOWED IN BG.–EDIT (M series) | Input and output are executed at a time in the background edition. Execute a correct operation. |
| 224 | RETURN TO REFERENCE POINT (M series) | Reference position return has not been performed before the automatic operation starts. Perform reference position return only when bit 0 of parameter 1005 is 0. |
| | TURN TO REFERENCE POINT (T series) | Reference position return is necessary before cycle start. |
| 225 | SYNCHRONOUS/MIXED CONTROL ERROR<br>T series (At two–path) | This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command.<br>1 When there is a mistake in axis number parameter setting.<br>2 When there is a mistake in control commanded.<br>Modify the program or the parameter. |
| 226 | ILLEGAL COMMAND IN SYNCHRO–MODE T series (At two–path) | A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the parameter. |
| 229 | CAN NOT KEEP SYNCHRO–STATE (T series) | This alarm is generated in the following circumstances.<br>1 When the synchro/mixed state could not be kept due to system overload.<br>2 The above condition occurred in CMC devices (hardware) and synchro–state could not be kept.<br>(This alarm is not generated in normal use conditions.) |
| 230 | R CODE NOT FOUND (GS series) | The infeed quantity R has not been instructed for the G161 block. Or the R command value is negative. Correct the program. |
| 231 | ILLEGAL FORMAT IN G10 OR L50 | Any of the following errors occurred in the specified format at the programmable–parameter input.<br>1 Address N or R was not entered.<br>2 A number not specified for a parameter was entered.<br>3 The axis number was too large.<br>4 An axis number was not specified in the axis–type parameter.<br>5 An axis number was specified in the parameter which is not an axis type. Correct the program.<br>6 An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program. |
| 232 | TOO MANY HELICAL AXIS COMMANDS (M series) | Three or more axes (in the normal direction control mode two or more axes) were specified as helical axes in the helical interpolation mode. |
| 233 | 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. |
| 239 | BP/S ALARM | While punching was being performed with the function for controlling external I/O units ,background editing was performed. |
| 240 | BP/S ALARM | Background editing was performed during MDI operation. |
| 241 | ILLEGAL FORMAT IN G02.2/G03.2 (M series) | The end point, I, J, K, or R is missing from a command for involute interpolation. |

| Number | Message | Contents |
|---|---|---|
| 242 | ILLEGAL COMMAND IN G02.2/G03.2 (M series) | An invalid value has been specified for involute interpolation.<br>• The start or end point is within the basic circle.<br>• I, J, K, or R is set to 0.<br>• The number of rotations between the start of the involute curve and the start or end point exceeds 100. |
| 243 | OVER TOLERANCE OF END POINT (M series) | The end point is not on the involute curve which includes the start point and thus falls outside the range specified with parameter No. 5610. |
| 244 | P/S ALARM (T series) | In the skip function activated by the torque limit signal, the number of accumulated erroneous pulses exceed 32767 before the signal was input. Therefore, the pulses cannot be corrected with one distribution. Change the conditions, such as feed rates along axes and torque limit, and try again. |
| 245 | T–CODE NOT ALOWEE IN THIS BLOCK (T series) | One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code. |
| 250 | Z AXIS WRONG COMMAND (ATC) (M series) | A value for the Z–axis has been specified in a block for the tool exchange command (M06T_) on a system with DRILL–MATE ARC installed. |
| 251 | ATC ERROR (M series) | This alarm is issued in the following cases (DRILL–MATE):<br>• An M06T_ command contains an unusable T code.<br>• An M06 command has been specified when the Z machine coordi nate is positive.<br>• The parameter for the current tool number (No. 7810) is set to 0.<br>• An M06 command has been specified in canned cycle mode.<br>• A reference position return command (G27 to G44) and M06 command have been specified in the same block.<br>• An M06 command has been specified in tool compensation mode (G41 to G44).<br>• An M06 command has been specified without performing reference position return after power–on or the release of emergency stop.<br>• The machine lock signal or Z–axis ignore signal has been turned on during tool exchange.<br>• A pry alarm has been detected during tool exchange.<br>Refer to diagnosis No. 530 to determine the cause. |
| 252 | ATC SPINDLE ALARM (M series) | An excessive error arose during spindle positioning for ATC. For details, refer to diagnosis No. 531. (Only for DRILL–MATE) |
| 253 | G05 IS NOT AVAILABLE (M series) | Alarm details<br>Binary input operation using high–speed remote buffer (G05) or high–speed cycle machining (G05) has been specified in advance control mode (G08P1). Execute G08P0; to cancel advance control mode, before executing these G05 commands. |
| 4500 | REPOSITIONING INHIBITED | A repositioning command was specified in the circular interpolation (G02, G03) mode. |
| 4502 | ILLEGAL COMMAND IN BOLT HOLE | In a bolt hole circle (G26) command, the radius (I) was set to zero or a negative value, or the number of holes (K) was set to zero. Alternatively, I, J, or K was not specified. |
| 4503 | ILLEGAL COMMAND IN LINE AT ANGLE | In a line-at-angle (G76) command, the number of holes (K) was set to zero or a negative value. Alternatively, I, J, or K was not specified. |
| 4504 | ILLEGAL COMMAND IN ARC | In an arc (G77) command, the radius (I) or the number of holes (K) was set to zero or a negative value. Alternatively, I, J, K, or P was not specified. |

| Number | Message | Contents |
|---|---|---|
| 4505 | ILLEGAL COMMAND IN GRID | In a grid (G78, G79) command, the number of holes (P, K) was set to zero or a negative value. Alternatively, I, J, K, or P was not specified. |
| 4506 | ILLEGAL COMMAND IN SHARE PROOFS | In a shear proof (G86) command, the tool size (P) was set to zero, or the blanking length (I) was 1.5 times larger than the tool size (P) or less. Alternatively, I, J, or P was not specified. |
| 4507 | ILLEGAL COMMAND IN SQUARE | In a square (G87) command, the tool size (P,Q) was set to zero or a negative value, or the blanking length (I, J) was three times larger than the tool size (P, Q) or less. Alternatively, I, J, P, or Q was not specified. |
| 4508 | ILLEGAL COMMAND IN RADIUS | In a radius (G88) command, the traveling pitch (Q) or radius (I) was set to zero or a negative value, or the traveling pitch (Q) was greater than or equal to the arc length. Alternatively, I, J, K, P, or Q was not specified. |
| 4509 | ILLEGAL COMMAND IN CUT AT ANGLE | In a cut-at-angle (G89) command, the traveling pitch (Q) was set to zero, negative value, or another value larger than or equal to the length (I). Alternatively, I, J, P, or Q was not specified. |
| 4510 | ILLEGAL COMMAND IN LINE-PUNCH | In a linear punching (G45) command, the traveling distance was set to zero or a value 1.5 times larger than the tool size (P) or less. Alternatively, P was not specified. |
| 4511 | ILLEGAL COMMAND IN CIRCLE-PUNCH | In a circular punching (G46, G47) command, the same position was specified for both start and end points of the arc, radius (R) of the arc was set to zero, or the pitch (Q) was set to a value exceeding the arc length. Alternatively, R or Q was not specified. |
| 4520 | T, M INHIBITED IN NIBBLING-MODE | T code, M code, G04, G70 or G75 was specified in the nibbling mode. |
| 4521 | EXCESS NIBBLING MOVEMENT (X, Y) | In the nibbling mode, the X-axis or Y-axis traveling distance was larger than or equal to the limit (No. 16188 to 16193). |
| 4522 | EXCESS NIBBLING MOVEMENT (C) | In the circular nibbling (G68) or usual nibbling mode, the C-axis traveling distance was larger than or equal to the limit (No. 16194). |
| 4523 | ILLEGAL COMMAND IN CIRCLE-NIBBL | In a circular nibbling (G68) command, the traveling pitch (Q) was set to zero, a negative value, or a value larger than or equal to the limit (No. 16186, 16187), or the radius (I) was set to zero or a negative value. Alternatively, I, J, K, P, or Q was not specified. |
| 4524 | ILLEGAL COMMAND IN LINE-NIBBL | In a linear nibbling (G69) command, the traveling pitch (Q) was set to zero, negative value, or a value larger than or equal to the limit (No. 16186, 16187). Alternatively, I, J, P, or Q was not specified. |
| 4530 | A/B MACRO NUMBER ERROR | The number for storing and calling by an A or B macro was set to a value beyond the range from 1 to 5. |
| 4531 | U/V MACRO FORMAT ERROR | An attempt was made to store a macro while storing another macro using a U or V macro.<br>A V macro was specified although the processing to store a macro was not in progress.<br>A U macro number and V macro number do not correspond with each other. |
| 4532 | IMPROPER U/V MACRO NUMBER | The number of an inhibited macro (number beyond the range from 01 to 99) was specified in a U or V macro command. |
| 4533 | U/V MACRO MEMORY OVERFLOW | An attempt was made to store too many macros with a U or V macro command. |
| 4534 | W MACRO NUMBER NOT FOUND | Macro number W specified in a U or V macro command is not stored. |
| 4535 | U/V MACRO NESTING ERROR | An attempt was made to call a macro which is defined three times or more using a U or V macro command.<br>An attempt was made to store 15 or more macros in the storage area for macros of number 90 to 99. |

| Number | Message | Contents |
|---|---|---|
| 4536 | NO W, Q COMMAND IN MULTI-PIECE | W or Q was not specified in the command for taking multiple workpieces (G73, G74). |
| 4537 | ILLEGAL Q VALUE IN MULTI-PIECE | In the command for taking multiple workpieces (G73, G74), Q is set to a value beyond the range from 1 to 4. |
| 4538 | W NO. NOT FOUND IN MULTI-PIECE | Macro number W specified in the command for taking multiple workpieces (G73, G74) is not stored. |
| 4539 | MULTI-PIECE SETTING IS ZERO | The command for taking multiple workpieces (G73, G74) was specified although zero is specified for the function to take multiple workpieces (No. 16206 or signals MLP1 and MLP2 (PMC address G231, #0 and #1)). |
| 4540 | MULTI-PIECE COMMAND WITHIN MACRO | The command for taking multiple workpieces (G73, G74) was specified when a U or V macro was being stored. |
| 4542 | MULTI-PIECE COMMAND ERROR | Although G98P0 was specified, the G73 command was issued.<br>Although G98K0 was specified, the G74 command was issued. |
| 4543 | MULTI-PIECE Q COMMAND ERROR | Although G98P0 was specified, the Q value for the G74 command was not 1 or 3.<br>Although G98K0 was specified, the Q value for the G73 command was not 1 or 2. |
| 4544 | MULTI-PIECE RESTART ERROR | In the command for resuming taking multiple workpieces, the resume position (P) is set to a value beyond the range from 1 to total number of workpieces to be machined. |
| 4549 | ILLEGAL TOOL DATA FORMAT | The quantity of tool data patterns to be saved is too large to fit the usable area (16 KB). |
| 4600 | T, C COMMAND IN INTERPOLATION | In the linear interpolation (G01) mode or circular interpolation (G02, G03) mode, a T command or C-axis command was specified. |
| 4601 | INHIBITED T, M COMMAND | In the block of G52, G72, G73, or G74, a T or M command was specified. |
| 4602 | ILLEGAL T-CODE | The specified T command is not cataloged on the tool register screen. |
| 4603 | C AXIS SYNCHRONOUS ERROR | The difference between the position deviation value of C1 axis and C2 axis exceeds the parameter value (No. 16364, 16365) with the C–axis synchronous control function. |
| 4604 | ILLEGAL AXIS OPERATION | A C-axis command was specified in the block containing a T command for multiple tools. |
| 4630 | ILLEGAL COMMAND IN LASER MODE | In the laser mode, a nibbling command or pattern command was specified.<br>In the tracing mode, an attempt was made to make a switch to the punching mode. |
| 4631 | ILLEGAL COMMAND IN PUNCH MODE | In the punching mode, a G code of laser control (G13, G24, G31, etc.) was specified. |
| 4650 | IMPROPER G-CODE IN OFFSET MODE | In the cutter compensation mode, an inhibited G code (pattern command, G73, G74, G75, etc.) was specified. |
| 4700 | PROGRAM ERROR (OT +) | The value specified in the X-axis move command exceeded the positive value of stored stroke limit 1. (Advance check) |
| 4701 | PROGRAM ERROR (OT –) | The value specified in the X-axis move command exceeded the negative value of stored stroke limit 1. (Advance check) |
| 4702 | PROGRAM ERROR (OT +) | The value specified in the Y-axis move command exceeded the positive value of stored stroke limit 1. (Advance check) |
| 4703 | PROGRAM ERROR (OT –) | The value specified in the Y-axis move command exceeded the negative value of stored stroke limit 1. (Advance check) |
| 4704 | PROGRAM ERROR (OT +) | The value specified in the Z-axis move command exceeded the positive value of stored stroke limit 1. (Advance check) |

| Number | Message | Contents |
|---|---|---|
| 4705 | PROGRAM ERROR (OT –) | The value specified in the Z-axis move command exceeded the negative value of stored stroke limit 1. (Advance check) |
| 5000 | ILLEGAL COMMAND CODE (M series) | The specified code was incorrect in the high–precision contour control (HPCC) mode. |
| 5003 | ILLEGAL PARAMETER (HPCC) (M series) | There is an invalid parameter. |
| 5004 | HPCC NOT READY (M series) | High–precision contour control is not ready. |
| 5006 | TOO MANY WORD IN ONE BLOCK (M series) | The number of words specified in a block exceeded 26 in the HPCC mode. |
| 5007 | TOO LARGE DISTANCE (M series) | In the HPCC mode, the machine moved beyond the limit. |
| 5009 | PARAMETER ZERO (DRY RUN) (M series) | The maximum feedrate (parameter No. 1422) or the feedrate in dry run (parameter No. 1410) is 0 in the HPCC model. |
| 5010 | END OF RECORD | The end of record (%) was specified.<br>I/O is incorrect. modify the program. |
| 5011 | PARAMETER ZERO(CUT MAX) (M series) | The maximum cutting feedrate (parameter No. 1422)is 0 in the HPCC mode. |
| 5012 | G05 P10000 ILLEGAL START UP (HPCC)<br>(M series) | Function category:<br>High–precision contour control<br>Alarm details:<br>G05 P10000 has been specified in a mode from which the system cannot enter HPCC mode. |
| 5013 | HPCC: CRC OFS REMAIN AT CANCEL (M series) | G05P0 has been specified in G41/G42 mode or with offset remaining. |
| 5014 | TRACE DATA NOT FOUND (M series) | Transfer cannot be performed because no trace data exists. |
| 5015 | (M series) | The specified rotation axis does not exist for tool axis direction handle feed. |
| 5016 | ILLEGAL COMBINATION OF M CODE | 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<br>(T series) | Function category:<br>Polygon turning<br>Alarm details:<br>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. |
| 5020 | PARAMETER OF RESTART ERROR | An erroneous parameter was specified for restarting a program.<br>A parameter for program restart is invalid. |
| 5030 | ILLEGAL COMMAND (G100) (T series) | The end command (G110) was specified before the registratioin start command (G101, G102, or G103) was specified for the B–axis. |
| 5031 | ILLEGAL COMMAND (G100, G102, G103) (T series) | While a registration start command (G101, G102, or G103) was being executed, another registration start command was specified for the B–axis. |
| 5032 | NEW PRG REGISTERED IN B–AXS MOVE (T series) | While the machine was moving about the B–axis, at attempt was made to register another move command. |
| 5033 | NO PROG SPACE IN MEMORY B–AXS (T series) | Commands for movement about the B–axis were not registered because of insufficient program memory. |
| 5034 | PLURAL COMMAND IN G110 (T series) | Multiple movements were specified with the G110 code for the B–axis. |
| 5035 | NO FEEDRATE COMMANDED B–AXS (T series) | A feedrate was not specified for cutting feed about the B–axis. |

| Number | Message | Contents |
|---|---|---|
| 5036 | ADDRESS R NOT DEFINED IN G81–G86 (T series) | Point R was not specified for the canned cycle for the B–axis. |
| 5037 | ADDRESS Q NOT DEFINED IN G83 (T series) | Depth of cut Q was not specified for the G83 code (peck drilling cycle). Alternatively, 0 was specified in Q for the B–axis. |
| 5038 | TOO MANY START M–CODE COMMAND (T series) | More than six M codes for starting movement about the B–axis were specified. |
| 5039 | START UNREGISTERED B–AXS PROG (T series) | An attempt was made to execute a program for the B–axis which had not been registered. |
| 5040 | CAN NOT COMMANDED B–AXS MOVE (T series) | The machine could not move about the B–axis because parameter No.8250 was incorrectly specified, or because the PMC axis system could not be used. |
| 5041 | CAN NOT COMMANDED G110 BLOCK (T series) | Blocks containing the G110 codes were successively specified in tool–tip radius compensation for the B–axis. |
| 5043 | TOO MANY G68 NESTING (M series) | Three–dimensional coordinate conversion G68 has been specified three or more times. |
| 5044 | G68 FORMAT ERROR (M series) | A G68 command block contains a format error. This alarm is issued in the following cases:<br>1. I, J, or K is missing from a G68 command block (missing coordinate rotation option).<br>2. I, J, and K are 0 in a G68 command block.<br>3. R is missing from a G68 command block. |
| 5046 | ILLEGAL PARAMETER (ST.COMP) | The parameter settings for straightness compensation contain an error. Possible causes are as follows:<br>1. A parameter for a movement axis or compensation axis contains an axis number which is not used.<br>2. More than 128 pitch error compensation points exist between the negative and positive end points.<br>3. Compensation point numbers for straightness compensation are not assigned in the correct order.<br>4. No straightness compensation point exists between the pitch error compensation points at the negative and positive ends.<br>5. The compensation value for each compensation point is too large or too small. |
| 5050 | ILL–COMMAND IN CHOPPING MODE (M series) | 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. |
| 5051 | M–NET CODE ERROR | Abnormal character received (other than code used for transmission) |
| 5052 | M–NET ETX ERROR | Abnormal ETX code |
| 5053 | M–NET CONNECT ERROR | Connection time monitoring error (parameter No. 175) |
| 5054 | M–NET RECEIVE ERROR | Polling time monitoring error (parameter No. 176) |
| 5055 | M–NET PRT/FRT ERROR | Vertical parity or framing error |
| 5057 | M–NET BOARD SYSTEM DOWN | Transmission timeout error (parameter No. 177)<br>ROM parity error<br>CPU interrupt other than the above |
| 5058 | G35/G36 FORMAT ERROR (T series) | 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. |
| 5059 | RADIUS IS OUT OF RANGE (T series) | A radius exceeding nine digits has been specified for circular interpolation with the center of the arc specified with I, J, and K. |

| Number | Message | Contents |
|---|---|---|
| 5063 | IS NOT PRESET AFTER REF.<br>(M series) | Function category:<br>Workpiece thickness measurement<br>Alarm details<br>The position counter was not preset before the start of workpiece thickness measurement. This alarm is issued in the following cases:<br>(1) An attempt has been made to start measurement without first establishing the origin.<br>(2) An attempt has been made to start measurement without first presetting the position counter after manual return to the origin. |
| 5064 | DIFFERRENT AXIS UNIT (IS–B, IS–C)<br>(M series) | Circular interpolation has been specified on a plane consisting of axes having different increment systems. |
| 5065 | DIFFERENT AXIS UNIT (PMC AXIS)<br>(M series) | 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. |
| 5066 | RESTART ILLEGAL SEQUENCE NUMBER<br>(M series) | Sequence number 7xxx has been read during search for the next sequence number at program restart for the return/restart function. |
| 5068 | G31 P90 FORMAT ERROR<br>(M series) | No movement axis or more than one movement axis has been 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. |
| 5082 | DATA SERVER ERROR | This alarm is detailed on the data server message screen. |

**NOTE**
HPCC : High precision contour control

### (2) Background edit alarm

| Number | Message | Contents |
|---|---|---|
| ??? | BP/S alarm | BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.) |
| 140 | BP/S alarm | It was attempted to select or delete in the background a program being selected in the foreground. **(Note)**<br>Use background editing correctly. |

**NOTE**
Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.