Continue

G-codes are also called preparatory codes (preparatory functions).Complete CNC G Code ListList of G-codes commonly found on Fanuc and similarly designed CNC controlsG CodeGroupFunctionABCG00G00G0001Positioning (Rapid traverse)G01G01G01Linear interpolation (Cutting feed)G02G02G02Circular interpolation CW or helical interpolation CWG03G03G03Circular interpolation CCW or helical interpolation CCWG04G04G0400DwellG07.1(G107)G07.1(G107)G07.1(G107)G07.1(G107)Cylindrical interpolationG08G08G08Advanced preview controlG10G10G10Programmable data inputG11G11G11Programmable data input cancelG12.1(G112)G12.1(G112)G12.1(G112)21Polar coordinate interpolation modeG13.1(G113)G13.1(G113)G13.1(G113)Polar coordinate interpolation cancel modeG17G17G1716X Y plane selectionG18G18G18Z X plane selectionG19G19G19Y Z plane selectionG20G20G7006Input in inchG21G21G71Input in mmG22G22G2209Stored stroke check function onG23G23G23Stored stroke check function offG25G25G2508Spindle speed fluctuation detection offG26G26G26Spindle speed fluctuation detection onG27G27G2700Reference position return checkG28G28G28Return to reference positionG30G30G302nd, 3rd and 4th reference position returnG31G31G31Skip functionG32G33G3301Thread cuttingG34G34G34Variablelead thread cuttingG36G36G3600Automatic tool compensation XG37G37G37Automatic tool compensation ZG40G40G4007Tool nose radius compensation cancelG41G41G41Tool nose radius compensation leftG42G42G42Tool nose radius compensation rightG50G92G9200Coordinate system setting or max. spindle speed settingG50.3G92.1G92.1Workpiece coordinate system presetG50.2(G250)G50.2(G250)G50.2(G250)20Polygonal turning cancelG51.2(G251)G51.2(G251)G51.2(G251)Polygonal turningG52G52G5200Local coordinate system settingG53G53G53Machine coordinate system settingG54G54G5414Workpiece coordinate system 1 selectionG55G55G55Workpiece coordinate system 2 selectionG56G56G56Workpiece coordinate system 3 selectionG57G57G57Workpiece coordinate system 4 selectionG58G58G58Workpiece coordinate system 5 selectionG59G59G59Workpiece coordinate system 6 selectionG65G65G6500Macro callingG66G66G6612Macro modal callG67G67G67Macro modal call cancelG68G68G6804Mirror image for double turrets ONG69G69G69Mirror image for double turrets OFFG70G70G7200Finishing cycleG71G71G73Stock removal in turningG72G72G74Stock removal in facingG73G73G75Pattern repeatingG74G74G76End face peck drillingG75G75G77Outer diameter/internal diameter drillingG76G76G78Multiple threading cycleG80G80G8010Canned cycle for drilling cancelG83G83G83Cycle for face drillingG84G84G84Cycle for face tappingG86G86G86Cycle for face boringG87G87G87Cycle for side drillingG88G88G88Cycle for side tappingG89G89G89Cycle for side boringG90G77G2001Outer diameter/internal diameter cutting cycleG92G78G21Thread cutting cycleG94G79G24Endface turning cycleG96G96G9602Constant surface speed controlG97G97G97Constant surface speed control cancelG98G94G9405Per minute feedG99G95G95Per revolution feedG90G9003Absolute programmingG91G91Incremental programmingG98G9811Return to initial levelG99G99Return to R point levelG00Positioning (Rapid traverse)CNC G-Code G00 Rapid TraverseCNC G Code Introduction for CNC ProgrammersG01Linear interpolation (Cutting feed)G02Circular interpolation CW or helical interpolation CWG03Circular interpolation CCW or helical interpolation CCWCNC Arc Programming ExerciseCNC Circular Interpolation Tutorial G02 G03G04DwellG04 Dwell Time Calculation Made EasyG10Programmable data inputFanuc G10 G-Code for CNC Machine Programmable Offset SettingG20Input in inchFanuc G20 Measuring in Inches with CNC Program ExampleG21Input in mmFanuc G21 Measuring in Millimeter with CNC Lathe Programming ExampleG32Thread cuttingCNC Fanuc G33 G32 Threading G CodeG32 Taper ThreadingTaper Threading with G32 a CNC Programming ExampleG33 Thread CuttingCNC Fanuc G33 G32 Threading G CodeG70Finishing cycleG71Stock removal in turningG72Stock removal in facingCNC Fanuc G72 Canned Cycle FacingG73Pattern repeatingG74End face peck drillingPeck drilling in Z axis.G74Peck Drilling CycleSimple CNC Lathe Drilling with Fanuc G74 Peck Drilling CycleG74 Face GroovingFace Grooving with G74 G Code a CNC Programming TutorialG75Outer diameter/internal diameter drillingGrooving in X axis.G76Multiple threading cycleThread cutting cycle, G76 threading cycle, Fanuc G76 threading cycle, G76 thread cycle.Fanuc G76 Threading CycleG76 Thread Cycle a CNC Programming ExampleG76 Threading Cycle One Line FormatG76 Threading Cycle One Line Format for Fanuc 10/11/15TG76Taper ThreadingTapered Threading with Fanuc G76G76 Multi Start ThreadingMulti-Start Threading with Fanuc G76G76 Thread InfeedControlling Thread Infeed with Fanuc G76G92Thread cutting cycleCNC Fanuc G92 Threading Cycle, G92 thread cutting cycle.CNC Fanuc G92 Threading CycleG92 Taper Threading CycleG94Facing Cycle (Endface turning cycle)Fanuc G94 Facing Cycle CNC Example ProgramG94 Feed Per Minute / G98 Feed Per MinuteG-Code G94 Feed Per MinuteG95 Feed Per Revolution / G99 Feed Per RevolutionG-Code G95 Feed Per Revolution 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 axis to be synchronously controlled. Any of the following alarms occurred in the operation with the simple synchronization control.;1) The program issued the move command to the slave axis.;2) The program issued the manual continuous feed/manual handle feed/incremental feed command to the slave axis.;3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.;4) The difference between the position error amount of the master and slave axes exceeded the value specified in parameter NO.8313.214 ILLEGAL COMMAND IN SYNCHROMODECoordinate 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)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 INDEPENDENTLYT series;G251 and G250 are not independent blocks.220 ILLEGAL COMMAND IN SYNCHRMODET 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 SYNCHRMODET 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.EDITM series;Input and output are executed at a time in the background edition. Execute a correct operation.224 RETURN TO REFERENCE POINTM 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 0.225 SYNCHRONOUS/MIXED CONTROL ERRORT series (At twopath);This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command.;1 When there is a mistake in axis number parameter (No. 1023) setting.;2 When there is a mistake in control commanded. During hobbing synchronization, a command to bring the Caxis under synchronous, composite, or superimposed control is made. Modify the program or the parameter.226 ILLEGAL COMMAND IN SYNCHRO MODET series (At twopath);A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the parameter.229 CAN NOT KEEP SYNCHROSTATET series;This alarm is generated in the following circumstances.;1 When the synchro/mixed state could not be kept due to system overload.;2 The above condition occurred in CMC devices (hardware) and synchro state could not be kept.;(This alarm is not generated in normal use conditions.)230 R CODE NOT FOUNDThe 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 L50Any of the following errors occurred in the specified format at the programmable parameter input.;1 Address N or R was not entered.;2 A number not specified for a parameter was entered.;3 The axis number was too large.;4 An axis number was not specified in the axistype parameter.;5 An axis number was specified in the parameter which is not an axis type. Correct the program.;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.233 DEVICE BUSYWhen an attempt was made to use a unit such as that connected via the RS232C interface, other users were using it.239 BP/S ALARMWhile punching was being performed with the function for controlling external I/O units,background editing was performed.240 BP/S ALARMBackground editing was performed during MDI operation244 P/S ALARMT 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 TCODE NOT ALOWEE IN THIS BLOCKT 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.251 ATC ERRORThis alarm is issued in the following cases; An M06T_ command contains an unusable T code.; An M06 command has been specified when the Z machine coordi nate is positive.; The parameter for the current tool number (No. 7810) is set to 0.; An M06 command has been specified in canned cycle mode.; A reference position return command (G27 to G44) and M06 command have been specified in the same block.; An M06 command has been specified in tool compensation mode (G41 to G44).; An M06 command has been specified without performing reference position return after poweron or the release of emergency stop.; The machine lock signal or Zaxis ignore signal has been turned on during tool exchange.; A pry alarm has been detected during tool exchange. Refer to diagnosis No. 530 to determine the cause. (Only for ROBODRILL)252 ATC SPINDLE ALARMAn excessive error arose during spindle positioning for ATC. For details, refer to diagnosis No. 531. (Only for ROBODRILL)253 G05 IS NOT AVAILABLEM series;Alarm details Binary input operation using highspeed 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 INHIBITEDA repositioning command was specified in the circular interpolation (G02, G03) mode.4502 ILLEGAL COMMAND IN BOLT HOLEIn 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 ANGLEIn 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 ARCIn 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.4505 ILLEGAL COMMAND I

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 ERRORA 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 ERRORThe 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 ERRORThe 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 ERRORThe 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 ERRORNth 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. CNC machines work by following the commands or instructions (G-codes / M-codes) which are given in Part Program. Here are G-codes for Fanuc cnc control which are necessary for a cnc machinists to learn to understand cnc programming.ContentsFanuc G-Code List (Lathe)Fanuc G-Code List (Mill)Fanuc G-Code List (Lathe)G codeDescriptionG00Rapid traverseG01Linear interpolationG02Circular interpolation CWG03Circular interpolation CCWG04DwellG09Exact stopG10Programmable data inputG20Input in inchG21Input in mmG22Stored stroke check function onG23Stored stroke check function offG27Reference position return checkG28Return to reference positionG32Thread cuttingG40Tool nose radius compensation cancelG41Tool nose radius compensation leftG42Tool nose radius compensation rightG70Finish machining cycleG71Turning cycleG72Facing cycleG73Pattern repeating cycleG74Peck drilling cycleG75Grooving cycleG76Threading cycleG92Coordinate system setting or max. spindle speed settingG94Feed Per MinuteG95Feed Per RevolutionG96Constant surface speed controlG97Constant surface speed control cancelFanuc G-Code List (Mill)G codeDescriptionG00Rapid traverseG01Linear interpolationG02Circular interpolation CWG03Circular interpolation CCWG04DwellG17X Y plane selectionG18Z X plane selectionG19Y Z plane selectionG28Return to reference positionG302nd, 3rd and 4th reference position returnG40Cutter compensation cancelG41Cutter compensation leftG42Cutter compensation rightG43Tool length compensation + directionG44Tool length compensation directionG49Tool length compensation cancelG53Machine coordinate system selectionG54Workpiece coordinate system 1 selectionG55Workpiece coordinate system 2 selectionG56Workpiece coordinate system 3 selectionG57Workpiece coordinate system 4 selectionG58Workpiece coordinate system 5 selectionG59Workpiece coordinate system 6 selectionG68Coordinate rotationG69Coordinate rotation cancelG73Peck drilling cycleG74Left-spiral cutting circleG76Fine boring cycleG80Canned cycle cancelG81Drilling cycle, spot boring cycleG82Drilling cycle or counter boring cycleG83Peck drilling cycleG84Tapping cycleG85Boring cycleG86Boring cycleG87Back boring cycleG88Boring cycleG89Boring cycleG90Absolute commandG91Increment commandG92Setting for work coordinate system or clamp at maximum spindle speedG98Return to initial point in canned cycleG99Return to R point in canned cycle A generic name for a plain-text language that CNC machines are able to understand, G-Codes are important to understand in the manufacturing, automation and engineering spaces. You can enter a G-Code manually if you wish, but you do not have to because of the CAD/CAM software abilities along with the machine controller. G-Codes are not necessarily readable by humans, but its possible to look through the file and determine what is generally occurring.What Is G Code?In the factory automation space, nobody likes downtime and receiving error codes. While using CNCs (view FANUC CNC parts here), many professionals are faced with G Codes. By definition, a G Code is a computer code language that is used to guide CNC machine devices to perform specific motions.What Do CNC G Codes Do?G Codes are important because they allow easy, repeatable control of the motion of a CNC machine. A few examples of specific motions that CNC G Codes can control, would be:canned cycleswork coordinatesseveral repetitive cycles.Canned CyclesAlso referred to as a fixed cycle, canned cycles are ways to effectively and efficiently perform repetitive CNC machining operations. They automate specific machining functions. Afew examples would be pocketing, threading, and drilling. Acannedcycle is almostalways stored as apre-program in amachines controller. To learn more about canned cycles, check out this article courtesy of zero-divide.net.Work CoordinatesThe G Code coordinate pipeline goes something like this:Unit conversion to metricConvert from relative to absolute and polar to Cartesian: g90g91XYZ()G52, G54, and G92 offsetsG51 scalingG68 coordinate rotationG-Code is the most popular programming language used for programming CNC machinery. Some G words alter the state of the machine so that it changes from cutting straight lines to cutting arcs. Other G words cause the interpretation of numbers as millimeters rather than inches. Some G words set or remove tool length or diameter offsets. Be sure to check out our article covering FANUC CNC Codes, including FANUC M Codes, here.MRO Electric and Supplyhas new and refurbishedFANUC CNCparts available. We also offer repair pricing. For more information, please call800-691-8511or emailsales@mroelectric.com.What Are the Different Types of G Code Commands?Listed below are some easily-understood G-code commands in which are used for setting the speed, feed, and tool parameters.F= FeedThe F value in G-code is used to set the feed rate, which determines the speed at which the machines extruder or tool head moves. F values are typically measured in millimeters per minute (mm/min), so dividing the F value by 60 converts it to millimeters per second (mm/s). For instance, an F value of F1500 means the feed rate is 25 mm/s. The machine operates at this specified feed rate when a G1 command is used, which is essential for precise control of the movement speed.It is crucial to set the feed rate (F) before the first G1 command to avoid errors. Heres an example of setting the feed rate: In this example, the machine will move to the coordinates X100 Y100 at a speed of 1500 mm/min (25 mm/s).S= Spindle SpeedThe S commands purpose is to set the spindle speed. The Spindle speed is almost always set in RPMs (revolutions per minute). Here is an example: T= ToolThe T commands purpose is paired with M6 in order to display the tool number to be used for cutting the current file. Here is an example: Common G Code Command ListBelow is a comprehensive list of common CNC G Codes, designed to guide you through the essential programming for CNC machines.G00 Rapid traverseG01 Linear interpolation with feed rateG02 Circular interpolation (clockwise)G03 Circular interpolation (counterclockwise)G2/G3 Helical interpolationG04 Dwell time in millisecondsG05 Spline definitionG06 Spline interpolationG07 Tangential circular interpolation, Helix interpolation, Polygon interpolation, Feedrate interpolationG08 Ramping function at block transition / Look ahead offG09 No ramping function at block transition / Look ahead onG10 Stop dynamic block preprocessingG11 Stop interpolation during block preprocessingG12 Circular interpolation (CW) with radiusG13 Circular interpolation (CCW) with radiusG14 Polar coordinate programming, absoluteG15 Polar coordinate programming, relativeG16 Definition of the pole point of the polar coordinate systemG17 Selection of the X, Y planeG18 Selection of the Z, X planeG19 Selection of the Y, Z planeG20 Selection of a freely definable planeG21 Parallel axes onG22 Parallel axes offG24 Safe zone programming; lower limit valuesG25 Safe zone programming; upper limit valuesG26 Safe zone programming offG27 Safe zone programming onG33 Thread cutting with constant pitchG34 Thread cutting with dynamic pitchG35 Oscillation configurationG38 Mirror imaging onG39 Mirror imaging offG40 Path compensations offG41 Path compensation left of the workpiece contourG42 Path compensation right of the workpiece contourG43 Path compensation left of the workpiece contour with altered approachG44 Path compensation right of the workpiece contour with altered approachG50 ScalingG51 Part rotation; programming in degreesG52 Part rotation; programming in radiansG53 Zero offset offG54 Zero offset #1G55 Zero offset #2G56 Zero offset #3G57 Zero offset #4G58 Zero offset #5G59 Zero offset #6G63 Feed/spindle override not activeG66 Feed/spindle override activeG70 Inch format activeG71 Metric format activeG72 Interpolation with precision stop offG73 Interpolation with precision stop onG74 Move to home positionG75 Curvature function activationG76 Curvature acceleration limitG78 Normalcy function on (rotational axis orientation)G79 Normalcy function offMilling ApplicationsG80 Canned cycle offG81 Drilling to final depth canned cycleG82 Spot facing with dwell time canned cycleG83 Deep hole drilling canned cycleG84 Tapping or Thread cutting with balanced chuck canned cycleG85 Reaming canned cycleG86 Boring canned cycleG87 Reaming with measuring stop canned cycleG88 Boring with spindle stop canned cycleG89 Boring with intermediate stop canned cycleCylindrical Grinding ApplicationsG81 Reciprocation without plungeG82 Incremental face grindingG83 Incremental plunge grindingG84 Multi-pass face grindingG85 Multi-pass diameter grindingG86 Shoulder grindingG87 Shoulder grinding with face plungeG88 Shoulder grinding with diameter plungeG90 Absolute programmingG91 Incremental programmingG92 Position presetG93 Constant tool circumference velocity on (grinding wheel)G94 Feed in mm / min (or inch / min)G95 Feed per revolution (mm / rev or inch / rev)G96 Constant cutting speed onG97 Constant cutting speed offG98 Positioning axis signal to PLCG99 Axis offsetG100 Polar transformation offG101 Polar transformation onG102 Cylinder barrel transformation on; cartesian coordinate systemG103 Cylinder barrel transformation on, with real-time-radius compensation (RRC)G104 Cylinder barrel transformation with centerline migration (CLM) and RRCG105 Polar transformation on with polar axis selectionsG106 Cylinder barrel transformation on polar-/cylinder-coordinatesG107 Cylinder barrel transformation on polar-/cylinder-coordinates with RRCG108 Cylinder barrel transformation polar-/cylinder-coordinates with CLM and RRCG109 Axis transformation programming of the tool depthG110 Power control axis selection/channel 1G111 Power control pre-selection V1, F1, T1/channel 1 (Voltage, Frequency, Time)G112 Power control pre-selection V2, F2, T2/channel 1G113 Power control pre-selection V3, F3, T3/channel 1G114 Power control pre-selection T4/channel 1G115 Power control pre-selection T5/channel 1G116 Power control pre-selection T6/pulsing outputG117 Power control pre-selection T7/pulsing outputG120 Axis transformation; orientation changing of the linear interpolation rotary axisG121 Axis transformation; orientation change in a planeG125 Electronic gearbox; plain teethG126 Electronic gearbox; helical gearing, axialG127 Electronic gearbox; helical gearing, tangentialG128 Electronic gearbox; helical gearing, diagonalG130 Axis transformation; programming of the type of the orientation changeG131 Axis transformation; programming of the type of the orientation changeG132 Axis transformation; programming of the type of the orientation changeG133 Zero lag thread cutting onG134 Zero lag thread cutting offG140 Axis transformation; orientation designation workpiece fixedcoordinatesG141 Axis transformation; orientation designation active coordinates G160 ART activationG161 ART learning function for velocity factors onG162 ART learning function deactivationG163 ART learning function for acceleration factorsG164 ART learning function for acceleration changingG165 Command filter onG166 Command filter offG170 Digital measuring signals; block transfer with hard stopG171 Digital measuring signals; block transfer without hard stopG172 Digital measuring signals; block transfer with smooth stopG175 SERCOS-identification number writeG176 SERCOS-identification number readG180 Axis transformation offG181 Axis transformation on with not rotated coordinate systemG182 Axis transformation on with rotated/displaced coordinate systemG183 Axis transformation; definition of the coordinate systemG184 Axis transformation; programming tool dimensionsG186 Look ahead; corner acceleration; circle toleranceG188 Activation of the positioning axesG190 Diameter programming deactivationG191 Diameter programming on and display of the contact pointG192 Diameter programming; only display contact point diameterG193 Diameter programming; only display contact point actual axes centerpointG200 Corner smoothing offG201 Corner smoothing on with defined radiusG202 Corner smoothing on with defined corner toleranceG203 Corner smoothing with defined radius up to maximum toleranceG210 Power control axis selection/Channel 2G211 Power control pre-selection V1, F1, T1/Channel 2G212 Power control pre-selection V2, F2, T2/Channel 2G213 Power control pre-selection V3, F3, T3/Channel 2G214 Power control pre-selection T4/Channel 2G215 Power control pre-selection T5/Channel 2G216 Power control pre-selection T6/pulsing output/Channel 2G217 Power control pre-selection T7/pulsing output/Channel 2G220 Angled wheel transformation offG221 Angled wheel transformation onG222 Angled wheel transformation on but angled wheel moves before othersG223 Angled wheel transformation on but angled wheel moves after othersG265 Distance regulation axis selectionG270 Turning finishing cycleG271 Stock removal in turningG272 Stock removal in facingG274 Peck finishing cycleG275 Outer diameter / internal diameter turning cycleG276 Multiple pass threading cycleG310 Power control axes selection /channel 3G311 Power control pre-selection V1, F1, T1/channel 3G312 Power control pre-selection V2, F2, T2/channel 3G313 Power control pre-selection V3, F3, T3/channel 3G314 Power control pre-selection T4/channel 3G315 Power control pre-selection T5/channel 3G316 Power control pre-selection T6/pulsing output/Channel 3G317 Power control pre-selection T7/pulsing output/Channel 3In conclusion, becoming well-versed on CNC G-Codes, along with other codes associated with CNCs is imperative in this day and age. By having up-to-speed knowledge of CNC codes, you could most definitely set yourself apart from the average Joe.Updated on June 24, 2024 by Leanna Cota 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. Below you will find a list of common codes many builders use. Please remember, all machines may be configured differently and the list below may not match your machine perfectly. If fact, virtually all M-Codes above M79 will vary from builder to builder. Please make sure to contact your machine tool builder to be sure.(G Codes are written by Fanuc)G00 Moves the tool in rapid travel (not necessarily a straight line)G01 Moves the tool using a set feedrateG02 Moves the tool along a clockwise arc pathG03 Moves the tool along a counter-clockwise pathG04 Sets a dwell time in seconds or revolutions of the spindleG10 Data settingG11 Data setting mode cancelG17 Establishes axis movement in the X and Y axis planesG18 Establishes axis movement in the X and Z axis planesG19 Establishes axis movement in the Y and Z axis planesG20 Values are in InchesG21 Values are in millimetersG28 Return to reference positionG30 Second reference positionG33 Thread cuttingG40 Cancel cutter compensationG41 Cutter compensation leftG42 Cutter compensation rightG43 Tool length compensation positiveG44 Tool length compensation negativeG49 Tool length compensation cancelG53 Machine Coordinate moveG54 Use workshift offset #1G55 Use workshift offset #2G56 Use workshift offset #3G57 Use workshift offset #4G58 Use workshift offset #5G59 Use workshift offset #6G60 Single direction positioningG65 Macro callG66 Macro modal callG67 Macro modal call cancelG73 Peck drilling cycleG76 Fine boring cycleG80 Canned cycle cancelG81 Drilling cycle or spot boring cycleG82 Drilling cycle or counter boring cycleG83 Peck drilling cycleG84 Tapping cycleG85 Boring cycleG86 Boring cycleG87 Back boring cycleG88 Boring cycleG89 Boring cycleG90 Absolute measurementsG91 Incremental measurementsG94 Feed per minuteG95 Feed per revolution of the spindleG96 Constant surface speed controlG97 Constant surface speed control cancelG98 Return to initial point in canned cycleG99 Return to R point is canned cycleM00 Program stopM01 Optional stopM02 End of programM03 Spindle on ClockwiseM04 Spindle on Counter-clockwiseM05 Spindle stopM06 Tool changeM08 Coolant onM09 Coolant offM10 ClampM11 UnclampM30 End of program and rewind to beginning of programM98 Call subprogramM99 End subprogramM06 T12; Performs a tool change to tool number 12S1000 M03; Turns spindle on clockwise to 1000 rpmG01 X10.5 F10.0: Moves the X axis to position 10.5 at a feedrate of 10.0G00 X. Y.. Z.. ;G00 Move in rapid travelX.. X axis addressY.. Y axis addressZ.. Z axis addressG01 X. Y.. Z.. F. ;G01 Move in a straight lineX.. X axis addressY.. Y axis addressZ.. Z axis addressF.. FeedrateG02 X. Y.. Z.. I. J.. K.. F. ;G02 Move along a clockwise circular pathX.. X axis addressY.. Y axis addressZ.. Z axis addressI .. I axis addressJ.. J axis addressK.. K axis addressF.. FeedrateG03 X. Y.. Z.. I. J.. K.. F. ;G03 Move along a counter-clockwise circular pathX.. X axis addressY.. Y axis addressZ.. Z axis addressI .. I axis addressJ.. J axis addressK.. K axis addressF.. FeedrateG04 X. ;G04 Pause machine operationX. (Specify a time/spindle speed with decimal point)G04 P. ; (Specify a time without decimal point)G04 Pause machine operationP. (Specify a time/spindle speed without decimal point)G28 G90 X10.0 Y3.0 ;G28 Return to reference pointG90 Absolute positioningX10.0 X axis locationY3.0 Y axis locationThis command sequence can be used to move the tool from point A to anABSOLUTE Coordinate first, then move the tool to the Reference Point Zero.G28 G91 X-4.0 Y-3.0 ;G28 Return to reference pointG91 Incremental positioningX-4.0 X axis locationY-3.0 Y axis locationThis command sequence can be used to make an incremental move from point Athen move the tool to the Reference Point ZeroG28 G91 X0.0 Y0.0 ;This line will take the tool back to the reference position direct from the currentlocation.G41 D. X ;G41 Cutter Comp LeftD.. Assigns Radius Offset numberX. X axis movementG42 D. X ;G42 Cutter Comp RightD.. Assigns Radius Offset numberX. X axis movementG40 X.. ;G40 Cancel Cutter CompX.. Cancels comp on way to HereG43 H. Z ;G43 Add offset amountH.. Offset numberZ. Z axis movementG44 H Z ;G44 Subtract offset amountH.. Offset numberZ. Z axis movementG49 H.. Z ;G49 Cancel OffsetH13 Assigns Offset Number 13Z0.0 Z axis movement to ZeroG65 P. L.. ;G65 Macro call (Modal)P.. Macro program NumberL.. Number of repetitionsG66 P. ;G66 Macro call (Non-Modal)P.. Macro program NumberG73 X. Y.. Z.. R. Q.. F.. K. ;G73 High speed peck drilling cycleX.. X axis addressY.. Y axis addressZ.. Z address location of bottom of holeR .. Retract planeQ.. Distance for each peck strokeF.. FeedrateK.. Number of repeatsG74 X. Y.. Z.. R. P.. F.. K. ;G74 Left handed tapping cycleX.. X axis addressY.. Y axis addressZ.. Z address location of bottom of holeR . Retract planeF.. FeedrateK.. Number of repeatsG76 X. Y.. Z.. R. Q.. P.. F.. K. ;G76 Fine boring cycleX.. X axis addressY.. Y axis addressZ.. Z address location of bottom of holeR .. Retract planeQ.. Distance for each peck strokeP.. Dwell time at bottom of boreF.. FeedrateK.. Number of repeatsG81 X. Y.. Z.. R. F.. K. ;G81 Spot drilling cycleX.. X axis addressY.. Y axis addressZ.. Z address location of bottom of holeR .. Retract planeF.. FeedrateK.. Number of repeatsG82 X. Y.. Z.. R. P.. F.. K. ;G82 Drilling cycle counter boring cycleX.. X axis addressY.. Y axis addressZ.. Z address location of bottom of holeR .. Retract planeP.. Dwell time at bottom of boreF.. FeedrateK.. Number of repeatsG83 X. Y.. Z.. R. Q.. F.. K. ;G83 Peck drilling cycleX.. X axis addressY.. Y axis addressZ.. Z address location of bottom of holeR .. Retract planeQ.. Distance for each peck strokeF.. FeedrateK.. Number of repeatsTIE offers same day shipping on more than 100,000 Fanuc CNC parts in stock. We offer a 1 year in service warranty on all parts and repairs. Shop Parts MS-Techs G-Codes are Fanuc G-code compatible and thus there is no need to have a special post. Our control includes built-in G-Code lists with every purchase that not only indicates format and structure of said G-Codes, but also function, description, and usage only a mouse-click away. M-code are cnc program instructions which help cnc machinist/programmer to control cnc machine hardware like chuck, tailstock, quill, coolant. Here are listed M-code which are mostly used on cnc lathe/mill with Fanuc cnc control.ContentsFanuc M-Code List (Lathe)Fanuc M-Code List (Mill)Fanuc M-Code List (Lathe)M codeDescriptionM00Program stopM01Optional program stopM02End of programM03Spindle start forward CWM04Spindle start reverse CCWM05Spindle stopM08Coolant onM09Coolant offM29Rigid tap modeM30End of program resetM40Spindle gear at middleM41Low Gear SelectM42High Gear SelectM68Hydraulic chuck closeM69Hydraulic chuck openM78Tailstock advancingM79Tailstock reversingM94Mirrorimage cancelM95Mirrorimage of X axisM98Subprogram callM99End of subprogramFanuc M-Code List (Mill)M codeDescriptionM00Program stopM01Optional program stopM02End of programM03Spindle start forward CWM04Spindle start reverse CCWM05Spindle stopM06Tool changeM07Coolant ON Mist coolant/Coolant thru spindleM08Coolant ON Flood coolantM09Coolant offM29Rigid tap modeM30End of program resetM40Spindle gear at middleM41Low Gear SelectM42High Gear SelectM68Hydraulic chuck closeM69Hydraulic chuck openM78Tailstock advancingM79Tailstock reversingM94Mirrorimage cancelM95Mirrorimage of X axisM98Subprogram callM99End of subprogramFanuc M-Code List (Mill)M codeDescriptionM00Program stopM01Optional program stopM02End of programM03Spindle start forward CWM04Spindle start reverse CCWM05Spindle stopM06Tool changeM07Coolant ON Mist coolant/Coolant thru spindleM08Coolant ON Flood coolantM09Coolant offM29Rigid tap modeM30End of program resetM40Spindle gear at middleM41Low Gear SelectM42High Gear SelectM68Hydraulic chuck closeM69Hydraulic chuck openM78Tailstock advancingM79Tailstock reversingM94Mirrorimage cancelM95Mirrorimage of X axisM98Subprogram callM99End of subprogram Complete CNC g code list with m-codes for milling in the Fanuc, LinuxCNC, GRBL,and Haas CNC machines dialects. We give a quick definition for all g codes along with a link to tutorials and examples of how to use it in CNC programming. Click the g-code link (if any) to visit a tutorial from our popuplar Online Free G-Code Training that shows examples and teaches how to program CNC machines using the g-code commands. This page is your ideal g-code reference.G Codes and CNC G-Code List CodeCategoryFunctionNotes G00Motiong00 rapid: Move in a straight line at rapids speed.XYZ of endpoint G01MotionMove in a straight line at last speed commanded by a (F)eedrateXYZ of endpoint G02MotionCircular Interpolation: Clockwise circular arc at (F)eedrateXYZ of endpoint IJK relative to center R for radius G03MotionCircular Interpolation: Counter-clockwise circular arc at (F)eedrateXYZ of endpoint IJK relative to center R for radius G04MotionDwell: Stop for a specified time.P for milliseconds X for seconds G05MotionFADAL Non-Modal Rapids G09MotionExact stop check G10CompensationProgrammable parameter input G15CoordinateTurn Polar Coordinates OFF, return to Cartesian Coordinates G16CoordinateTurn Polar Coordinates ON G17CoordinateSelect X-Y plane (Plane Selection) G18CoordinateSelect X-Z plane (Plane Selection) G19CoordinateSelect Y-Z plane (Plane Selection) G20CoordinateProgram coordinates are inches G21CoordinateProgram coordinates are mm G27MotionReference point return check G28MotionReturn to home position G29MotionReturn from the reference position G30MotionReturn to the 2nd, 3rd, and 4th reference point G32CannedConstant lead threading (like G01 synchronized with spindle) G40CompensationTool cutter compensation off (radius comp.) G41CompensationTool cutter compensation left (radius comp.) G42CompensationTool cutter compensation right (radius comp.) G43CompensationApply tool length compensation (plus) G44CompensationApply tool length compensation (minus) G49CompensationTool length compensation cancel G50CompensationReset all scale factors to 1.0 G51CompensationTurn on scale factors G52CoordinateLocal workshift for all coordinate systems: add XYZ offsets G53CoordinateMachine coordinate system (cancel work offsets) G54CoordinateWork coordinate system (1st Workpiece) G55CoordinateWork coordinate system (2nd Workpiece) G56CoordinateWork coordinate system (3rd Workpiece) G57CoordinateWork coordinate system (4th Workpiece) G58CoordinateWork coordinate system (5th Workpiece) G59CoordinateWork coordinate system (6th Workpiece) G61OtherExact stop check mode G62OtherAutomatic corner override G63OtherTapping mode G64OtherBest speed path G65OtherCustom macro simple call G68CoordinateCoordinate System Rotation G69CoordinateCancel Coordinate System Rotation G73CannedHigh speed drilling cycle (small retract) G74CannedLeft hand tapping cycle G76CannedFine boring cyle G80CannedCancel canned cycle G81CannedSimple drilling cycle G82CannedDrilling cycle with dwell (counterboring) G83CannedPeck drilling cycle (full retract) G84CannedTapping cycle G85CannedBoring cycle, no dwell, feed out G86CannedBoring cycle, spindle stop, rapid out G87CannedBack boring canned cycle G88CannedBoring canned cycle, spindle stop, manual out G89CannedBoring canned cycle, dwell, feed out G90CoordinateAbsolute programming of XYZ (type B and C systems) G90.1CoordinateAbsolute programming IJK (type B and C systems) G91CoordinateInc

series)5436PARAMETER SETTING ERROR 1 (TLAC) (M series)5437PARAMETER SETTING ERROR 2 (TLAC) (M series)5440ILLEGAL DRILLING AXIS SELECTED (M series)5445CRC:MOTION IN G39 (M series)5446CRC:NO AVOIDANCE (M series)5447CRC:DANGEROUS AVOIDANCE (M series)5448CRC:INTERFERENCE TO AVD. (M series)5452IMPROPER GCODE (5AXIS MODE) (M series)5453NOTE: G68 IS CANCELED (HPCC) (M series)5455ILLEGAL ACC. PARAMETER (M series)4800ZONE: PUNCHING INHIBITED 14801ZONE: PUNCHING INHIBITED 24802ZONE: PUNCHING INHIBITED 34803ZONE: PUNCHING INHIBITED 44804ZONE: ENTERING INHIBITED 1 +X4805ZONE: ENTERING INHIBITED 1X4806ZONE: ENTERING INHIBITED 2 +X4807ZONE: ENTERING INHIBITED 2X4808ZONE: ENTERING INHIBITED 3 +X4809ZONE: ENTERING INHIBITED 4 +X4810ZONE: ENTERING INHIBITED 4X4811ZONE: ENTERING INHIBITED 1 +Y4812ZONE: ENTERING INHIBITED 1Y4813ZONE: ENTERING INHIBITED 2 +Y4814ZONE: ENTERING INHIBITED 2Y4815ZONE: ENTERING INHIBITED 3 +Y4816ZONE: ENTERING INHIBITED 3Y4817ZONE: ENTERING INHIBITED 4 +Y4818ZONE: ENTERING INHIBITED 4Y4870AUTO SETTING FEED ERROR4871AUTO SETTING PIECES ERROR4872AUTO SETTING COMMANDERROR Bonus: Check Out our Other CNC Cookbooks for More In-Depth CNC Information!If you're a CNC Beginnner, check out our CNC Beginner's Cookbook. It'll get you up to speed with a solid CNC foundation fast.We also have Cookbooks for Feeds and Speeds, G-Code Programming, CNC Manufacturing and Shop Management, DIY CNC, and don't forget the CNC Cookbook Blog-with over 4 million visitors a year it's the most popular CNC blog by far on the web.More ResourcesMazatrol Training ClassesFanuc CNC Training Classes Join our newsletter to get updates on what's next at CNC Cookbook. Mill G Code GuideTurning G CodesFanuc Alarm CodesGCode SimulatorThese are the common g-codes for CNC Lathes and turning. Code categories are the groupings for the g-code Wizard (type Ctrl+G for the Wizard). Function tells what the g-code does, Notes gives a little more information such as the parameters, and Tutorial is a link (if any) to a g-code tutorial that uses G-Wizard Editor to teach how to use the g-code. For specifics on CNC Lathe Programming, see our intro article.CodeCategoryFunctionNotesTutorialsG00MotionMove in a straight line at rapids speed.XYZ of endpointG00 and MDI.Linear Motion: G00 and G01G01MotionMove in a straight line at last speed commanded by a (F)eedrateXYZ of endpointG01 and MDI.Linear Motion: G00 and G01G02MotionClockwise circular arc at (F)eedrateXYZ of endpointIJK relative to centerR for radiusCircular Arcs: G02 and G03G03MotionCounter-clockwise circular arc at (F)eedrateXYZ of endpointIJK relative to centerR for radiusCircular Arcs: G02 and G03G04MotionDwell: Stop for a specified time.P for millisecondsX for secondsPrecise Timing and Speed: Dwell, Exact Stop, Backlash CompensationG09MotionExact stop checkPrecise Timing and Speed: Dwell, Exact Stop, Backlash CompensationG10CompensationProgrammable parameter inputG17CoordinateSelect X-Y planeSelect X-Y planeG18CoordinateSelect X-Z planeSelect X-Z planeG19CoordinateSelect Y-Z planeSelect Y-Z planeG20CoordinateProgram coordinates are inchesG20 and G21: Unit ConversionG21CoordinateProgram coordinates are mmG20 and G21: Unit ConversionG27MotionReference point return checkG28: Return to Reference PointG28MotionReturn to home positionG28: Return to Reference PointG29MotionReturn from the reference positionG28: Return to Reference PointG30MotionReturn to the 2nd, 3rd, and 4th reference pointG28: Return to Reference PointG32CannedConstant lead threading (like G01 synchronized with spindle)G40CompensationTool cutter compensation off (radius comp.)G41CompensationTool cutter compensation left (radius comp.)G42CompensationTool cutter compensation right (radius comp.)G43CompensationApply tool length compensation (plus)G44CompensationApply tool length compensation (minus)G49CompensationTool length compensation cancelG50CompensationReset all scale factors to 1.0G51CompensationTurn on scale factorsG52CoordinateLocal workshift for all coordinate systems: add XYZ offsetsG53CoordinateMachine coordinate system (cancel work offsets)G54CoordinateWork coordinate system (1st Workpiece)G55CoordinateWork coordinate system (2nd Workpiece)G56CoordinateWork coordinate system (3rd Workpiece)G57CoordinateWork coordinate system (4th Workpiece)G58CoordinateWork coordinate system (5th Workpiece)G59CoordinateWork coordinate system (6th Workpiece)G61OtherExact stop check modePrecise Timing and Speed: Dwell, Exact Stop, Backlash CompensationG62OtherAutomatic corner overrideG63OtherTapping modeG64OtherBest speed pathG65OtherCustom macro simple callSubprograms and MacrosG70CannedFinish Turning CycleG71CannedRough Turning CycleG71: Rough Turning CycleG71 Type II: Rough Turning With "Pockets"G72CannedRough Facing CycleG73CannedPattern Repeating CycleG74CannedPeck Drilling CycleG75CannedGrooving CycleG76CannedThreading CycleG76 Lathe Threading CycleG80CannedCancel canned cycleG83CannedFace drilling cycleG84CannedFace Tapping cycleG86CannedBoring canned cycle, spindle stop, rapid outG87CannedSide Drilling CycleG88CannedSide Tapping CycleG89CannedSide Boring CycleG90CoordinateAbsolute programming of XYZ (type B and C systems)G90 and G91 Absolute vs Incremental ModeG90.1CoordinateAbsolute programming IJK (type B and C systems)G91CoordinateIncremental programming of XYZ (type B and C systems)G90 and G91 Absolute vs Incremental ModeG91.1CoordinateIncremental programming IJK (type B and C systems)G92CoordinateThread Cutting CycleG92 (alternate)MotionClamp of maximum spindle speedSG94MotionEndface Turning CycleG96MotionConstant Surface Speed ONG96: Constant Surface SpeedG97MotionConstant Surface Speed CancelG96: Constant Surface SpeedG98MotionFeedrate per MinuteG98 G-Code and G99 G-Code: Canned Cycle Return or Feedrate ModesG99MotionFeedrate per RevolutionG98 G-Code and G99 G-Code: Canned Cycle Return or Feedrate ModesG190MotionRadius modeCNC Lathe ProgrammingG191MotionDiameter modeCNC Lathe ProgrammingM-Codes CodeCategoryFunctionNotesTutorials M00M-CodeProgram Stop (non-optional) M01M-CodeOptional Stop: Operator Selected to Enable M02M-CodeEnd of Program M03M-CodeSpindle ON (CW Rotation) M03 Spindle On Clockwise M04M-CodeSpindle ON (CCW Rotation) M04 Spindle on Counter Clockwise M05M-CodeSpindle Stop M05 Spindle Off. M06M-CodeTool Change M07M-CodeMist Coolant ON M07 and MDI. M08M-CodeFlood Coolant ON M08 and MDI. M09M-CodeCoolant OFF M09 and MDI. M13M-CodeSpindle ON (CW Rotation) + Coolant ON M13 and MDI. M14M-CodeSpindle ON (CCW Rotation) + Coolant ON M14 and MDI. M30M-CodeEnd of Program, Rewind and Reset Modes M97M-CodeHaas-Style Subprogram Call Subprograms and Macros M98M-CodeSubprogram Call Subprograms and Macros M99M-CodeReturn from Subprogram Subprograms and Macros Bonus: Check Out our Other CNC Cookbooks for More In-Depth CNC Information!If you're a CNC Beginnner, check out our CNC Beginner's Cookbook. It'll get you up to speed with a solid CNC foundation fast.We also have Cookbooks for Feeds and Speeds, G-Code Programming, CNC Manufacturing and Shop Management, DIY CNC, and don't forget the CNC Cookbook Blog-with over 4 million visitors a year it's the most popular CNC blog by far on the web.More ResourcesG75 and Peck Parting Off For LathesMazatrol Training ClassesFanuc CNC Training ClassesTurning Feeds and Speeds Calculator Join our newsletter to get updates on what's next at CNC Cookbook. Whether youre a CNC machinist, programmer, or manufacturer looking to optimize production, understanding M codes and G codes is non-negotiable. These two types of commands form the foundation of CNC programming, enabling precise control of machine operations like cutting, tool changes, coolant flow, and motion paths.This article provides a comprehensive guide to common G and M codes, how they function, and where they're usedespecially within FANUC-controlled CNC systems.What Are G Codes and M Codes?G Codes (Geometric Codes) define machine movement and tool paths.M Codes (Miscellaneous Codes) handle machine functions such as turning on coolant, stopping the spindle, or tool changing.Think of it this way:G codes = How the machine moves.M codes = What the machine does while moving (or before/after).Both are essential for a successful CNC program.G Code List Common Geometric CommandsG CodeFunctionDescriptionG00Rapid positioningMoves tool quickly from point to point without cuttingG01Linear interpolationMoves tool in a straight line at a set feed rateG02Circular interpolation (clockwise)Cuts arcs or circles in a clockwise directionG03Circular interpolation (counter-clockwise)Cuts arcs or circles counter-clockwiseG04DwellPauses the machine for a set timeG17Select XY planeDefault plane for 2D machiningG18Select ZX planeUseful in turning centersG19Select YZ planeFor vertical or sidewall machiningG20Inch unitsSets machine units to inchesG21Metric unitsSets machine units to millimetersG28Return to machine zeroSends tool to home positionG40Cancel cutter compensationTurns off tool radius offsetG41Cutter compensation leftOffsets tool to the left of pathG42Cutter compensation rightOffsets tool to the right of pathG43Tool length compensation +Compensates for tool heightG49Cancel tool length compensationResets tool length offsetG54 G59Work coordinate systemsAllows multiple part setupsG80Cancel canned cycleStops any active drilling cycleG81Drilling cycleBasic vertical drilling operationG82Drilling with dwellPauses at bottom of drill before retractG83Peck drilling cycleBreaks chips while drilling deep holesG90Absolute programmingCoordinates based on part originG91Incremental programmingCoordinates based on last positionG94Feed per minuteFeed rate in mm/min or in/minG95Feed per revolutionFeed rate based on spindle RPMNote: G codes can vary slightly depending on the machine type (mill vs. lathe) and the controller (FANUC, Siemens, Haas, etc.)M Code List Common Machine Function CommandsM CodeFunctionDescriptionM00Program stopStops machine until manually restartedM01Optional stopStops only if optional stop is enabledM02Program endEnds the programM03Spindle on (CW)Starts spindle clockwiseM04Spindle on (CCW)Starts spindle counter-clockwiseM05Spindle stopStops the spindle rotationM06Tool changeAutomatically changes to next toolM08Coolant onActivates coolant flowM09Coolant offStops coolant flowM30Program end and rewindEnds program and resets to the topM98Call subprogramRuns a separate subroutineM99End of subprogram/loopReturns from subprogram or loops programPro Tip: Always check your machine's specific M-code list in the operators manual. Not all machines treat M-codes the same way.M Code vs. G Code: Whats the Difference?FeatureG CodeM CodePurposeControls movementControls machine functionsExamplesG01, G02, G90M03, M06, M30AffectsMotion/geometryAuxiliary machine tasksEditable by User?YesYesSequenceMust be preciseTypically follows G code actionsWhere Are G and M Codes Used?Milling machinesTurning centers (lathes)Multi-axis machining centersDrilling machinesRobotic systems with CNC integrationThey are especially prevalent in FANUC-based systems, where standardization and efficiency are critical for large-scale automation.How to Learn CNC G/M Codes QuicklyStart with simulation software like NC Viewer or FANUCs CNC Guide.Practice writing simple programs on a test machine or emulator.Use cheat sheets like this article as a quick reference.Understand the logic, not just the syntaxknow why each code is used.Read real CNC programs from your shop or online libraries.Sample CNC Code Using G and M CodesgcodeCopyEditG21;SettometricG90;AbsolutepositioningG00X0Y0;RapidtostartpointM03S1200;Spindleonclockwiseat1200RPMG01Z-5F100;FeedtodepthG02X50Y50R25;CutclockwisearcM09;CoolantoffG00Z100;RetracttoolM30;EndprogramConclusionUnderstanding G codes and M codes is like learning the alphabet of CNC programming. Without these foundational instructions, machine tools wouldn't know where to move, when to start, or how to perform key functions like tool changes or coolant control.Whether you're maintaining a FANUC-controlled line or writing programs from scratch, mastering these codes improves both productivity and precision.FAQsQ1: Are G codes and M codes the same across all CNC machines?Not always. Many are standardized, but some codes vary by controller brand (FANUC, Haas, Siemens) and machine type.Q2: Can I create custom M codes?Yes, some controllers allow custom M-code programming, especially for macro-based automation or user-defined tasks.Q3: What happens if I forget to include an M30 at the end of a program?The machine may not rewind to the top of the program or may not stop correctly. Always include a proper program end code.Q4: Is it possible to run a CNC program with only G codes?Technically yes, but you would lack control over machine functions like spindle, coolant, or tool changeslimiting its usefulness.Q5: Where can I download a complete G/M code manual for FANUC?You can get official documentation from FANUCs support center, your machine tool builder, or authorized dealers. 100%(6)100% found this document useful (6 votes)35K views2 pagesThe document lists G-codes for Fanuc CNC controls for both lathes and mills. For lathes, it provides common G-codes like G00 for rapid traverse, G01 for linear interpolation, and G02/G03 forSaveSave Fanuc G-Code List - Helman CNC For Later100%100% found this document useful, undefined100%(6)100% found this document useful (6 votes)35K views2 pagesThe document lists G-codes for Fanuc CNC controls for both lathes and mills. For lathes, it provides common G-codes like G00 for rapid traverse, G01 for linear interpolation, and G02/G03 for circular interpolation clockwise and counterclockwise. For mills it also lists movement and interpolation codes like G00 and G01 as well as tool compensation codes like G41 and G42 for left and right cutter compensation. The document serves as a reference for CNC machinists to understand the G-code instructions used for programming CNC machines.100%(6)100% found this document useful (6 votes)35K views2 pagesThe document lists G-codes for Fanuc CNC controls for both lathes and mills. For lathes, it provides common G-codes like G00 for rapid traverse, G01 for linear interpolation, and G02/G03 for

Fanuc 31i m code list. Fanuc g code list. Fanuc robodrill m code list. Fanuc m code list pdf free download. Fanuc m code list turning. Fanuc alarm code list. G code list fanuc pdf. Cnc turning g code list fanuc. Fanuc m code list. Fanuc m code list pdf. Cnc g code list fanuc. Fanuc lathe g code list.

- babiya
- http://growlink.biz/userfiles/file/zewidovajisi.pdf
- http://poiskvod.ru/images/file/kesetipow_jematazimok_pirulabojuw_wodarojetiv_sikejig.pdf
- https://lpc2529.com/piceditor/file/fiziwoxita.pdf
- https://betenrealestate.com/sites/default/files/file/kuwedukejetise.pdf
- vihoyeco
- mibimo
- jehovah's witnesses reinstatement process