Continue

FA ROBOT ROBOMACHINE IoT The three businesses of FA, ROBOT and ROBOMACHINE are unified with SERVICE as one FANUC, to provide innovation and reassurance to manufacturing sites around the world. News List News List The latest news, case studies, as well as upcoming events from the world's leading supplier of factory automation. Advertisement Fanuc Alarm Codes Wondering what that 8.(dot)alarm is on your Fanuc Servo Drive, This blog will help with your troubleshooting on your Fanuc Servo, Spindle and Power supplies, Alpha drive and older systems Below is a Fanuc alarm code list of Fanuc parts, We can repair most alarms or offer a service exchange part to get your machine back up and running as quickly as possible, If you need any help please call 07568431113 Select your Fanuc Part to quickly jump to that section Fanuc Alpha Servo Drives Fanuc Alpha Spinde Drives Fanuc Alpha Power Supply Fanuc Servo Drives, 6050, 6057, 6058 Fanuc C Series Drives, 6066, 6089, 6090 Fanuc Alpha Servo Drives Alarm Codes A06B-6079-H, A06B-6096-H, etc Alarm CodeFaultRemedyNot ReadyDrive is Good0Control ReadyDrive is Good1Internal Fan StoppedReplace/Repair Servo Drive2Low control power voltage alarm (LV)Replace/Repair Servo Drive3Low DC link voltage alarm (LVDC)Check Alpha Power Supply for Alarm4Regenerative dis-charge control circuit failure alarm (DCSW)Replace/Repair Servo Drive5Over-regenerative discharge alarm (DCOH)Replace/Repair Servo Drive6L-and M-axis over-current alarm (HCLM)Servo Drive or Motor Fault6. DotL-and M-axis IPM alarm (IPMLM)Replace/Repair Servo Drive7Dynamic brake circuit failure alarm (DBRLY)Replace/Repair Servo Drive8L-axis over-current alarm (HCL)Servo Drive or Motor Fault8. DotL-axis IPM alarm (IPML)Replace/Repair Servo Drive9M-axis over-current alarm (HCM)Servo Drive or Motor Fault9. DotM-axis IPM alarm (IPMM)Replace/Repair Servo DriveAN-axis over-current alarm (HCN)Servo Drive or Motor FaultA. DotN-axis IPM alarm (IPMN)Replace/Repair Servo Drive Fanuc Alpha Spindle Drives Alarm Codes A6B-6082-H, A06B-6088-H, etc Alarm CodeFault--Not Ready00Control Ready01Motor Overheat02Excess Speed Deviation03Fuse on DC Link Blown04 Input Fuse Blown05 Control Power Supply Fuse Blown06 Overspeed (Analog) (Temperature Sensor Disconnected)07 Overspeed (Digital)08 High 24v Input Voltage09 Overheat Power Semiconductors10 Low 15v Input Voltage11 DC Link High Voltage12 DC Link Overcurrent13 Data Memory Fault CPU14 Defective ROM15 Defective Optional Circuit16 NVRAM Fault17 NVRAM Checksum Fault18 ROM Checksum Fault19 IU Offset Current Alarm20 IV Offset Current Alarm21 VCMD Offset Alarm (Position Sensor Polarity Setting Incorrect)22 TSA Offset Alarm23 ER Offset Alarm24 Serial Transfer Data Error25 Serial Transfer Stop26 C Velocity Detector Disconnected27 Position Coder Disconnected28 C Position Detector Disconnected29 Short Period Overload30 Overcurrent Power Circuit31 Motor Lock or V Signal Lost32 Serial communication LSI RAM Error33 DC Link Pre Charge Failure34 Parameter Setting Error35 Gear Ratio Parameter Error36 Error Counter Overflow37 Speed Detector Parameter Error39 One Rotation Cs Signal Error40 Cs One Rotation Signal Detection Error41 Position Coder One Rotation Signal Detection Error42 One Rotation Position Coder Detected Error43 Disconnected PC for DIF. SP. MOD. 44 Control Circuit AD Error46 Thread Cutting One Rotation Position Coder Alarm47 Position Coder Signal Abnormal49 High Conversion DIF. SPEED 50 Excessive speed command calculation value during spindle synchronization51 Low Volt DC Link52 ITP Signal Abnormal 153 ITP Signal Abnormal 2 54 Current Overload Alarm55 Abnormal Switching Status of Power Leads56 Inner Fanuc Cooling Fan Stopped57 Excessive Deceleration Power58 Overload in Fanuc Power Supply Module59 Cooling Fan Stopped in PSM 66 Communication alarm between spindle and amplifier69 Safety speed exceeded70 Abnormal axis data71 Abnormal safety parameter72 Motor speed mismatch73 Motor sensor disconnected74 CPU test alarm75 CRC test alarm76 Safety function not executed77 Axis number mismatch78 Safety parameter mismatch79 Abnormal initial test operation81 Motor sensor one-rotation signal detection error82 Motor sensor one-rotation signal not detected83 Motor sensor signal error84 Spindle sensor disconnected85 Spindle sensor one-rotation signal detection error86 Spindle sensor one-rotation signal not detected87 Spindle sensor signal error88 Cooling fan stopped of the radiator A0 Program ROM errorA1 Program ROM errorA2 Program ROM errorb0 Communication error between amplifier and moduleb1 6 Converter control power supply low voltageb2 8 Converter excessive regenerative powerb3 A Converter cooling fan stopped of the radiatorC0 Communication data alarmC1 Communication data alarmC2 Communication data alarmC3 Spindle switching circuit error Fanuc Alpha Power Supplies Alarm Codes A06B-6087-H, A06B-6077-H, etc Alarm CodeFault01Overcurrent in the Main Power Module02Cooling Fan Stopped03The temperature of the main circuit heat sink has risen abnormally04The DC Voltage has dropped05The main Capacitor was not recharged within the specified time06The Input Power Supply is Abnormal07In the main circuit the DC Voltage is abnormally high Fanuc Alarm Codes for the Fanuc AC Servo Dives, A0BB-6050-H, A06B-6057-H, A06B-6058-H etc Alarm CodeFaultHV High Voltage Alarm HC High Current Alarm LV Low Voltage Alarm OVC Overcurrent Alarm TG Tacho Generator Loss DC Discharge Alarm Fanuc Alarm Codes for the Fanuc C Series Servo Dives, A0BB-6066-H, A06B-6089-H, A06B-6089-H etc Alarm Code Fault 1 (HV)DC link over voltage alarm 2 (LV)Low control power voltage alarm 3 (LVDC)Low DC link voltage 4 (DCSW)Regenerative discharge control circuit failure alarm 5 (DCOH)Over-Regenerative discharge alarm 7 DBRLY)Dynamic brake circuit failure alarm 8 (HCL)L axis over- current alarm 8 DOT (IPML)L axis IPM alarm 9 (HCM)M axis over- current alarm 9 DOT (IPMM)M axis IPM alarm b (HCLM)L, M axis over-current alarm b DOT (IPMLM)L, M axis IPM alarm If you like this Fanuc Alarm Code list keep a eye out for more coming soon or give our Facebook page a like hereMore details on Fanuc Repairs here back Built on many decades of expertise and innovation, FANUC has become a trusted partner in delivering reliable and intelligent automation solutions for manufacturing industries worldwide. At FANUC, we are the worlds leading automation company, committed to driving success through innovative solutions and lasting partnerships. Our focus is on building strong, enduring relationships with our customers, ensuring that their growth is at the heart of everything we do. By working closely with businesses, we deliver customised automation solutions that not only meet but exceed expectations. Trust, collaboration, and mutual growth define our approach, making FANUC the partner you can rely on to achieve sustainable success in todays competitive landscape.CNC & DRIVESFANUC CNC provides the answer to demanding machining operations involving multiple axes and multiple path applications. From very small to very large workpieces, from very soft to very hard materials, FANUC CNC provides the ideal balance between utmost precision, flawless surface quality and the shortest possible cycle times. This translates into less non-productive time and eliminates the inaccuracies incurred in multiple-part set-up operations.Discover FANUC CNC SystemsCNC FinderROBOTSWith more than 100 models, FANUC offers the widest range of industrial robots in the world. Covering a diverse range of applications and industries, FANUC robots are easy to operate and provide complete flexibility thanks to a range of application-specific options, straightforward integration, payloads up to 2.3 t and maximum reaches up to 4.7 m.Discover FANUC RobotsROBOT FinderROBODRILLThe new generation ROBODRILL vertical machining centre promises unrivalled quality and precision at great hourly rates. With an unbeatable tool change time of 0.7 seconds and a turret capable of handling tools weighing 4 kg, the new advanced versions are both the fastest and strongest 5-axis CNC machines on the market. Advanced or standard, every ROBODRILL vertical machining centre is a real high-speed all-rounder offering incredible performance and unbeatable efficiency.Discover the ROBODRILL SeriesROBODRILL FinderROBOSHOTFANUC ROBOSHOT takes state-of-the-art CNC precision technology from turning and milling and applies it to electric injection moulding. The results are greater acceleration, ultimate precision of movement and extremely short cycle times to produce large quantities of consistently high-quality parts.Discover the ROBOSHOT SeriesROBOSHOT FinderROBOCUTWhere wire EDM is concerned, accuracy has traditionally come at the cost of speed. Thats why FANUC has developed a next generation ROBOCUT wire-cutting machine. The -CC series comprises versatile all-rounders. With incredibly long mean times between failures, low maintenance, longevity and excellent uptimes, these future-proof spark erosion machines are designed to save time and drive down unit costs while ensuring superlative accuracy and cutting efficiency.Discover the ROBOCUT SeriesROBOCUT HardwareIIoT SolutionsFANUC brings together automation with advanced analytics to drive smarter, faster business decisions for industrial companies, regardless of their size. Discover scalable IIoT products that increase manufacturing efficiency through better production quality, performance, uptime and OEE. Make your production processes more efficient thanks to our IIoT solutions.Discover the IIoT SolutionsDiscover FANUC digital service platform to simplify your business:E-StoreProduct registration and overviewOrders & QuotesExplore our various case studies, covering a wide range of automation solutions for small and large manufacturers operating in all kinds of industries around the world. FANUC CORPORATION has been recognised as one of the Top 100 Global Innovators for 2025, in recognition for its activities in R&D and intellectual property.When partnering with FANUC, you get access to a global network of automation experts who are ready to help you take your manufacturing operations to the next level. We are present in more than 105 countries, covering over 250 locations worldwide. The strong local presence lets you easily get in contact with a local FANUC office near you because we know the important value of being able to meet face-to-face when considering integrating automation into your manufacturing process. Automation solutions are often a considerable investment for your business, and as such, you want to maximise their value throughout their entire lifecycle. At FANUC, we put service first and offer flexible service and support for all FANUC products whenever and wherever you need it, 24/7. At FANUC, we are deeply committed to sustainability, integrating environmental protection into our automation solutions. Our manufacturing technologies prioritise responsible machine operations, aiming to reduce energy consumption, promote waste reduction and minimise environmental impact. With accolades and partnerships in nature conservation, we strive for continuous improvement in environmental credentials worldwide. FANUC CORPORATION has been recognised as one of the Top 100 Global Innovators for 2025, in recognition for its activities in R&D and intellectual property.Join the FANUC Team! Explore exciting career opportunities and be part of a global leader in factory automation. Discover your potential with FANUC today! A FANUC alarm code, also called a FANUC fault or error code, is how a CNC control indicates there is a problem. This error message could indicate issues with either the machine itself, be that electric or mechanical. The FANUC error code might also indicate an issue with the g-code program.How to Diagnose Your FANUC Alarm CodeUse the list below to interpret what exactly the fault code is trying to communicate. If you require a replacement part, know that MRO Electric stocks thousands of FANUC CNC replacements. Get your new FANUC servo amplifier or check out our FANUC servo motors. To order a replacement part or a repair job, please call 800-691-8511 or email sales@mroelectric.com. Curious about other common FANUC CNC problems? Check out our FAQ here. These FANUC Alarms are valid for following FANUC Controls: 0i Model A 0i/0iMate Model B 16/18 Model PB 16/18 Model C 16i/18i Model A 16i/18i Model B 16iL Model A 20i 21 Model B 21i Model A 21i Model B 21i Model AAlarmDescription0 PLEASE TURN OFF POWERA parameter which requires the power off was input, turn off power.1 TH PARITY ALARMTH alarm (A character with incorrect parity was input). Correct the tape.2 TV PARITY ALARMTV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.3 TOO MANY DIGITSData exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)4 ADDRESS NOT FOUNDA numeral or the sign was input without an address at the beginning of a block. Modify the program.5 NO DATA AFTER ADDRESSThe address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.6 ILLEGAL USE OF NEGATIVE SIGNSign input error (Sign was input after an address with which it cannot be used. Or two or more signs were input.) Modify the program.7 ILLEGAL USE OF DECIMAL POINTDecimal point . input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.9 ILLEGAL ADDRESS INPUTUnusable character was input in significant area. Modify the program.10 IMPROPER GCODEAn unusable G code or G code corresponding to the function not provided is specified. Modify the program.11 NO FEEDRATE COMMANDEDFeedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.14 CAN NOT COMMAND G95 (M Series)A synchronous feed is specified without the option for threading / synchronous feed.14 ILLEGAL LEAD COMMAND (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. Modify the program.15 TOO MANY AXES COMMANDEDM Series;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. Modify the program.20 OVER TOLERANCE OF RADIUSIn 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. 3410.21 ILLEGAL PLANE AXIS COMMANDEDAn axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.22 NO CIRCULAR RADIUSWhen circular interpolation is specified, neither R (specifying an arc radius), nor I, J, and K (specifying the distance from a start point to the center) is specified.23 ILLEGAL RADIUS COMMAND (T series)In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.25 CANNOT COMMAND F0 IN G02/G03F0 (fast feed) was instructed by F1 digit column feed in circular interpolation. Modify the program.27 NO AXES COMMANDED IN G43/G44No 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. Modify the program.28 ILLEGAL PLANE SELECTIn the plane selection command, two or more axes in the same direction are commanded. Modify the program.29 ILLEGAL OFFSET VALUEM Series;The offset values specified by H code is too large. Modify the program.30 ILLEGAL OFFSET NUMBERM Series;The offset values specified by D/H code for tool length offset, cutter compensation or 3dimensional cutter compensation is too large. Otherwise, the number specified by P code for the additional workpiece coordinate system is too large. Modify the program.31 ILLEGAL P COMMAND IN G10In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.32 ILLEGAL OFFSET VALUE IN G10In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.33 NO SOLUTION AT CRCM Series: A point of intersection cannot be determined for cutter compensation C. Modify the program. T Series: A point of intersection cannot be determined for tool nose radius compensation. Modify the program.34 NO CIRC ALLOWED IN STUP /EXT BLKM Series;The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. Modify the program.35 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.35 CAN NOT COMMANDED G31T series;Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.36 CAN NOT COMMANDED G31Skip cutting (G31) was specified in cutter compensation mode.Modify the program.37 CAN NOT CHANGE PLANE IN CRCM 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.38 INTERFERENCE IN CIRCULAR BLOCKOvercutting will occur in tool radius/tool nose radius/cutter compensation because the arc start point or end point coincides with the arc center. Modify the program.39 CHF/CNR NOT ALLOWED IN NRCChamfering or corner R was specified with a startup, 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.40 INTERFERENCE IN G90/G94 BLOCKOvercutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.41 INTERFERENCE IN CRCM 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.42 G45/G48 NOT ALLOWED IN CRCTool offset (G45 to G48) is commanded in cutter compensation. Modify the program.44 G27G30 NOT ALLOWED IN FIXED CYCOne of G27 to G30 is commanded in canned cycle mode.Modify the program.45 ADDRESS Q NOT FOUND(G73/G83)In canned cycle G73/G83, the depth of each cut (Q) is not specified. Alternatively,Q0 is specified. Correct the program.46 ILLEGAL REFERENCE RETURN COMMANDOther than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.47 ILLEGAL AXIS SELECTTwo or more parallel axes (in parallel with a basic axis) have been specified upon startup of threedimensional tool compensation or threedimensional coordinate conversion.48 BASIC 3 AXIS NOT FOUNDFor startup of threedimensional tool compensation or threedimensional coordinate conversion, the three basic axes used when Xp,Yp and Zp are omitted were not specified in parameter No. 1022.49 ILLEGAL OPERATION (G68/G69)The commands for threedimensional coordinate conversion (G68,G69) and tool length compensation (G43, G44, G45) are not nested.Modify the program.50 CHF/CNR NOT ALLOWED IN THRD BLKChamfering or corner R is commanded in the thread cutting block. Modify the program.51 MISSING MOVE AFTER CHF/CNRT series;Improper movement or the move distance was specified in the block next to the chamfering or corner R block.Modify the program.52 CODE IS NOT G01 AFTER CHF/CNRThe block next to the chamfering or corner R block is not G01, G02,or G03.Modify the program.53 TOO MANY ADDRESS COMMANDSM 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.54 NO TAPER ALLOWED AFTER CHF/CNRT series;A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.55 MISSING MOVE VALUE IN CHF/CNRIn chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.56 NO END POINT & ANGLE IN CHF/CNRT 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.57 NO SOLUTION OF BLOCK ENDBlock end point is not calculated correctly in direct dimension drawing programming. Modify the program.58 END POINT NOT FOUNDIn a arbitrary angle chamfering or corner R cutting block, a specified axis is not in the selected plane. Correct the program.59 PROGRAM NUMBER NOT FOUNDIn an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Alternatively, the program with the program number specified in a onetouch macro call is not found in memory. Check the program number and external signal. Or discontinue the background editing.60 SEQUENCE NUMBER NOT FOUNDCommanded sequence number was not found in the sequence number search.Check the sequence number.61 ADDRESS P/Q NOT FOUND IN G70G73T series;Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.62 ILLEGAL COMMAND IN G71G76T series;1. The depth of cut in G71 or G72 is zero or negative value.;2. The repetitive count in G73 is zero or negative value.;3. the negative value is specified to i or k is zero in G74 or G75.;4. A value other than zero is specified to address U or W though i or k is zero in G74 or G75.;5. A negative value is specified to d, though the relief direction in G74 or G75 is determined.;6. Zero or a negative value is specified to the height of thread or depth of cut of first time in G76.;7. The specified minimum depth of cut in G76 is greater than the height of thread.;8. An unusable angle of tool tip is specified in G76.;Modify the program.63 SEQUENCE NUMBER NOT FOUNDT series;The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.64 SHAPE PROGRAM NOT MONOTONOUSLYT series;A target shape which cannot be made by monotonic machining was specified in a repetitive canned cycle (G71 or G72).65 ILLEGAL COMMAND IN G71G73T 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.;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. Modify the program.66 IMPROPER GCODE IN G71G73T series;An unallowable G code was commanded beween two blocks specified by address P in G71, G72, or G73. Modify the program.67 CAN NOT ERROR IN MDI MODET series;G70, G71, G72, or G73 command with address P and Q. Modify the program.69 FORMAT ERROR IN G70G73T 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.70 NO PROGRAM SPACE IN MEMORYThe memory area is insufficient.Delete any unnecessary programs, then retry.71 DATA NOT FOUNDThe address to be searched was not found.Or the program with specified program number was not found in program number search.Check the data.72 TOO MANY PROGRAMSThe number of programs to be stored exceeded 63 (basic), 125 (option),200 (option), or 400 (option) or 1000 (option). Delete unnecessary programs and execute program registeration again.73 PROGRAM NUMBER ALREADY IN USEThe commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registeration again.74 ILLEGAL PROGRAM NUMBERThe program number is other than 1 to 9999.Modify the program number.75 PROTECTAn attempt was made to register a program whose number was protected.76 ADDRESS P NOT DEFINEDAddress P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.77 SUB PROGRAM NESTING ERRORThe subprogram was called in five folds. Modify the program.78 NUMBER NOT FOUNDA 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.79 PROGRAM VERIFY ERRORIn 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.80 G37 ARRIVAL SIGNAL NOT ASSERTEDM 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 (value).This is due to a setting or operator error.81 OFFSET NUMBER NOT FOUND IN G37M series;Tool length automatic measurement (G37) was specified without a H code.(Automatic tool length measurement function) Modify the program.82 HCODE NOT ALLOWED IN G37M series;H code and automatic tool compensation (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.82 TCODE NOT ALLOWED IN G37T series;T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.83 ILLEGAL AXIS COMMAND IN G37M series;In automatic tool length measurement, an invalid axis was specified or the command is incremental.Modify the program.85 COMMUNICATION ERRORWhen 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.86 DR SIGNAL OFFWhen entering data in the memory by using Reader / Puncher interface,the ready signal (DR) of reader / puncher was off.Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.87 BUFFER OVERFLOWWhen 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.88 LAN FILE TRANS ERROR (CHANNEL1)File data transfer over the OSIEthernet was terminated as a result of a transfer error.89 LAN FILE TRANS ERROR (CHANNEL2)File data transfer over the OSIEthernet was terminated as a result of a transfer error.90 REFERENCE RETURN INCOMPLETE21 Model B;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.91 REFERENCE RETURN INCOMPLETEIn the automatic operation halt state, manual reference position return cannot be performed.92 AXES NOT ON THE REFERENCE POINTThe commanded axis by G27 (Reference position return check) did not return to the reference position.94 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 operators manual.95 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 Users manual.96 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 Users manual.97 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 alarm 94 to 97 were reset, no automatic operation is performed).Perform automatic operation.98 G28 FOUND IN SEQUENCE RETURNA 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.99 MDI EXEC NOT ALLOWED AFT.SEARCHAfter completion of search in program restart, a move command is given with MDI. Move axis before a move command or dont interrupt MDI operation.100 PARAMETER WRITE ENABLEOn the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system.101 PLEASE CLEAR MEMORYThe 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.Register the deleted program.109 FORMAT ERROR IN G08A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.110 DATA OVERFLOWThe absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.111 CALCULATED DATA OVERFLOWThe result of calculation turns out to be invalid, an alarm No.111 is issued. 10 47 to 10 29, 0, 10 29 to 10 47 Modify the program.112 DIVIDED BY ZERODivision by zero was specified.(including tan 90) Modify the program.113 IMPROPER COMMANDA function which cannot be used in custom macro is commanded. Modify the program.114 FORMAT ERROR IN MACROThere is an error in other formats than (Formula). Modify the program.115 ILLEGAL VARIABLE NUMBERA value not defined as a variable number is designated in the custom macro. Modify the program.116 WRITE PROTECTED VARIABLEThe left side of substitution statement is a variable whose substitution is inhibited. Modify the program.118 PARENTHESIS NESTING ERRORThe nesting of bracket exceeds the upper limit (quintuple). Modify the program.119 ILLEGAL ARGUMENTThe SQRT argument is negative, BCD argument is negative, or other values than 0 to 9 are present on each line of BIN argument. Modify the program.122 FOUR FOLD MACRO MODALCALLThe macro modal call is specified four fold. Modify the program.123 CAN NOT USE MACRO COMMAND IN DNCMacro control command is used during DNC operation. Modify the program.124 MISSING END STATEMENTDO 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 NUMBERIn DOn, 1 = n = 3 is not established. Modify the program.127 NC, MACRO STATEMENT IN SAME BLOCKNC and custom macro commands coexist. Modify the program.128 ILLEGAL MACRO SEQUENCE NUMBERThe sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched.Modify the program.129 ILLEGAL ARGUMENT ADDRESSAn address which is not allowed in (Argument Designation) is used.Modify the program.130 ILLEGAL AXIS OPERATIONAn 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 MESSAGESFive or more alarms have generated in external alarm message.Consult the PMC ladder diagram to find the cause.132 ALARM NUMBER NOT FOUNDNo alarm No.concerned exists in external alarm message clear.Check the PMC ladder diagram.133 ILLEGAL DATA IN EXT. ALARM MSGSmall section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.135 ILLEGAL ANGLE COMMANDM series;The index table indexing positioning angle was instructed in other than an integral multiple of the value of the minimum angle. Modify the program.135 SPINDLE ORIENTATION PLEASET series;Without any spindle orientation, an attept was made for spindle indexing. Perform spindle orientation.136 ILLEGAL AXIS COMMANDM series;In index table indexing.Another control axis was instructed together with the B axis. Modify the program.136 C/HCODE & 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 MCODE & MOVE CMD IN SAME BLK.A move command of other axes was specified to the same block as Mcode related to spindle indexing. Modify the program.138 SUPERIMPOSED DATA OVERFLOWThe 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 AXISAn axis is selected in commanding by PMC axis control. Modify the program.141 CAN NOT COMMAND G51 IN CRCM series;G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.142 ILLEGAL SCALE RATEM series;Scaling magnification is commanded in other than 1 to 999999. Correct the scaling magnification setting (G51 Pp or parameter 5411 or 5421).143 SCALED MOTION DATA OVERFLOWM 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 SELECTEDM 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 INTERPOLATIONThe conditions are incorrect when the polar coordinate interpolation starts or it is canceled.;1) In modes other than G40, G12.1/G13.1 was specified.;2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified.;Modify the value of program or parameter.146 IMPROPER G CODEG codes which cannot be specified in the polar coordinate interpolation mode was specified. See section II to 4.4 and modify the program.148 ILLEGAL SETTING DATAM series;Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714)149 FORMAT ERROR IN G10L3M 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 NUMBERTool Group No. exceeds the maximum allowable value. Modify the program.151 TOOL GROUP NUMBER NOT FOUNDThe tool group commanded in the machining program is not set. Modify the value of program or parameter.152 NO SPACE FOR TOOL ENTRYThe number of tools within one group exceeds the maximum value registerable. Modify the number of tools.153 TCODE NOT FOUNDIn tool life data registration, a T code was not specified where one should be. Correct the program.154 NOT USING TOOL IN LIFE GROUPM series;When the group is not commanded, H99 or D99 was commanded. Correct the program.155 ILLEGAL TCODE IN M06M series;In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.156 P/L COMMAND NOT FOUNDP and L commands are missing at the head of program in which the tool group is set. Correct the program.157 TOO MANY TOOL GROUPSThe 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 DATAThe tool life to be set is too excessive. Modify the setting value.159 TOOL DATA SETTING INCOMPLETEDuring executing a life data setting program, power was turned off. Set again.163 COMMAND G68/G69 INDEPENDENTLYG68 and G69 are not independently commanded in balance cut. Modify the program.175 ILLEGAL G107 COMMANDConditions 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 to axis name radius of cylinder.176 IMPROPER GCODE IN G107M series;Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified.;1) G codes for positioning: G28,, G73, G74, G76, G81 G89, including the codes specifying the rapid traverse cycle;2) G codes for setting a coordinate system: G52,G92;3) G code for selecting coordinate system: G53 G54G59 Modify the program.177 CHECK SUM ERROR (G05 MODE)Check sum error Modify the program.178 G05 COMMANDED IN G41/G42 MODEG05 was commanded in the G41/G42 mode. Correct the program.179 PARAM. (NO. 7510) SETTING ERRORThe 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 BLOCKG81 block format error (hobbing machine);1) T (number of teeth) has not been instructed.;2) Data outside the command range was instructed by either T, L, Q or P.;3) An overflow occurred in synchronization coefficient calculation. Modify the program.182 G81 NOT COMMANDEDG83 (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)G83 was instructed before canceled by G82 after compensating for the C axis servo lag quantity by G83. (hobbing machine)184 ILLEGAL COMMAND IN G81A command not to be instructed during synchronization by G81 was instructed. (hobbing machine);1) A C axis command by G00, G27, G28, G29, G30, etc. was instructed.;2) Inch/Metric switching by G20, G21 was instructed.185 RETURN TO REFERENCE POINTG81 was instructed without performing reference position return after power on or emergency stop. (hobbing machine) Perform reference position return.186 PARAMETER SETTING ERRORParameter error regarding G81 (hobbing machine);1) The C axis has not been set to be a rotary axis.;2) A hob axis and position coder gear ratio setting error Modify the parameter.187 HOB COMMAND IS NOT ALLOWEDError in the modal state when G81.4 or G81 is specified;1. The canned cycle mode (G81 to G89) is set.;2. The thread cutting mode is set.;3. The Caxis is under synchronous, composite, or superimposed control.190 ILLEGAL AXIS SELECTM 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. Correct the program.194 SPINDLE COMMAND IN SYNCHROMODEA contour control mode, spindle positioning (Csaxis 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 ERRORSwitching command to contouring mode, Cs axis control or rigid tap mode or switching to spindle command mode is not correctly completed. (This occurs when the response to switch to the spindle control unit side with regard to the switching command from the NC is incorrect. 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 CAXIS COMMANDED IN SPINDLE MODEThe program specified a movement along the Csaxis 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 UNDEFINEDUndefined macro word was used. Modify the custom macro.200 ILLEGAL S CODE COMMANDIn the rigid tap, an S value is out of the range or is not specified. Modify the program.201 FEEDRATE NOT FOUND IN RIGID TAPIn the rigid tap, no F value is specified. Correct the program.202 POSITION LSI OVERFLOWIn the rigid tap, spindle distribution value is too large. (System error)203 PROGRAM MISS AT RIGID TAPPINGIn the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.204 ILLEGAL AXIS OPERATIONIn 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 OFFRigid 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. Correct the program.207 RIGID DATA MISMATCHThe specified distance was too short or too long in rigid tapping.210 CAN NOT COMAND M198/M199M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.;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.;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 G99G31 is commanded in the per revolution command when the high speed skip option is provided. Modify the program.212 ILLEGAL PLANE SELECTM series;The arbitrary angle chamfering or a corner R is commanded or the plane including an additional axis. Correct the program.213 ILLEGAL COMMAND IN SYNCHROMODEMovement is commanded for the

attempt has been made to start measurement without first presetting the position counter after manual return to the origin.5064 DIFFERENT AXIS UNIT (ISB, ISC)Circular interpolation has been specified on a plane consisting of axes having different increment systems.5065 DIFFERENT 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. 8010.5067 (HPCC)HPCC mode cannot be canceled during G51 (scaling) or G68 (coordinate system rotation). Correct the program.5068 G31 FORMAT ERRORThe continuous highspeed skip command (G31 P90) has one of the following errors;1. The axis along which the tool is moved is not specified.;2. More than one axis is specified as the axis along which the tool is moved.5069 WHLC:ILLEGA PDATAThe P data in selection of the grindingwheel wear compensation center is illegal.5073 NO DECIMAL POINTA decimal point is not specified for a command for which a decimal point must be specified.5074 ADDRESS DUPLICATION ERORThe 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 ERORThis alarm is detailed on the data server message screen.5085 SMOOTH IPL ERROR 1A block for specifying smooth interpolation contains a syntax error.5096 MISMATCH WAITING MCODEDifferent wait codes (M codes) were specified in HEAD1 and HEAD2. Correct the program.5110 IMPROPER GCODEAn illegal G code was specified in lookahead control mode (multi blocks are read in advance). A command was specified for the index table indexing axis in look ahead control mode (multi blocks are read in advance).5111 IMPROPER MODAL GCODEM series;An illegal G code is left modal when lookahead control mode was specified.5112 G08 CAN NOT BE COMMANDEDM series;Lookahead control (G08) was specified in lookahead control mode (multi blocks are read in advance).5113 CAN NOT ERROR IN MDI MODELookahead control (multi blocks are read in advance) (G05.1) was specified in MDI mode.5114 NOT STOP POSITION (G05.1 Q1)At the time of restart after manual intervention, the coordinates at which the manual intervention occurred have not been restored.5115 SPL: ERORThere is an error in the specification of the rank. No knot is specified. The knot specification has an error. The number of axes exceeds the limits. Other program errors5116 SPL: ERORThere is a program error in a block under lookahead control. Monotone increasing of knots is not observed. In NURBS interpolation mode, a mode that cannot be used together is specified.5117 SPL: ERORThe first control point of NURBS is incorrect.5118 SPL: ERORAfter manual intervention with manual absolute mode set to on, NURBS interpolation was restarted.5122 ILLEGAL COMMAND IN SPIRALA 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) Three or more axes are specified as the height axes.;6) A height increment is specified when there are two height axes.;7) Conical interpolation is specified when the helical interpolation function is not selected.;8) Q 0 is specified when radius difference 0.;9) Q 0 is specified when radius difference 0.;10) A height increment is specified when no height axis is specified.5123 OVER TOLERANCE OF END POINTThe difference between a specified end point and the calculated end point exceeds the allowable range (parameter 3471).5124 CAN NOT COMMAND SPIRALA spiral interpolation or conical interpolation was specified in any of the following modes;1) Scaling;2) Programmable mirror image;3) Polar coordinate interpolation In cutter compensation C mode, the center is set as the start point or end point.5134 FSSB: OPEN READY TIME OUTInitialization did not place FSSB in the open ready state.5135 FSSB: ERROR MODEFSSB has entered error mode.5136 FSSB: NUMBER OF AMPS IS SMALLIn comparison with the number of controlled axes, the number of amplifiers recognized by FSSB is not enough.5137 FSSB: CONFIGURATION ERRORFSSB detected a configuration error.5138 FSSB: AXIS SETTING NOT COMPLETEIn automatic setting mode, axis setting has not been made yet. Perform axis setting on the FSSB setting screen.5139 FSSB: ERORServo initialization did not terminate normally. The optical cable may be defective, or there may be an error in connection to the amplifier or another module. Check the optical cable and the connection status.5155 NOT RESTART PROGRAM BY G05During servo leaning control by G05, an attempt was made to perform restart operation after feed hold or interlock. This restart operation cannot be performed. (G05 leaning control terminates at the same time.)5156 ILLEGAL AXIS OPERATIONIn lookahead control mode (multi blocks are read in advance), the controlled axis selection signal (PMC axis control) changes. In lookachead control mode (multi blocks are read in advance), the simple synchronous axis selection signal changes.5157 PARAMETER ZEROZero is set in the parameter for the maximum cutting feedrate (parameter No. 1422 or 1432). Zero is set in the parameter for the acceleration/deceleration before interpolation (parameter No. 1770 or 1771). Set the parameter correctly.5195 DIRECTION CAN NOT BE JUDGEDWhen the touch sensor with a single contact signal input is used in the direct input B function for tool offset measurement values, the stored pulse direction is not constant. One of the following conditions exists; The stop state exists in offset write mode.; Servo off state; The direction varies.; Movement takes place simultaneously along two axes.5196 ILLEGAL OPERATION (HPCC)Detach operation was performed in HPCC mode. (If detach operation is performed in HPCC mode, this alarm is issued after the currently executed block terminates.)5197 FSSB: OPEN TIME OUTThe CNC permitted FSSB to open, but FSSB was not opened.5198 FSSB: ID DATA NOT READTemporary assignment failed, so amplifier initial ID information could not be read.5199 FINE TORQUE SENSING PARAMETERA parameter related to the fine torque sensing function is illegal.; The storage interval is invalid.; An invalid axis number is set as the target axis. Correct the parameter.5212 SCREEN COPY: PARAMETER ERORThere is a parameter setting error. Check that 4 is set as the I/O channel.5213 SCREEN COPY: COMMUNICATION ERORThe memory card cannot be used. Check the memory card. (Check whether the memory card is writeprotected or defective.)5214 SCREEN COPY: DATA TRANSFER ERORData transfer to the memory card failed. Check whether the memory card space is insufficient and whether the memory card was removed during data transfer.5218 ILLEGAL PARAMETER (INCL. COMP)There is an inclination compensation parameter setting error. Cause;1. The number of pitch error compensation points between the negative () end and positive (+) end exceeds 128.;2. The relationship in magnitude among the inclination compensation point numbers is incorrect.;3. An inclination compensation point is not located between the negative () end and positive (+) end of the pitch error compensation points.;4. The amount of compensation per compensation point is too large or too small. Correct the parameter.5219 CAN NOT RETURNManual intervention or return is not allowed during threedimensional coordinate conversion.5220 REFERENCE POINT ADJUSTMENT MODEA parameter for automatically set a reference position is set. (Bit 2 of parameter No. 1819 = 1) Perform automatic setting. (Position the machine at the reference position manually, then perform manual reference position return.) Supplementary: Automatic setting sets bit 2 of parameter No. 1819 to 0.5222 SRAM CORRECTABLE ERORThe SRAM correctable error cannot be corrected. Cause: A memory problem occurred during memory initialization. Action: Replace the master printed circuit board (SRAM module).5227 FILE NOT FOUNDA specified file is not found during communication with the builtin Handy File.5228 SAME NAME USEDThere are duplicate file names in the builtin Handy File.5229 WRITE PROTECTEDA floppy disk in the builtin Handy File is write protected.5231 TOO MANY FILESThe number of files exceeds the limit during communication with the builtin Handy File.5232 DATA OVERFLOWThere is not enough floppy disk space in the builtin Handy File.5235 COMMUNICATION ERORA communication error occurred during communication with the builtin Handy File.5237 READ ERORA floppy disk in the builtin Handy File cannot be read from. The floppy disk may be defective, or the head may be dirty. Alternatively, the Handy File is defective.5238 WRITE ERORA floppy disk in the builtin Handy File cannot be written to. The floppy disk may be defective, or the head may be dirty. Alternatively, the Handy File is defective.5242 ILLEGAL AXIS NUMBERThe axis number of the synchronous master axis or slave axis is incorrect. (This alarm is issued when flexible synchronization is turned on.) Alternatively, the axis number of the slave axis is smaller than that of the master axis.5243 DATA OUT OF RANGEThe gear ratio is not set correctly. (This alarm is issued when flexible synchronization is turned on.)5244 TOO MANY DI ONEven when an M code was encountered in automatic operation mode, the flexible synchronization mode signal was not driven on or off. Check the ladder and M codes.5245 OTHER AXIS ARE COMMANDEDOne of the following command conditions was present during flexible synchronization or when flexible synchronization was turned on;1. The synchronous master axis or slave axis is the EGB axis.;2. The synchronous master axis or slave axis is the chopping axis.;3. In reference position return mode5251 ILLEGAL PARAMETER IN G54.2A fixture offset parameter (No. 7580 to 7588) is illegal. Correct the parameter.5252 ILLEGAL P COMMAND IN G54.2The P value specifying the offset number of a fixture offset is too large. Correct the program.5257 G41/G42 NOT ALLOWED IN MDI MODEM series;G41/G42 (cutter compensation C: M series) was specified in MDI mode. (Depending on the setting of bit 4 of parameter No. 5008)5300 SET ALL OFFSET DATAS AGAINAfter the inch/metric automatic conversion function (OIM: Bit 0 of parameter No. 5006) for tool offset data is enabled or disabled, all the tool offset data must be reset. This message reminds the operator to reset the data. If this alarm is issued, reset all the tool offset data. Operating the machine without resetting the data will result in a malfunction.5302 ILLEGAL COMMAND IN G68 MODEA command to set the coordinate system is specified in the coordinate system rotation mode.5303 TOUCH PANEL ERORA touch panel error occurred. Cause;1. The touch panel is kept pressed.;2. The touch panel was pressed when power was turned on. Remove the above causes, and turn on the power again.5306 MODE CHANGE ERORIn a onetouch macro call, mode switching at the time of activation is not performed correctly.5307 INTERNAL DATA OVER FLOWIn the following function, internal data exceeds the allowable range.;1) Improvement of the rotation axis feedrate5311 FSSB:ILLEGAL CONNECTIONA connection related to FSSB is illegal. This alarm is issued when either of the following is found;1. Two axes having adjacent servo axis numbers (parameter No. 1023), odd number and even number, are assigned to amplifiers to which different FSSB systems are connected.;2. The system does not satisfy the requirements for performing HRV control, and use of two pulse modules connected to different FSSB systems having different FSSB current control cycles is specified.5321 SCOMP. VALUE OVERFLOWThe straightness compensation value has exceeded the maximum value of 32767.After this alarm is issued, make a manual reference position return.5400 SPL:ILLEGAL AXIS COMMANDAn axis specified for spline interpolation or smooth interpolation is incorrect. If an axis that is not the spline axis is specified in spline interpolation mode, this alarm is issued. The spline axis is the axis specified in a block containing G06.1 or the next block. For smooth interpolation, the axis specified in G5.1Q2 is incorrect.5401 SPL:ILLEGAL COMMANDIn a G code mode in which specification of G06.1 is not permitted, G06.1 is specified.5402 SPL:ILLEGAL AXIS MOVINGA movement is made along an axis that is not the spline interpolation axis. For example, in threedimensional tool compensation mode using an offset vector of which components are the X, Y, and Zaxes, when twoaxis spline interpolation is performed with the two spline axes set to the X and Yaxes, a movement along the Zaxis occurs, resulting in this alarm.5403 SPL:CAN NOT MAKE VECTORThreedimensional tool compensation vectors cannot be generated.; When a threedimensional tool compensation vector is created for the second or subsequent point, that point, previous point, and next point are on the same straight line, and that straight line and the three dimensional tool compensation vector for the previous point are in parallel.; When a threedimensional tool compensation vector is created at the end point of smooth interpolation or spline interpolation, the end point and the point two points before are the same.5405 ILLEGAL PARAMETER IN G41.2/ G42.2The parameter setting that determines the relationship between the rotation axis and rotation plane is incorrect.5406 G41.3/G40 FORMAT ERROR1) A G41.3 or G40 block contains a move command.;2) A G1.3 block contains a G code or M code for which buffering is suppressed.5407 ILLEGAL COMMAND IN G41.31) A G code that belongs to group 01 except G00 and G01 is specified in G41.3 mode.;2) An offset command (a G code belonging to group 07) is specified in G41.3 mode.;3) The block next to G41.3 (startup) contains no movement.5408 G41.3 ILLEGAL START_UP1) In a mode of group 01 except G00 and G01, G41.3 (startup) is specified.;2) At startup, the included angle of the tool direction vector and move direction vector is 0 or 180 degrees.5409 ILLEGAL PARAMETER IN G41.3The parameter setting (No. xxxx to xxxx) that determines the relationship between the rotation axis and rotation plane is incorrect.5411 NURBS:ILLEGAL ORDERThe number of steps is specified incorrectly.5412 NURBS:NO KNOT COMMANDNo knot is specified. Alternatively, in NURBS interpolation mode, a block not relating to NURBS interpolation is specified.5413 NURBS:ILLEGAL AXIS COMMANDAn axis not specified with controlled points is specified in the first block.5414 NURBS:ILLEGAL KNOTThe number of blocks containing knots only is insufficient.5415 NURBS:ILLEGAL CANCELAlthough NURBS interpolation is not completed yet, the NURBS interpolation mode is turned off.5416 NURBS:ILLEGAL MODEA mode that cannot be used with NURBS interpolation mode is specified in NURBS interpolation mode.5417 NURBS:ILLEGAL MULTIKNOTAs many knots as the number of steps are not specified at the start and end points.5418 NURBS:ILLEGAL KNOT VALUEKnots do not increase in monotone.5420 ILLEGAL PARAMETER IN G43.4/ G43.5A parameter related to pivot tool length compensation is incorrect.5421 ILLEGAL COMMAND IN G43.4/ G43.5In pivot tool length compensation (type 2) mode, a rotation axis is specified.5422 EXCESS VELOCITY IN G43.4/ G43.5As a result of pivot tool length compensation, an attempt was made to move the tool along an axis at a feedrate exceeding the maximum cutting feedrate.5425 ILLEGAL OFFSET VALUEThe offset number is incorrect.5430 ILLEGAL COMMAND IN 3D CIRIn a modal state in which threedimensional circular interpolation cannot be specified, a threedimensional circular interpolation (G02.4/G03.4) is specified. Alternatively, in threedimensional circular interpolation mode, a code that cannot be specified is specified.5432 G02.4/G03.4 FORMAT ERORA threedimensional circular interpolation command (G02.4/G03.4) is incorrect.5433 MANUAL INTERVENTION IN 3D CIRIn threedimensional circular interpolation mode (G02.4/G03.4), manual intervention was made when the manual absolute switch was on.5435 PARAMETER OUT OF RANGEIncorrect parameter setting (set value range)5436 PARAMETER SETTING ERROR 1Incorrect parameter setting (setting of the rotation axis)5437 PARAMETER SETTING ERROR 2Incorrect parameter setting (setting of the tool axis)5440 ILLEGAL DRILLING AXIS SELECTEDThe drilling axis specified for the drilling canned cycle is incorrect. The G code command block of the canned cycle does not specify the Z point of the drilling axis. When there is a parallel axis with the drilling axis, the parallel axis is also specified at the same time.5445 CRC:MOTION IN G39Corner circular interpolation (G39) of cutter compensation is not specified alone but is specified with a move command.5446 CRC:NO AVOIDANCEBecause there is no interference evade vector, the interference check evade function of cutter compensation cannot evade interference.5447 CRC:DANGEROUS AVOIDANCEThe interference check evade function of cutter compensation determines that an evade operation will lead to danger.5448 CRC:INTERFERENCE TO AVD.In the interference check evade function of cutter compensation, a further interference occurs for an already created interference evade vector.5452 IMPROPER GCODE (5AXIS MODE)A G code that cannot be specified is found. (5axis mode) This alarm is issued when;1) Threedimensional cutter compensation (sideface offset and leading edge offset) is applied during cutter compensation, or cutter compensation is applied during threedimensional cutter compensation (sideface offset and leadingedge offset).;2) A leadingedge offset of threedimensional cutter compensation is applied during sideface offsetting of threedimensional cutter compensation, or a sideface offset of threedimensional cutter compensation is applied during leadingedge offsetting of threedimensional cutter compensation.;3) Tool axis direction tool length compensation is applied during tool length compensation, or tool length compensation is applied during tool axis direction tool length compensation.;4) Tool center point control is provided during tool length compensation, or tool length compensation is applied during tool center point control.;5) Tool center point control is provided during tool axis direction tool length compensation, or tool axis direction tool length compensation is applied during tool center point control.;If this alarm is issued, cancel the relevant mode, then specify a different mode.5453 NOTE: G68 IS CANCELEDWhen bit 2 of parameter No. 5400 is set to 1, and a reset does not cancel G68, this alarm is issued at the time of program restart. To release this alarm, press RESET and CAN. Once this operation is performed, the alarm will not be issued at the next restart.5455 ILLEGAL ACC. PARAMETERA permissible acceleration parameter for optimum torque acceleration/ deceleration is incorrect. The cause is one of the following;1) The ratio of the deceleration rate to the acceleration rate is below the limit.;2) The time required for deceleration to a speed of 0 exceeds the maximum value.400 SERVO ALARM: nTH AXIS OVERLOADThe nth axis (axis 14) overload signal is on. Refer to diagnosis display No. 200, 201 for details.401 SERVO ALARM: nTH AXIS VRDY OFFThe nth axis (axis 18) servo amplifier READY signal (DRDY) went off. Refer to procedure of trouble shooting.402 SERVO ALARM: SV CARD NOT EXISTThe axis control card is not provided.403 SERVO ALARM: CARD/SOFT MISMATCHThe combination of the axis control card and servo software is illegal.404 SERVO ALARM: nTH AXIS VRDY ONEven though the nth axis (axis 18) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the servo interface module and servo amp are connected.405 SERVO ALARM: (ZERO POINT RETURN FAULT)Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.407 SERVO ALARM: EXCESS ERORThe difference in synchronous axis position deviation exceeded the set value.409 SERVO ALARM: n AXIS TORQUE ALMAbnormal load has been detected on the servo motor(s). Alternatively, abnormal spindle motor load has been detected in Cs mode.410 SERVO ALARM: nTH AXIS EXCESS ERORThe position deviation value when the nth axis (axis 18) stops is larger than the set value. Refer to procedure of trouble shooting.411 SERVO ALARM: nTH AXIS EXCESS ERORThe position deviation value when the nth axis (axis 18) moves is larger than the set value. Refer to procedure of trouble shooting.413 SERVO ALARM: nth AXIS LSI OVERFLOWThe contents of the error register for the nth axis (axis 18) exceeded 231 power. This error usually occurs as the result of an improperly set parameters.414 SERVO ALARM: nTH AXIS DETECTION RELATED ERORNth axis (axis 14) digital servo system fault. Refer to diagnosis display No. 200, 201, and No.204 for details.415 SERVO ALARM: nTH AXIS EXCESS SHIFTA speed higher than 524288000 units/s was attempted to be set in the nth axis (axis 18). This error occurs as the result of improperly set CMR.416 SERVO ALARM: nTH AXIS DISCONNECTIONPosition detection system fault in the nth axis (axis 14) pulse coder (disconnection alarm). Refer to diagnosis display No. 200, 201 for details.417 SERVO ALARM: nTH AXIS PARAMETER INCORRECT420 SERVO ALARM: n AXIS SYNC TORQUE421 SERVO ALARM: n AXIS EXCESS ER (D)422 SERVO ALARM: n AXIS423 SERVO ALARM: n AXIS430 n AXIS: SV. MOTOR OVERHEAT431 n AXIS: CNV. OVERLOAD432 n AXIS: CNV. LOW VOLT CONTROL433 n AXIS: CNV. LOW VOLT DC LINK434 n AXIS: INV. LOW VOLT CONTROL435 n AXIS: INV. LOW VOLT DC LINK436 n AXIS: SOFTTHERMAL (OVC)437 n AXIS: CNV. OVERCURRENT POWER438 n AXIS: INV. ABNORMAL CURRENT439 n AXIS: CNV. OVER VOLT DC LINK440 n AXIS: CNV. EX DECELERATION POW.441 n AXIS: ABNORMAL CURRENT OFFSET442 n AXIS: CNV. CHARGE FAILURE443 n AXIS: CNV. COOLING FAN FAILURE444 n AXIS: INV. COOLING FAN FAILURE445 n AXIS: SOFT DISCONNECT ALARM446 n AXIS: HARD DISCONNECT ALARM447 n AXIS: HARD DISCONNECT (EXT)448 n AXIS: UNMATCHED FEEDBACK ALARM449 n AXIS: INV. IPM ALARM453 n AXIS: SPC SOFT DISCONNECT ALARM456 n AXIS: ILLEGAL CURRENT LOOP457 n AXIS: ILLEGAL HI HRV (250US)458 n AXIS: CURRENT LOOP ERROR459 n AXIS: HI HRV SETTING ERROR460 n AXIS: FSSB DISCONNECT461 n AXIS: ILLEGAL AMP INTERFACE462 n AXIS: SEND CNC DATA FAILED463 n AXIS: SEND SLAVE DATA FAILED464 n AXIS: WRITE ID DATA FAILED465 n AXIS: READ ID DATA FAILED466 n AXIS: MOTOR/AMP COMBINATION467 n AXIS: ILLEGAL SETTING OF AXIS468 n AXIS: HI HRV SETTING ERROR (AMP)360 n AXIS: ABNORMAL CHECKSUM (INT)361 n AXIS: ABNORMAL PHASE DATA (INT)362 n AXIS: ABNORMAL REV.DATA (INT)363 n AXIS: ABNORMAL CLOCK (INT)364 n AXIS: SOFT PHASE ALARM (INT)365 n AXIS: BROKEN LED (INT)366 n AXIS: PULSE MISS (INT)367 n AXIS: COUNT MISS (INT)368 n AXIS: SERIAL DATA ERROR (INT)369 n AXIS: DATA TRANS. ERROR (INT)380 n AXIS: BROKEN LED (EXT)381 n AXIS: ABNORMAL PHASE (EXT LIN)382 n AXIS: COUNT MISS (EXT)383 n AXIS: PULSE MISS (EXT)384 n AXIS: SOFT PHASE ALARM (EXT)385 n AXIS: SERIAL DATA ERROR (EXT)386 n AXIS: DATA TRANS. ERROR (EXT)387 n AXIS: ABNORMAL ENCODER (EXT)500 OVER TRAVEL: +n501 OVER TRAVEL: n502 OVER TRAVEL: +n503 OVER TRAVEL: n504 OVER TRAVEL: +n505 OVER TRAVEL: n506 OVER TRAVEL: +n507 OVER TRAVEL: n700 OVERHEAT: CONTROL UNIT701 OVERHEAT: FAN MOTOR704 OVERHEAT: SPINDLE740 RIGID TAP ALARM: EXCESS ERROR741 RIGID TAP ALARM: EXCESS ERROR742 RIGID TAP ALARM: LSI OVER FLOW749 SSPINDLE LSI ERROR750 SPINDLE SERIAL LINK START FAULT751 FIRST SPINDLE ALARM DETECTION (ALXX)752 FIRST SPINDLE MODE CHANGE FAULT754 SPINDLE1 ABNORMAL TORQUE ALM761 SECOND SPINDLE ALARM DETECTION (ALXX)762 SECOND SPINDLE MODE CHANGE FAULT764 SPINDLE2 ABNORMAL TORQUE ALM771 SPINDLE3 ALARM DETECT (ALXX)772 SPINDLE3 MODE CHANGE ERROR774 SPINDLE3 ABNORMAL TORQUE ALM782 SPINDLE4 MODE CHANGE ERROR784 SPINDLE4 ABNORMAL TORQUE ALM4800 ZONE: PUNCHING INHIBITED 14801 ZONE: PUNCHING INHIBITED 24802 ZONE: PUNCHING INHIBITED 34803 ZONE: PUNCHING INHIBITED 44810 ZONE: ENTERING INHIBITED 1 +X4811 ZONE: ENTERING INHIBITED 1 X4812 ZONE: ENTERING INHIBITED 2 +X4813 ZONE: ENTERING INHIBITED 2 X4814 ZONE: ENTERING INHIBITED 3 +X4815 ZONE: ENTERING INHIBITED 3 X4816 ZONE: ENTERING INHIBITED 4 +X4817 ZONE: ENTERING INHIBITED 4 X4830 ZONE: ENTERING INHIBITED 1 +Y4831 ZONE: ENTERING INHIBITED 1 Y4832 ZONE: ENTERING INHIBITED 2 +Y4833 ZONE: ENTERING INHIBITED 2 Y4834 ZONE: ENTERING INHIBITED 3 +Y4835 ZONE: ENTERING INHIBITED 3 Y4836 ZONE: ENTERING INHIBITED 4 +Y4837 ZONE: ENTERING INHIBITED 4 Y4870 AUTO SETTING FEED ERROR4871 AUTO SETTING PIECES ERROR4872 AUTO SETTING COMMAND ERROR900 ROM PARITY910 SRAM PARITY: (BYTE 0)911 SRAM PARITY: (BYTE 1)912 DRAM PARITY: (BYTE 0)913 DRAM PARITY: (BYTE 1)914 DRAM PARITY: (BYTE 2)915 DRAM PARITY: (BYTE 3)916 DRAM PARITY: (BYTE 4)917 DRAM PARITY: (BYTE 5)918 DRAM PARITY: (BYTE 6)919 DRAM PARITY: (BYTE 7)920 SERVO ALARM (14 AXIS)921 SERVO ALARM (58 AXIS)926 FSSB ALARM930 CPU INTERRUPT935 SRAM ECC ERROR950 PMC SYSTEM ALARM951 PMC WATCH DOG ALARM970 NMI OCCURRED IN PMCLSI971 NMI OCCURRED IN SLC972 NMI OCCURRED IN OTHER MODULE973 NON MASK INTERRUPT974 FBUS ERROR975 BUS ERROR976 LBUS ERROR Encountering an issue with your FANUC system? Get expert assistance now, to minimize downtime and keep your operations running smoothly. MRO offers a wide range of repair services and replacement parts. Learn More About Our FANUC Repair Services Shop FANUC Replacement PartsUpdated on February 14, 2024 by Joe Kaminski Joe Kaminski is an industrial automation specialist at MRO Electric. He has a background in industrial engineering and supply chain management. Joe has worked in the automation industry for over 10 years providing support to some of the largest companies in the world. For more info, visit www.mroelectric.com. Skip to content Thanks for visiting our Fanuc Alarm Code and Error page. Here hopefully surmised the various FANUC alarms & codes, error messages and various other fault codes present on FANUC drives, psu, CNC control systems from respective maintenance manuals dating back to 1979. If you require parts then please visit the links above or goto FANUC ALPHA MODULES, FANUC DRIVES, FANUC MOTORS, FANUC CONTROLS, FANUC PSU, etc.Check for alphanumeric characters in the single or twin 7 segment LED displays on the ALPHA modules and LEDs on BETA units.Check for LEDs lit on drive control PCBs or numbers/letters such as AL.11 in the multi section 7 segment LED displays.Alarms for control systems are on the screens or LEDs lite on the PSU, I/P Unit, Master PCB, Main CPU or I/O controller and more.

Fanuc alarm codes list. Fanuc alarm code 5. Fanuc alarm codes. Fanuc 31i alarm list. Fanuc 31 alarm. Fanuc alarm list. Fanuc alarm list pdf.