ERROR SOLUTIONS

CNC8070

FAGOR

INDEX

FAGOR

CNC8070

ERROR SOLUTIONS

INDEX

ERRORS 0000-0999

0001 'SYSTEM ERROR'

DETECTION   During execution.
CAUSE   Software or hardware error that cause corrupt data and/or incoherent results.
SOLUTION   This type of errors usually force the CNC output.
              Contact your supplier.

0002 'SYSTEM WARNING'

DETECTION   During execution.
CAUSE   Warning of internal situations that could become system errors.
SOLUTION   They're usually restored with a simple ESCAPE.
              Contact your supplier.

0003 'Error when requesting memory'

DETECTION   On CNC power-up.
CAUSE   The CNC does not have enough memory.
SOLUTION   Contact your supplier.

0004 'Checksum error in PLC data'

DETECTION   On CNC power-up.
CAUSE   The PLC data saved into a disk are not valid. The file containing the data (\FagorCNC\ PLC8070\ User\ project\ objects\ plcData.bin) is either missing, cannot be accessed or is damaged. As a result, some PLC data (marks, counters, timers and registers) has been lost.
SOLUTION   If the error persists after powering the CNC up several times, contact your supplier.

0005 'The CNC was not turned off properly, it must be homed'

DETECTION   On CNC power-up.
CAUSE   The CNC data saved into a disk are not valid (coordinates, offsets, PARTC). The file containing the data (\FagorCNC\ Data\ orgData.tab) is either missing, cannot be accessed or is damaged. As a result, some data has been lost. It loses the values for coordinates, offsets, PARTC, kinetics, etc.
SOLUTION   If the error persists after powering the CNC up several times, contact your supplier.

0020 'Wrong access to a variable'

DETECTION   CNC power-up or page change at the interface.
CAUSE   A variable is been accessed that is not recognized by the CNC.
SOLUTION   Contact the machine manufacturer or the person who designed the interface screens to remove the access to a variable unrecognized by the CNC.

0040 'M before-before or Before-After with subroutine does not admit movements in the block'

DETECTION   Machine parameter validation.

CAUSE   An M function with associated subroutine cannot be Before-Before or Before-After. The subroutine associated with an M function is always executed at the end of the block, after the rest of the block.
- Before-Before means that the M is executed before the rest of the block and the synchronization with the PLC is also done before the block.
- Before-After means that the M is executed before the rest of the block and the synchronization with the PLC is done after the block.

SOLUTION   Either define the M without subroutine or define the M as After-After.

0041 'Duplicate M in the table'

DETECTION   Machine parameter validation.

CAUSE   There is an M repeated in the M table.

SOLUTION   Delete the repeated M from the table.

0042 'Wrong machine parameter value'

DETECTION   Machine parameter validation.

CAUSE   Machine parameter out of range.

SOLUTION   Set the machine parameter to a value between the maximum and minimum values shown at the warning.

0043 'Restart the CNC to assume the new value'

DETECTION   Machine parameter validation.

CAUSE   A machine parameter has been modified that requires restarting the CNC in order to assume the new value.

SOLUTION   Restart the CNC.

0044 'Wrong axis name or undefined axis name'

DETECTION   Machine parameter validation.

CAUSE   The wrong name has been assigned to a machine parameter that represents the name of an axis.

SOLUTION   Assign a correct name to the parameter. The first character is mandatory A, B, C, X, Y, Z, U, V or W and the second one optional, a number from 1 to 9.

0045 'Wrong spindle name'

DETECTION   Machine parameter validation.

CAUSE   The wrong name has been assigned to a machine parameter that represents the name of a spindle.

SOLUTION   Assign a correct name to the parameter. The first character S is mandatory and the second one optional, a number from 1 to 9.

0046 'Nonexistent axis'

DETECTION   Machine parameter validation.

CAUSE   A machine parameter that represents the name of an axis has been assigned a name that does not exist in parameter AXISNAME.

SOLUTION   Assign a correct name to the parameter. A name that appears in AXISNAME.

0047 'A main axis cannot be defined as slave'

DETECTION   Machine parameter validation.

CAUSE   In a Gantry pair, an axis has been set as slave that is the master of another pair.

SOLUTION   Do not set it as slave.

0048 'An axis cannot be a slave of several masters'

DETECTION   Machine parameter validation.

CAUSE   In a Gantry pair, an axis has been set as slave that is already a slave in another pair.

SOLUTION   Do not set it as slave again.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 0000-0999

Page 2 of 128

0049 ''A master axis cannot be a slave and vice versa'

DETECTION   Machine parameter validation.

CAUSE   If an axis is set as slave in another Gantry pair, cannot be the master of another pair. An axis already set as master cannot be the slave in another pair either.

SOLUTION   Check the Gantry axes table.

0050 'The master and slave axes must be of the same type (AXISTYPE)'

DETECTION   Machine parameter validation.

CAUSE   The axes of a Gantry pair must have the same machine parameter AXISTYPE (linear, rotary).

SOLUTION   Check the Gantry axes table or the AXISTYPE parameters of both axes of the pair.

0051 'The master and slave axes must have certain parameters with the same value'

DETECTION   Machine parameter validation.

CAUSE   If they are two linear axes, the machine parameters that must have the same values are: AXISMODE, FACEAXIS and LONGAXIS. If they are rotary axes, the parameters that must have the same values are: AXISMODE, SHORTESTWAY and CAXIS.

SOLUTION   Check the parameters mentioned earlier.

0052 'Module difference too small'

DETECTION   Machine parameter validation.

CAUSE   The difference between machine parameters MODUPLIM and MODLOWLIM is lower than the resolution of the axis.

SOLUTION   Increase MODUPLIM or decrease MODLOWLIM.

0053 'Parameter MGPAXIS repeated in several handwheels'

DETECTION   Machine parameter validation.

CAUSE   There are two or more handwheels in the manual table that have the same axis name assigned to them.

SOLUTION   Assign each handwheel to a different axis.

0054 'The MOVAXIS and COMPAXIS axis must be different'

DETECTION   Machine parameter validation.

CAUSE   In one of the cross compensation tables, the compensated axis and the axis whose movement affects the compensated axis are the same.

SOLUTION   Assign different axis names.

0055 'The same axis is causing and suffering the cross compensation error'

DETECTION   Machine parameter validation.

CAUSE   Checking the associations of compensated (affected) axes (COMPAXIS) and those (MOVAXIS) whose movements affect the other ones in the cross compensation tables, there is an axis defined as being affected by its own movement which does not make sense.

SOLUTION   Define MOVAXIS and COMPAXIS properly in the cross compensation table.

0056 'Compensation table positions not in ascending order'

DETECTION   Machine parameter validation.

CAUSE   The POSITION parameter within the compensation tables must have ascending values.

SOLUTION   Assign a value between the previous and the next value.

0057 'Compensation table with error slope greater than 1'

DETECTION   Machine parameter validation (leadscrew error compensation table)

CAUSE   The error slope in leadscrew compensation tables cannot be greater than 1. In other words, the difference between consecutive errors cannot be greater than the gap between them.

SOLUTION   Write errors corresponding to greater gaps. If this is not possible, the error entered for the leadscrew is so large that cannot be compensated for.

0058 'The CNC must be restarted too assume the changes in the HMI table,'

DETECTION	Machine parameter validation.
CAUSE	The CNC must be restarted in order to assume the changes made to the HMI table.
SOLUTION	Restart the CNC.

0059 'The CNC must be restarted too assume the changes in the tool magazine table,'

DETECTION	Machine parameter validation.
CAUSE	The CNC must be restarted in order to assume the changes made to the tool magazine table.
SOLUTION	Restart the CNC.

0060 'The maximum jogging feedrate exceeds the maximum feedrate set for the axis'

DETECTION	Machine parameter validation.
CAUSE	Machine parameter MAXMANFEED greater than G00FEED.
SOLUTION	Decrease MAXMANFEED.

0061 'The manual rapid feedrate exceeds the maximum feedrate set for the axis'

DETECTION	Machine parameter validation.
CAUSE	Machine parameter JOGRAPFEED greater than G00FEED.
SOLUTION	Decrease JOGRAPFEED.

0062 'The Continuous Jog feedrate exceeds the maximum feedrate set for the axis'

DETECTION	Machine parameter validation.
CAUSE	Machine parameter JOGFEED greater than G00FEED.
SOLUTION	Decrease JOGFEED.

0063 'The Incremental Jog feedrate exceeds the maximum feedrate set for the axis'

DETECTION	Machine parameter validation.
CAUSE	Machine parameter INCJOGFEED greater than G00FEED.
SOLUTION	Decrease INCJOGFEED.

0064 'The master and slave axes must have the same I0TYPE'

DETECTION	Machine parameter validation.
CAUSE	The Io types of the Gantry axes are not the same.
SOLUTION	Change the value of machine parameter I0TYPE.

0065 'A Hirth axis cannot be Gantry"

DETECTION	Machine parameter validation.
CAUSE	The CNC does not admit pairs of Grantry axes that are Hirth axes.
SOLUTION	Remove the Gantry pair or make a Gantry pair with axes that are not Hirth.

0066 'A Gantry axis cannot have REFSHIFT'

DETECTION	Machine parameter validation.
CAUSE	One of the axes of the Gantry pair has the machine parameter REFSHIFT (offset referred to the reference point) other than 0 in one of its SETs (parameter sets).
SOLUTION	REFSHIFT = 0.

0067 'A Gantry axis cannot be Unidirectional'

DETECTION	Machine parameter validation.
CAUSE	An axis with a single rotating direction (UNIDIR) cannot be Gantry.
SOLUTION	UNIDIR = 0.

0068 'Gantry Axes: the slave cannot go before the master in AXISNAME'

DETECTION	Machine parameter validation.
CAUSE	The slave axis is above the master axis in the axis name assigning tables (AXISNAME).
SOLUTION	Swap the axis names in AXISNAME or swap the master and the slave axes of the Gantry pair.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 0000-0999

Page 4 of 128

0069   'Gantry Axes: The slave cannot have DECINPUT (Home switch) if the master does not have one'

DETECTION   Machine parameter validation.
CAUSE   An axis of a Gantry pair cannot be homed with a home switch (DECINPUT = TRUE).
SOLUTION   DECINPUT = FALSE.

0070   'Gantry Axes: LIMIT+ and LIMIT- must be the same for the master and for the slave'

DETECTION   Machine parameter validation.
CAUSE   The software limits of the Gantry axes are different.
SOLUTION   Assign to the slave the same limits (LIMIT+ and LIMIT-) as for the master.

0071   'Following error monitoring not active at the CNC'

DETECTION   Machine parameter validation.
CAUSE   This monitoring has not been activated.
SOLUTION   Parameter of the axis set FLWEMONITOR = TRUE.

0072   'Feedback alarm not activated'

DETECTION   Machine parameter validation.
CAUSE   The feedback alarm of the axis is not activated.
SOLUTION   Parameter of the axis set FBACKAL = TRUE.

0073   'Software limits not activated'

DETECTION   Machine parameter validation.
CAUSE   The software limits are not activated. Axis parameters LIMIT+ and LIMIT- are set to 0.
SOLUTION   Set axis parameters LIMIT+ and LIMIT- to the axis limit values.

0074   'Tendency test not activated'

DETECTION   Machine parameter validation.
CAUSE   The tendency test is not activated. This test consists in checking that the axis moves in the commanded direction.
SOLUTION   Set axis parameter TENDENCY to TRUE.

0075   'Wrong I/O configuration table'

DETECTION   Machine parameter validation.
CAUSE   Parameter NDINPUT or NDOUTPUT must be equal to the number of I/O modules detected by hardware.
SOLUTION   Change the value of machine parameter NDINPUT or INDOUTPUT.

0076   'The sum of axes or spindles per channel exceeds the total number of axes or spindles'

DETECTION   Machine parameter validation.
CAUSE   Parameter CHNAXIS or CHNSPDL (number of axes and spindles per channel) exceeds the value of parameter NAXIS or NSPDL (number of axes and spindles of the system).
SOLUTION   Change CHNAXIS or CHNSPDL.

0077   'Axis or spindle assigned to more than one channel'

DETECTION   Machine parameter validation.
CAUSE   There is an axis or a spindle assigned to several channels.
SOLUTION   Check the table of parameters CHAXISNAME and CHSPDLNAME in all the channels.

0078   'The master and slave axes must belong to the same channel'

DETECTION   Machine parameter validation.
CAUSE   There is a Gantry pair made up of axes from different channels.
SOLUTION   Check the Gantry axis tables and the tables for assigning axes to channels CHAXISNAME.

0079 'A Gantry slave axis cannot be parked'

DETECTION	Validation of machine parameters or startup processes.
CAUSE	The PARKED signal of the Gantry slave axis is active. While powering the CNC up, it has been detected that the slave axis of a Gantry pair is parked.
SOLUTION	Unpark the axis or cancel the Gantry pair.

0080 'To validate the axis, validate the GENERAL PARAMETERS table'

DETECTION	Machine parameter validation.
CAUSE	The NAXIS value of the general parameter table has been increased and, without validating this table, an attempt has been made to validate the table of one of the new axes of the system.
SOLUTION	Validate the general parameter table of the axis before validating the parameters of the axis. Since this will detect the change in NAXIS; to solve it, restart the CNC.

0081 'The in-position zone cannot be smaller than the resolution of the axis'

DETECTION	Machine parameter validation.
CAUSE	
SOLUTION	Increase the in-position zone of the axis, parameter INPOSW within the axis' SET table.

0082 'Impossible leadscrew error or cross compensation table for all the ranges of the axis'

DETECTION	Machine parameter validation.
CAUSE	A rotary and module axis has different machine parameters MODUPLIM/MODLOWLIM in a SET and leadscrew compensation or is part of a cross compensation as the affecting axis (MOVAXIS)
SOLUTION	Set the same module limits for all the SETs of the Gantry pair. If this is not possible, there is no solution.

0083 'The master and slave axes must be of the same type (DRIVEAXIS)'

DETECTION	Machine parameter validation.
CAUSE	There is a Gantry axis with axes having different types of drives (sercos/analog).
SOLUTION	Check the machine parameters. Assign the same type of drive (DRIVEAXIS) to both axes.

0084 'An axis or spindle that cannot be swapped cannot be left unassigned to a channel'

DETECTION	Machine parameter validation.
CAUSE	Spindles or axes with parameter AXISEXCH = NO must be assigned to a channel (showing in the CHAXISNAME or CHSPDLNAME table of a channel).
SOLUTION	Assign the axis or spindle to a channel (CHAXISNAME or CHSPDLNAME).

0085 'There is no SERCOS axis in the system'

DETECTION	Machine parameter validation.
CAUSE	A SERCOS variable has been defined in the relevant parameter table, but the system has no Sercos axis.
SOLUTION	Eliminate the sercos variables defined.

0086 'It is not a Sercos axis'

DETECTION	Machine parameter validation.
CAUSE	A SERCOS variable has been defined for a non-sercos axis.
SOLUTION	Eliminate the variable for this axis.

0087 'Too many sercos variables'

DETECTION	Machine parameter validation.
CAUSE	The maximum number of SERCOS variables (100) has been exceeded.
SOLUTION	Decrease the number of variables.

0088 'Trace of internal variables activated'

DETECTION	Machine parameter validation.
CAUSE	A trace of an internal variable is being executed.
SOLUTION	Contact Fagor.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 0000-0999

Page 6 of 128

0089 **'It starts up with a single channel due to errors detected in machine parameters.'**

DETECTION	Machine parameter validation.
CAUSE	Errors or warnings have come up while validating machine parameters related to the axes or spindles of a channel.
	For example, a channel has an axis (CHAXISNAME) associated to it, but it is not on the list of the system axes (AXISNAME). Being impossible to configure the n channels, the CNC starts up with a configuration of a single channel.
SOLUTION	Change the machine parameters to prevent the other warnings or errors related to the other machine parameters from coming up. This warning is removed without having to change parameter NCHANNEL (number of channels)

0090 **'It starts up with the default axis configuration due to errors in machine parameters'**

DETECTION	Machine parameter validation.
CAUSE	Errors or warnings have come up while validating machine parameters related to the axes or spindles.
	For example, the number of axes (NAXIS) is greater than the elements of the list that contains the names of the axes (AXISNAME). Being impossible to start up with the user configuration, the CNC starts up with the default configuration.
SOLUTION	Change the machine parameters to prevent the other warnings or errors related to the other machine parameters from coming up.

0091 **'Sercos variables having the same identifier (ID) cannot have different mnemonic'**

DETECTION	Machine parameter validation.
CAUSE	In the sercos variables table of machine parameters (OEM, DRIVEVAR, DATA) some variables have been written with the same name (MNEMONIC) and different sercos identifier (ID).
SOLUTION	Assign the same name to the sercos variables having the same identifier (ID).

0092 **'Sercos variables having the same identifier (ID) cannot have different MODE or TYPE'**

DETECTION	Machine parameter validation.
CAUSE	In the sercos variables table of machine parameters (OEM, DRIVEVAR, DATA) some variables have been written with the same identifier (ID) and different TYPE (Synchronous or Asynchronous) or different MODE (Read or Write).
SOLUTION	Assign the same MODE and TYPE to the sercos variables having the same identifier (ID).

0093 **'Sercos variables having the same name (MNEMONIC) cannot have different ID, MODE or TYPE'**

DETECTION	Machine parameter validation.
CAUSE	In the sercos variables table of machine parameters (OEM, DRIVEVAR, DATA) some variables have been written with the same name (MNEMONIC) and different TYPE (Synchronous or Asynchronous) or different MODE (Read or Write) or different Sercos identifier (ID).
SOLUTION	Assign the same MODE, TYPE and ID to the sercos variables having the same name (MNEMONIC).

0100 **'Too many variables waiting to be reported'**

DETECTION	During execution.
CAUSE	It informs that the number of variables that have been modified and that must be reported to the interface exceeds the maximum allowed.
SOLUTION	Press the [ESC] key.

0104 **'Communication time out'**

DETECTION	During execution.
CAUSE	It does not end successfully the reading / writing of an external variable.
SOLUTION	Contact your supplier.

0105 'Parameters cannot be validated while executing a program'

DETECTION During execution.

CAUSE   An attempt has been made to validate a machine parameter table while the program is in execution or interrupted.

SOLUTION   Finish or abort the program in order to be able to validate the machine parameter table.

0150 'Too many open files'

DETECTION While executing a part-program with external subroutines.

CAUSE   The number of open files (main program plus external subroutines) is greater than 20.

SOLUTION   Decrease the number of external subroutines open in the part-program at the same time.

0151 'Writing access denied'

DETECTION Access to a file.

CAUSE   An attempt has been made to write into a file that has no writing permission.

SOLUTION   Give the file writing permission.

0152 'The file cannot be open'

DETECTION Access to a file.

CAUSE   A file could not be opened for reading or writing.

SOLUTION   Check that the file exists in the directory and that it has the right permissions for the action to be carried out.

Check that the file is not corrupted by some previous action.

0153 'Reading access denied'

DETECTION Access to a file.

CAUSE   An attempt has been made to read into a file that has no reading permission.

SOLUTION   Give the file reading permission.

0160 'Axis/Set not available in the system'

DETECTION Execution of the commands SET AX, CALL AX, G112.

CAUSE   When changing axis configurations or parameter sets, an attempt has been made to access a nonexistent axis or parameter set, or an axis currently associated with another channel.

SOLUTION   Do not access the unavailable axis or parameter set or free the axis in the other channel using a FREE AX.

0168 'LR Overflow'

DETECTION In the position loop of Sercos axes.

CAUSE   The position loop of the Sercos axes exceeds the time allowed.

SOLUTION   Adjust parameter LOOPTIME.

0200 'Failure when requesting a VxD'

DETECTION When reading the battery status.

CAUSE   It is not possible to connect with VcompciD.

SOLUTION   Contact your supplier.

0201 'Mains failure. PC powered by a battery'

DETECTION When reading the battery status.

CAUSE   The mains supply to the PC has dropped and it is now powered by the battery.

SOLUTION   If the mains failure fortuitous, is look for possible causes. If it is caused by the operator, let the automatic PC-turn-off sequence run to its completion.

Whether the mains failure is fortuitous or caused by the operator, let the automatic PC-turn-off sequence run to its completion. If the mains failure is fortuitous, look for the possible causes.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 0000-0999

Page 8 of 128

ERRORS 1000-1999

1000 'The function of instruction requires programming the axes'

DETECTION	During execution.
CAUSE	The axes affected by the programmed instruction or G function have not been programmed.
SOLUTION	Check the program.

1004 'Zero spindle speed'

DETECTION	During execution.
CAUSE	Being function G63 active, a zero spindle speed has been programmed.
SOLUTION	Program the spindle speed S.

1005 'Motion block with zero feedrate'

DETECTION	During execution.
CAUSE	No feedrate (F) has been programmed to move the axes.
SOLUTION	Program the feedrate F.

1006 'G20: spindle not allowed'

DETECTION	During execution.
CAUSE	Function G20 does not allow programming of the spindle.
SOLUTION	Check the program.

1007 'The programmed function requires a nonexistent main axis'

DETECTION	During execution.
CAUSE	The following functions require the existence of both axes of the main plane: • G2, G3, G8, G9, G30, G36, G37, G38, G39 and G73. • G20 if collision detection is active. • Activate collision detection, G20. The following functions require the existence of one of the two axes of the main plane: G11, G12, G13 and G14.
SOLUTION	Check the program.

1008 'Coordinates out of range'

DETECTION	During execution.
CAUSE	The possible causes are: • Using function G101, an attempt has been made to include in the axis a measuring offset too large. • The coordinate programmed for the axis is too large.
SOLUTION	Check the program.

1009 'G4: the dwell has been programmed twice, directly and using K'

DETECTION	During execution.
CAUSE	The dwell G4 has been programmed twice in the same block, directly with a number and with K.
SOLUTION	Program the dwell function G4 only once.

1010 'Program G4 K'

DETECTION	During execution.
CAUSE	The function has not been programmed correctly.
SOLUTION	There are two ways to program the dwell with G4:

- G4 <time>
- G4 K<time> .

In both cases, the dwell must be programmed after G4.

The second case does not allow "=" after "K".

1011 'G4: dwell out of range'

DETECTION	During execution.
CAUSE	Too large a value has been programmed for the dwell function G4.
SOLUTION	The maximum value allowed for the dwell is 2147483646.

1012 'G4: the dwell cannot be programmed using K'

DETECTION	During execution.
CAUSE	The letter K is associated with the third axis of the channel and in this case there is no third axis.
SOLUTION	If a third axis is not desired in the channel, the dwell may be programmed directly with a number.

1013 'G4: the dwell cannot be negative'

DETECTION	During execution.
CAUSE	A negative dwell has been programmed using function G4 or programming the #TIME instruction.
SOLUTION	The programmed dwell must be equal to or greater than zero.

1014 'It is no t possible to program in diameters with mirror image on the face axis'

DETECTION	During execution.
CAUSE	The face axis (machine parameter FACEAXIS = Yes) cannot have both the mirror image and programming in diameters active at the same time.
SOLUTION	Activate either the mirror image or diameter programming for the face axis.

1015 'Center coordinates out of range'

DETECTION	During execution.
CAUSE	Too large values of I, J, K have been programmed for the center of the circular interpolation or for the center of rotation of the coordinate system.
SOLUTION	Program smaller values.

1016 'Negative values cannot be used when programming an axis in diameters'

DETECTION	During execution.
CAUSE	When programming in absolute coordinates (function G90) a negative coordinate has been programmed for an axis that is in diameters (machine parameter DIAMPROG).
SOLUTION	Coordinates programmed in absolute coordinates for the axes in diameters must be positive.

1017 'G198: negative software limit out of range'

DETECTION	During execution.
CAUSE	Too high a value has been programmed for the negative software limit.
SOLUTION	Check the program.

1018 'G199: positive software limit out of range'

DETECTION	During execution.
CAUSE	Too high a value has been programmed for the positive software limit.
SOLUTION	Check the program.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 10 of 128

1019 'No measurement has been taken on the requested axis (axes)'

DETECTION	During execution.
CAUSE	Function G101is being used to include the measurement offset on an axis for which:
	• No previous measurement has been taken (G100).
	• The measurement taken (G100) has been canceled (G102).
SOLUTION	Take a measurement with the axis on which to apply function G101.

1020 'Negative ramp time'

DETECTION	During execution.
CAUSE	Using function G132, a negative ramp time has been programmed.
SOLUTION	The ramp time must be equal to or greater than zero.

1021 'Ramp time out of range'

DETECTION	During execution.
CAUSE	With function G132, a ramp time value too high has been programmed.
SOLUTION	Check the program.

1022 'Percentage of Feed-Forward out of range'

DETECTION	During execution.
CAUSE	The probable causes are:
	• With function G134, too high a value has been programmed for the percentage of Feed-Forward.
	• With function G135, too high a value has been programmed for the percentage of AC-Forward.
SOLUTION	The percentage of Feed-Forward or AC-Forward must be greater than zero and smaller than 120.

1023 'Wrong set number

DETECTION	During execution.
CAUSE	The programmed set number for the axis is wrong.
SOLUTION	The set programmed for the axis must be greater than zero and smaller than or equal to machine parameter NPARSETS of the axis.

1024 'Set number out of range'

DETECTION	During execution.
CAUSE	The programmed parameter set value for the axis is too high.
SOLUTION	The maximum value allowed for the set number is 65535.

1025 'Programmed distance equal to zero'

DETECTION	During execution.
CAUSE	No movement has been programmed in the G63 block.
SOLUTION	Program a movement.

1026 'Wrong circular path with the programmed radius'

DETECTION	During execution.
CAUSE	The programmed radius for the circular interpolation is too small.
SOLUTION	Check the program.

1027 'The starting point and the end point of the circular path are the same (infinite solutions)'

DETECTION	During execution.
CAUSE	A zero radius has been programmed for the circular interpolation and there are infinite solutions.
SOLUTION	Check the program.

1028 'The difference between the programmed center and the calculated one is too large'

DETECTION	During execution.
CAUSE	In a circular interpolation with function 265 active, the difference between the initial radius and the final one exceeds the values of machine parameters CIRINERR and CIRINFACT.
SOLUTION	Program the circular interpolation correctly.

1029 'Zero radius on circular path'

DETECTION	During execution.
CAUSE	The probable causes are: • A zero radius has been programmed in a circular interpolation. • Being function G265 active, the CNC calculates a zero radius based on the center coordinates programmed for the circular interpolation. • Being function G265 not active, null center coordinates have been programmed for the circular interpolation.
SOLUTION	In a circular interpolation, the radius cannot be zero. Both center coordinates cannot be null either.

1030 '#AXIS programmed without G200/G201/202'

DETECTION	During execution.
CAUSE	The #AXIS instruction has been programmed without programming G200/201/202 on the same line.
SOLUTION	Program both functions on the same line.

1031 '#AXIS expected'

DETECTION	During execution.
CAUSE	Function G201 has been programmed without progromming #axis on the same line.
SOLUTION	Program both functions on the same line.

1032 'Spindle position missing for M19'

DETECTION	During execution.
CAUSE	Function M19 has been programmed, but the spindle position has not been programmed.
SOLUTION	Check the program.

1035 '#SLOPE: parameter out of range'

DETECTION	During execution.
CAUSE	The value programmed for the parameter is too high.
SOLUTION	Program smaller values.

1037 'Center coordinates ignored with G0/G1/G100/G63 active'

DETECTION	During execution.
CAUSE	It informs that the values programmed with I, J, K will be ignored.
SOLUTION	These functions do not require the programming of these parameters.

1038 'Radius compensation cannot be active while measuring'

DETECTION	During execution.
CAUSE	An attempt has been made to execute G100 while radius compensation is active, G41-G42.
SOLUTION	Check the program.

1039 'There is a previous measurement value for the axis (axes)'

DETECTION	During execution.
CAUSE	An attempt has been made to do a G100 when the axis already has a measurement offset included by a previous G101.
SOLUTION	The offset included in the axis may be eliminated with function G102.

1040   **'Home search not allowed on an axis in G201'**

DETECTION   During execution.

CAUSE   An axis cannot be homed if it is in additive manual mode (G201).

SOLUTION   Use function G202 to cancel the additive manual mode of the axis in order to home it. Then, activate function G201 again if you wish.

1041   **'Corrected circular path center out of range'**

DETECTION   During execution.

CAUSE   A circular interpolation has been defined:
- Using the radius "R" or "R1" and the coordinates of the end point.
- Using the coordinates of the middle and final points being function G265 (correction of the center of the circle) active.

The coordinates values of the center of the interpolation calculated by the CNC are too large.

SOLUTION   The values programmed in the block for the middle or final points for the interpolation radius are too large.

1043   **'The third axis of the plane cannot be the same as the first or the second one'**

DETECTION   During execution.

CAUSE   Parameter 5 programmed in function G20 (plane change) is the same as 1 or 2.

SOLUTION   If the longitudinal axis of the tool (parameter 3) is the same as the first or second axis of the plane (parameters 1 and 2), the third axis must be programmed with parameter 5. This parameter must not be the same as the first or the second one.

1044   **'The first and second axis of the plane cannot be the same'**

DETECTION   During execution.

CAUSE   Using function G20 (plane change) the same axis has been programmed as the first (parameter 1) and the second (parameter 2) axis of the plane.

SOLUTION   Check the program.

1045   **'The first axis of the plane has been programmed wrong'**

DETECTION   During execution.

CAUSE   Using function G20 (plane change) the first axis of the plane (parameter 1) has been programmed wrong.

SOLUTION   The first axis of the plane must be one of the main three axes of the configuration.

1046   **'The second axis of the plane has been programmed wrong'**

DETECTION   During execution.

CAUSE   Using function G20 (plane change) the second axis of the plane (parameter 2) has been programmed wrong.

SOLUTION   The first axis of the plane must be one of the main three axes of the configuration.

1047   **'A third axis is required for the plane (index 5) '**

DETECTION   During execution.

CAUSE   Parameter 5 programmed with function G20 (plane change) missing or wrong.

SOLUTION   If the longitudinal axis of the tool (parameter 3) is the same as the first or second axis of the plane (parameters 1 and 2), the third axis must be programmed with parameter 5 which must not be the same as the first or the second one and must be one of the main three axes of the configuration.

1048   **'Tool length compensation with radius out of range'**

DETECTION   During execution.

CAUSE   The tool dimensions exceed the maximum values.

SOLUTION   Modify the tool dimensions.

1049   **'Face axis (FACEAXIS) defined twice in the active plane'**

DETECTION   During execution.

CAUSE   It informs that the two axes of the main plane are face axes. They both have machine parameter FACEAXIS = Yes.

SOLUTION   There can only be one face axis in the work plane.

1050   'Considering the tool offsets, it exceeds the data range'

DETECTION	During execution.
CAUSE	The tool dimensions exceed the maximum values.
SOLUTION	Modify the tool dimensions.

1052   'Values resulting from the measurement out of range'

DETECTION	During execution.
CAUSE	When taking a measurement with function G100, the obtained value is too large either for the coordinate where it probed, either for the offset obtained in that probing move.
SOLUTION	The value obtained in that probing move must be between (-2147483647, 2147483646)

1054   'Nonexistent fixture'

DETECTION	During execution.
CAUSE	The programmed fixture index is wrong.
SOLUTION	The value of the fixture index must be more than zero and less than ten.

1055   'D and the tool radius cannot be modified in the same block'

DETECTION	During execution.
CAUSE	An attempt has been made to write the "V.G.TOR" variable in the same block where a tool change or tool offset change is programmed.
SOLUTION	Programming in different lines.

1056   'Too many external variables'

DETECTION	During execution.
CAUSE	The maximum number of external variables (500) supported by the CNC has been reached.
SOLUTION	Check the program.

1057   'Variable without reading permission'

DETECTION	During execution.
CAUSE	An attempt has been made from part-program or MDI to read a variable without reading permission via program.
SOLUTION	The variable cannot be read from a part-program or MDI. The programming manual indicates whether or not each variable has reading permission via program / MDI, interface and PLC.

1059   'Variable without writing permission'

DETECTION	During execution.
CAUSE	The user has tried to write a variable without writing permission. An attempt has been made from part-program or MDI to write a variable without writing permission via program.
SOLUTION	The variable cannot be written. The variable cannot be written from a part-program or MDI.
	The programming manual indicates whether or not each variable has writing permission via program / MDI, interface and PLC.

1060   'N label value out of range'

DETECTION	During execution.
CAUSE	An "N" block number has been defined that is not within the permitted range.
SOLUTION	The block number must be a positive value lower than 2147483646.

1061   'Nonexistent G function'

DETECTION	During execution.
CAUSE	The programmed G function does not exist.
SOLUTION	Check the program.

1062   'Incompatible G functions'

DETECTION	During execution.
CAUSE	Two G functions incompatible with each other have been programmed in the same block.
SOLUTION	Program them in different blocks.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 14 of 128

1063 'Incompatible G functions (G108/G109/G193)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1064 'Incompatible G functions (G196/G197)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1065 'Incompatible G functions (G17/G18/G19/G20)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1066 'Incompatible G functions (G136/G137)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1067 'Incompatible G functions (G40/G41/G42)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1068 'Incompatible G functions (G151/G152)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1069 'Incompatible G functions (G54-G59/G159)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1070 'Incompatible G functions (G5/G7/G50/G60/G61)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1071 'Incompatible G functions (G70/G71)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1072 'Incompatible G functions (G80-G88/G160-G166/G281-G286/G287-G297)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1073 'Incompatible G functions (G90/G91)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1074 'Incompatible G functions (G93/G94/G95)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1075 'Incompatible G functions (G96/G97/G192)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1077 'Incompatible G functions (G115/G116/G117)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

11079 'Incompatible G functions (G138/G139)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1080 'Incompatible G functions (G6/G261/G262)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1081 'Incompatible G functions (G264/G265)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1082 'Incompatible G functions (G200/G201/G202)'

DETECTION	During execution.
CAUSE	Two or more G functions of the same group cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1084 'Plane change not allowed while tool radius compensation is active'

DETECTION	During execution.
CAUSE	Function G17-G20 (plane change) has been used to change the first or second axis of the plane while tool radius compensation was active.
SOLUTION	Cancel the compensation to define the new work plane.

1085 'G41/G42 not allowed if the first or second axis of the active plane is missing'

DETECTION	During execution.
CAUSE	Tool radius compensation is impossible if one of the two axes of the active plane is missing in the channel.
SOLUTION	Restore the missing axis with the instruction #CALL AX or #SET AX.

1087 '"=" expected'

DETECTION	During execution.
CAUSE	Wrong syntax of the programmed instruction or function.
SOLUTION	Refer to the programming manual. Check the program.

1088 'Wrong offset number'

DETECTION	During execution.
CAUSE	When programming in high level language, the wrong value is assigned to the parameter of the G function.
SOLUTION	The range of values admitted for function G159 is from 1 to 20.

1089 'Incompatible M functions (M3/M4/M5/M19)'

DETECTION	During execution.
CAUSE	More than one M function (M3/M4/M5/M19) have been programmed for the same spindle in the same block.
SOLUTION	Program the M functions for the same spindle in different blocks.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 16 of 128

1090 **'Nonexistent H function'**

DETECTION	During execution.
CAUSE	The programmed H function does not exist.
SOLUTION	The maximum H function number is 65534.

1091 **'T function programmed twice'**

DETECTION	During execution.
CAUSE	More than one T function have been programmed in the same block.
SOLUTION	Program them in different blocks.

1093 **'D function programmed twice'**

DETECTION	During execution.
CAUSE	More than one D function have been programmed in the same block.
SOLUTION	Program them in different blocks.

1094 **'F feedrate programmed twice'**

DETECTION	During execution.
CAUSE	More than one F function have been programmed in the same block.
SOLUTION	Program them in different blocks.

1095 **'Feedrate F cannot be negative or zero'**

DETECTION	During execution.
CAUSE	The feedrate (F) must be positive and other than zero.
SOLUTION	Check the program.

1096 **'The feedrate cannot be programmed with E'**

DETECTION	During execution.
CAUSE	An attempt has been made to program the axis feedrate with the letter E.
SOLUTION	Program the axis feedrate with F.

1097 **'Unknown spindle name'**

DETECTION	During execution.
CAUSE	The possible causes are:
	• The spindle name is wrong.
	• The spindle name is valid, but it does not exist in the system.
	• The spindle exists in the system, but it does not belong to the channel.
SOLUTION	The valid names for the spindle are S, S1, ..., S9.
	The spindle referred to in the block must exist in the system configuration and depending on which instruction it is, in the channel configuration.

1098 **'S speed programmed twice'**

DETECTION	During execution.
CAUSE	More than one S function have been programmed for the same spindle in the same block.
SOLUTION	Program them in different blocks.

1100 **'Parameter index out of range'**

DETECTION	During execution.
CAUSE	An attempt has been made to access an arithmetic parameter whose index is out of the range established by the relevant machine parameters.
	• MINLOCP-MAXLOCP for local parameters.
	• MINGLBP-MAXGLBP for global parameters.
	• MINCOMP-MAXCOMP for common parameters.
SOLUTION	The solutions are:
	• Access a parameter whose index is within the established range.
	• Modify the previous machine parameters accordingly and restart the CNC to assume the new values.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 17 of 128

1101 '#SET IPOPOS instruction programmed wrong'

DETECTION   During execution.

CAUSE   The possible causes are:
- The syntax of the instruction is wrong.
- Only the block number of a label may be programmed in the same block as the instruction.

SOLUTION   In the first case, refer to the programming manual. In the second case, program in different blocks.

1102 'The index for R cannot be other than 1'

DETECTION   During execution.

CAUSE   The radius can only be programmed with R or R1.

SOLUTION   Check the program.

1103 'Nonexistent O function'

DETECTION   During execution.

CAUSE   The programmed O function does not exist.

SOLUTION   Check the program.

1104 'The "%" character is not allowed inside the main program'

DETECTION   During execution.

CAUSE   The "%" character can only be used as first character in the definition of the name of the main program or of a local subroutine.

SOLUTION   Remove this character from the program.

1105 'Assignment operator expected'

DETECTION   During execution.

CAUSE   No assignment operator has been programmed after the variable or parameter.

SOLUTION   The valid assignment operators are "=", "+=", "-=", "*=", "/=".

1106 '"]" expected'

DETECTION   During execution.

CAUSE   The closing bracket "]" is missing in the programmed expression or instruction.

SOLUTION   Check the syntax of the programmed block.

1107 'The axis does not exist or is not available'

DETECTION   During execution.

CAUSE   The possible causes are:
- An attempt has been made to move an axis that does not exist or is not available in the system or in the channel.
- In a high level statement, an axis has been programmed that does not exist or is not available in the system or in the channel.
- In an axis variable, an axis number has been programmed that does not exist in the system.
- In an axis variable, an axis index has been programmed that does not exist in the channel.

SOLUTION   Verify that the programmed axis exists in the system or channel and that it is available (is not parked).

1108 'Axis programmed twice'

DETECTION   During execution.

CAUSE   With one of the following functions, an axis has been programmed more than once in the same block:
- Axis movement in G0, G1, G2, G3, G8 or G9.
- Threading G33 or G63.
- Instructions #FACE or #CYL.
- Free plane selection, G20.

With functions that imply axis movements, double programming may also be due to having programmed the axis in both Cartesian and Polar coordinates.

SOLUTION   Check the program.

1109   **'Wrong axis index'**

DETECTION	During execution.
CAUSE	In functions G20 and G74, an index must be programmed with the axis name. That index is wrong.
SOLUTION	The axis index must be greater than 0 and must not exceed the maximum number of axes of the system or channel.

1110   **'Values for I, J, K programmed twice'**

DETECTION	During execution.
CAUSE	I, J, K values have been programmed more than once in the block.
SOLUTION	Program I, J, K values only once in the block.

1111   **'Control instructions $ must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Only the block number of a label may be programmed in the same block as a "$" instruction.
	The only exception is programming the instructions $IF and $GOTO in the same block.
SOLUTION	Programning in different blocks.

1112   **'The $IF instruction <condition> can only be followed by $GOTO'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block. The only exception is to program $IF and $GOTO in the same block.
SOLUTION	Program them in different blocks.

1113   **'$ELSE not expected'**

DETECTION	During execution.
CAUSE	The $ELSE instruction has been programmed in a block, but no $IF has been previously programmed.
SOLUTION	Check the program.

1114   **'The $ELSE instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1115   **'$ELSEIF not expected'**

DETECTION	During execution.
CAUSE	The $ELSEIF instruction has been programmed in a block, but no $IF has been previously programmed.
SOLUTION	Check the program.

1116   **'The $ELSEIF <condition> instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1117   **'$ENDIF not expected'**

DETECTION	During execution.
CAUSE	The $ENDIF instruction has been programmed in a block, but no $IF has been previously programmed.
SOLUTION	Check the program.

1118   **'The $ENDIF instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 19 of 128

1119 'The $SWITCH <expression> instruction must be programmed alone in the block'

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1120 '$CASE not expected'

DETECTION	During execution.
CAUSE	The $CASE instruction has been programmed in a block, but no $SWITCH has been previously programmed.
SOLUTION	Check the program.

1121 'The $CASE <expression> instruction must be programmed alone in the block'

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1122 '$DEFAULT not expected'

DETECTION	During execution.
CAUSE	The $DEFAULT instruction has been programmed in a block, but no $SWITCH has been previously programmed.
SOLUTION	Check the program.

1123 'The $DEFAULT instruction must be programmed alone in the block'

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1124 '$ENDSWITCH not expected'

DETECTION	During execution.
CAUSE	The $ENDSWITCH instruction has been programmed in a block, but no $SWITCH has been previously programmed.
SOLUTION	Check the program.

1125 'The $ENDSWITCH instruction must be programmed alone in the block'

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1126 '$FOR: invalid counter variable'

DETECTION	During execution.
CAUSE	An invalid counter has been used in the $FOR control loop.
SOLUTION	Only variables and arithmetic parameters are valid as counters in $FOR control loop.

1127 'The $FOR instruction<condition> instruction must be programmed alone in the block'

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1128 '$FOR: too many characters in the condition'

DETECTION	During execution.
CAUSE	The line that contains the $FOR instruction has more than 5100 characters.
SOLUTION	Make the line that defines the $FOR shorter.

1129 '$ENDFOR not expected'

DETECTION	During execution.
CAUSE	The $ENDFOR instruction has been programmed in a block, but no $FOR has been previously programmed.
SOLUTION	Check the program.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 20 of 128

1130 **'The $ENDFOR instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1131 **'The $WHILE <condition> instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1132 **'$WHILE: too many characters in the condition'**

DETECTION	During execution.
CAUSE	The condition of the control loop $WHILE exceeds the maximum number of characters allowed.
SOLUTION	The maximum number of characters allowed is 5000.

1133 **'$ENDWHILE not expected'**

DETECTION	During execution.
CAUSE	The $ENDWHILE instruction has been programmed in a block, but no $WHILE has been previously programmed.
SOLUTION	Check the program.

1134 **'The $ENDWHILE instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1135 **'The $DO instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1136 **'$ENDDO not expected'**

DETECTION	During execution.
CAUSE	The $ENDDO instruction has been programmed in a block, but no $DO has been previously programmed.
SOLUTION	Check the program.

1137 **'The $ENDDO <expression> instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1138 **'The $BREAK instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Control instructions must be programmed alone in the block.
SOLUTION	Check the program.

1139 **'$BREAK not expected'**

DETECTION	During execution.
CAUSE	The $BREAK instruction has been programmed in a block, but no control loop is open: $IF, $ELSE, $FOR, $WHILE, $DO or $CASE.
SOLUTION	Check all the programmed "$" control loops. The $BREAK instruction is used to end a $CASE or to exit from a $IF, $ELSE, $WHILE, $FOR or $DO loop before it ends.

1140 **'$CONTINUE not expected'**

DETECTION	During execution.
CAUSE	No control loop is open for which the $CONTINUE instruction might make sense.
SOLUTION	Check the sequence of programmed blocks.
	The $CONTINUE instruction is programmed inside a control loop $FOR, $WHILE or $DO to return to its starting point.

1141 'The $CONTINUE instruction must be programmed alone in the block'

DETECTION   During execution.
CAUSE   Control instructions must be programmed alone in the block.
SOLUTION   Check the program.

1142 'The #TIME instruction must be programmed alone in the block'

DETECTION   During execution.
CAUSE   The dwell must be programmed in the same block as the #TIME instruction.
SOLUTION   Check the program.

1146 'The path before a G37 must be linear'

DETECTION   During execution.
CAUSE   The motion block before the tangential entry is not linear.
SOLUTION   Check the program.

1147 'The path after a G38 must be linear'

DETECTION   During execution.
CAUSE   The motion block after the tangential exit is not linear.
SOLUTION   Check the program.

1149 'The programmed G36/G37/G38/G39 cannot be executed'

DETECTION   During execution.
CAUSE   The desired joining path between the first and the last block cannot be carried out
              with the programmed radius.
SOLUTION   Check the value programmed for the radius of the joining path. Check that the joint
              is actually possible between the first and the last block.

1150 'Functions G36/G37/G38/G39 must be followed by a motion block'

DETECTION   During execution.
CAUSE   The second motion block for the joining path is missing.
SOLUTION   Do not program any block between the G function that defines the joining path and
              the second motion block.

1151 'Functions G8/G36/G37/G38/G39 must be preceded by a motion block'

DETECTION   During execution.
CAUSE   The first motion block for the joining path is missing.
SOLUTION   Do not program any block between the G function that defines the joining path and
              the first motion block.

1152 'Too many nested subroutines'

DETECTION   During execution.
CAUSE   The maximum level of nesting allowed by the CNC has been exceeded.

              A subroutine may be called from a main program (or a subroutine), another subroutine
              from this subroutine and so forth. The CNC limits this calls to a maximum of 20 nesting
              levels.
SOLUTION   Redesign the program decreasing the nesting level on calls to subroutines (local and
              global).

1153 'Too many local subroutines defined in the program'

DETECTION   During execution.
CAUSE   The maximum number of local subroutines allowed by the CNC has been exceeded.
              This maximum is 100.
SOLUTION   Redesign the program decreasing the number of local subroutines defined.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 22 of 128

1154 'File name too long'

DETECTION	During execution.
CAUSE	The maximum number of characters allowed for the name of a program or subroutine has been exceeded.

- The name of a program or subroutine may have a maximum of 63 characters.
- The path of a program or subroutine may have a maximum of 120 characters.

When programming the name of a program or subroutine with a path, the maximum number of characters will be the sum of both values.

SOLUTION	Decrease the number of characters of the name of the program or subroutine. Move the program or subroutine to another directory to reduce the number of characters of the path.

1155 'No access to the file'

DETECTION	During execution.
CAUSE	The program or subroutine could not be accessed.
SOLUTION	Check that the files are valid and are not corrupted.

When calling subroutines, check that the name and the path are correct.

If the path of a subroutine is not explicitly indicated in the search statement, it will be done in the following order:

1. Path programmed with the #PATH instruction.
2. Path of the program being executed.
3. Path of machine parameter SUBPATH.

1156 'Main program not found'

DETECTION	During execution.
CAUSE	The main program has not been found.
SOLUTION	Having local subroutines defined in the file, the main program MUST begin with "%name".

1157 'Global subroutine not found'

DETECTION	During execution.
CAUSE	The global subroutine called upon from the program has not been found.
SOLUTION	Check that the name and the path of the subroutine are correct.

If the path of a subroutine is not explicitly indicated in the search statement, it will be done in the following order:

- Path programmed with the #PATH instruction.
- Path of the program being executed.
- Path of machine parameter SUBPATH.

1159 'Name of the local subroutine too long'

DETECTION	During execution.
CAUSE	The maximum number of characters for the name of a local subroutine is 63.
SOLUTION	Check the program.

1160 'Local subroutine not found'

DETECTION	During execution.
CAUSE	The local subroutine to be executed has not been found.
SOLUTION	Check the name of the local subroutine in the calling block is the same as the name that appears in its definition.

The local subroutines are defined at the beginning of the file. Its definition begins with %L and ends with #RET, M17 or M29.

1161 '$ control blocks open'

DETECTION	During execution.
CAUSE	In a subroutine, there is a "$" control block that does not have its corresponding closing instruction.
SOLUTION	Check the programmed control blocks and close them properly.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 23 of 128

1162   'M17/M29/#RET not expected'

DETECTION   During execution.

CAUSE   M17/M29/#RET has been detected as end of program.

SOLUTION   Program M30/M02 as the end of the main program.

              If the error persists, check that all the local subroutines (%L) and global ones end with M17/M29/#RET.

1163   'M30/M02 not expected'

DETECTION   During execution.

CAUSE   An M30 or M02 has been detected as end of subroutine.

SOLUTION   Check that all the local subroutines (%L) and global ones end with M17/M29/#RET.

1164   'Unknown term in mathematical expression'

DETECTION   During execution.

CAUSE   In the mathematical expression a term has been written that is wrong for the control syntax.

SOLUTION   Check all the terms of the expression: variables, parameters, operators, etc.

1165   'Nonexistent variable'

DETECTION   During execution.

CAUSE   The possible causes are:
              • The requested variable does not exist.
              • Syntax error in the name of the variable.
              • The variable is an array and the array index has not been indicated.
              • A general variable has been requested for a particular axis or the other way around.

SOLUTION   Check the program.

1166   'Square root of a negative number'

DETECTION   During execution.

CAUSE   In a mathematical expression, the square root "SQRT" of a negative number has been programmed.

SOLUTION   Check the program.

1167   'Logarithm of a negative number or zero'

DETECTION   During execution.

CAUSE   In a mathematical expression, the logarithm "LOG" or "LN" of a negative number or zero has been requested.

SOLUTION   Check the program.

1168   'Variable index out of range'

DETECTION   During execution.

CAUSE   The index requested for the array variable is wrong.

SOLUTION   The minimum admissible index for an array variable is 1 and the maximum depends on which variable it is. There are particular instances where the index 0 is admitted: G.GS, G.MS, G.LUP1, … G.LUP7, G.LUPACT and MTB.P.

1170   'The #SYNC POS instruction must be programmed alone in the block'

DETECTION   During execution.

CAUSE   Only the block number of a label may be programmed in the same block as an instruction.

SOLUTION   Program them in different blocks.

1171   '# instructions must be programmed alone in the block'

DETECTION   During execution.

CAUSE   Only the block number of a label may be programmed in the same block as a "#" instruction.

              The only exception is the #AXIS instruction that must be programmed in the same block as function G201.

SOLUTION   Programning in different blocks.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 24 of 128

1172   'Instruction not allowed while tool radius compensation is active'

DETECTION   During execution.

CAUSE   An attempt has been made to activate a function that is incompatible with tool radius compensation.

SOLUTION   Cancel tool radius compensation to be able to activate the function.

1173   'The #UNLINK instruction must be programmed alone in the block'

DETECTION   During execution.

CAUSE   Only the block number of a label may be programmed in the same block as a "#" instruction.

SOLUTION   Check the program.

1174   '#LINK: a new coupling (slaving) cannot be defined if a previous one is active'

DETECTION   During execution.

CAUSE   An attempt has been made to activate a second coupling (slaving) without deactivating the first one.

SOLUTION   Check the syntax of the instruction in the programming manual.
- If only the second coupling is desired, deactivate the first one with the #UNLINK instruction before activating it.
- If both couplings are desired, deactivate the first one with the #UNLINK instruction and then activate both at the same time with a single #LINK instruction.

1175   '#LINK: no coupling (slaving) has been defined'

DETECTION   During execution.

CAUSE   No axis coupling (slaving) has been defined with the #LINK command.

SOLUTION   Program the master and slave axes in the #LINK instruction.

1176   '#LINK: the master axis does not belong to the current axis configuration'

DETECTION   During execution.

CAUSE   The possible causes are:
- In the #LINK instruction, a master axis has been programmed that does not exist or is not available in the channel.
- With the #LINK instruction, an attempt has been made to deactivate a coupling whose master axis does not exist or is not available in the channel.

SOLUTION   The master and slave axes must exist in the channel that executes the instruction.

1177   '#LINK: the slave axis does not belong to the current axis configuration'

DETECTION   During execution.

CAUSE   The possible causes are:
- In the #LINK instruction, a slave axis has been programmed that does not exist or is not available in the channel.
- With the #LINK instruction, an attempt has been made to deactivate a coupling whose slave axis does not exist or is not available in the channel.

SOLUTION   The master and slave axes must exist in the channel that executes the instruction.

1178   '#LINK: the slave axis cannot be an axis of the main plane'

DETECTION   During execution.

CAUSE   An attempt has been made to activate a coupling with one of the main three axes as slave.

SOLUTION   Check the program.

1179   '#LINK: The master and slave axes must be of the same type (AXISTYPE)'

DETECTION   During execution.

CAUSE   An attempt has been made to activate a coupling (slaving) where the master and slave axes have different machine parameter AXISTYPE.

SOLUTION   The master and slave axes must have the same machine parameter AXISTYPE.

1180   '#LINK: The master and slave axes must have the same mode (AXISMODE)'

DETECTION   During execution.

CAUSE   An attempt has been made to activate a coupling (slaving) where the master and slave axes have different machine parameter AXISMODE.

SOLUTION   The master and slave axes must have the same machine parameter AXISMODE.

1181 '#LINK: An axis active in G201 cannot be defined as slave'

DETECTION	During execution.
CAUSE	An attempt has been made to define as slave of a coupling an axis that is in additive manual mode with function G201.
SOLUTION	If the coupling is desired, the additive manual mode of the axis may be canceled with function G202.

1182 '#LINK: too many couplings programmed'

DETECTION	During execution.
CAUSE	The number of coupling to be activated with the #LINK instruction exceeds the maximum allowed.
SOLUTION	The maximum number of couplings that can be activated with the #LINK instruction is limited depending on the number of axes of the system (without counting spindles). The limit is the number of axes - 3.

1183 'The #LINK instruction must be programmed alone in the block'

DETECTION	During execution.
CAUSE	Only the block number of a label may be programmed in the same block as a "#" instruction.
SOLUTION	Check the program.

1184 '#LINK: The master and slave axes are the same'

DETECTION	During execution.
CAUSE	An attempt has been made to activate a coupling with the same axis as master and slave.
SOLUTION	The master and slave axes must be different.

1185 '#LINK: An axis cannot be a slave of several masters'

DETECTION	During execution.
CAUSE	An attempt has been made to define an axis as slave of different masters.
SOLUTION	A slave axis has only one master axis.

1186 ''#LINK: A master axis cannot be a slave in another coupling and vice versa'

DETECTION	During execution.
CAUSE	An attempt has been made to define an axis as master in a coupling and as slave in another.
SOLUTION	The axis cannot be master and slave at the same time.

1187 '#AXIS: repeated axis name'

DETECTION	During execution.
CAUSE	The same axis has been programmed more than once in the instruction.
SOLUTION	Check the program.

1188 '"["not expected'

DETECTION	During execution.
CAUSE	Syntax error in the instruction.
SOLUTION	Check the syntax of the instruction in the programming manual.

1189 '#MPG: too many parameters'

DETECTION	During execution.
CAUSE	The instruction has been programmed with more parameters than allowed.
SOLUTION	The #MPG instruction admits a maximum of three parameters. Each one of them represents the distance moved per handwheel pulse in each position of the switch.

1190 '#MPG: Negative or zero handwheel resolutions are not allowed'

DETECTION	During execution.
CAUSE	A negative or zero value has been programmed for handwheel resolution.
SOLUTION	The movement distance per handwheel pulse must be positive and other than zero.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 26 of 128

1191 '#INCJOG: Negative or zero incremental jog distances are not allowed'

DETECTION   During execution.
CAUSE   A negative or zero value has been programmed for incremental jog distance.
SOLUTION   The value of the incremental movement of the axis in each position of the switch must be a positive value and other than zero.

1192 '#INCJOG: Negative or zero incremental jog feedrates are not allowed'

DETECTION   During execution.
CAUSE   A negative or zero value has been programmed for incremental jog feedrate.
SOLUTION   The value of the axis feedrate in each position of the switch must be a positive value and other than zero.

1193 '#CONTJOG/#INCJOG: programmed feedrate out of range'

DETECTION   During execution.
CAUSE   The value of the programmed axis feedrate is too high.
SOLUTION   Program smaller values.

1194 '#INCJOG: too many parameters'

DETECTION   During execution.
CAUSE   The instruction has been programmed with more parameters than allowed.
SOLUTION   The #INCJOG instruction admits a maximum of five groups of parameters. Each of them represents the axis feedrate and movement for each position of the switch in incremental jog.

1195 '#CONTJOG: too many parameters'

DETECTION   During execution.
CAUSE   The instruction has been programmed with more parameters than allowed.
SOLUTION   The #CONTJOG instruction only admits one parameter that represents the axis feedrate when the switch is in continuous jog.

1196 '#CONTJOG: Negative or zero continuous jog feedrates are not allowed'

DETECTION   During execution.
CAUSE   A negative or zero value has been programmed for continuous jog feedrate.
SOLUTION   The value of the axis feedrate must be a positive value and other than zero.

1197 '#SET OFFSET: positive lower offset'

DETECTION   During execution.
CAUSE   A positive value has been programmed for the lower limit of the axis travel.
SOLUTION   The lower offset value must be negative or zero.

1198 '#SET OFFSET: negative limit out of range'

DETECTION   During execution.
CAUSE   The programmed value for the axis limit is too low.
SOLUTION   Program greater values.

1199 '#SET OFFSET: negative upper offset'

DETECTION   During execution.
CAUSE   A negative value has been programmed for the upper limit of the axis travel.
SOLUTION   The upper offset value must be positive or zero.

1200 '#SET OFFSET: positive limit out of range'

DETECTION   During execution.
CAUSE   The programmed value for the axis limit is too high.
SOLUTION   Program smaller values.

1201 '#SET OFFSET: zero upper and lower offsets'

DETECTION   During execution.
CAUSE   It informs that the travel limits for the axes are zero.
SOLUTION   Check the program.

1203 'The #SET IPOPOS instruction must be programmed alone in the block'

DETECTION   During execution.

CAUSE   Only the block number of a label may be programmed in the same block as a "#" instruction.

SOLUTION   Check the program.

1204 'Nonexistent instruction or programmed wrong'

DETECTION   During execution.

CAUSE

SOLUTION   Check the syntax of the block.

1205 '#CALL AX/#SET AX: unknown offset type'

DETECTION   During execution.

CAUSE   The type of offset programmed in the instruction does not exist.

SOLUTION   The valid offset types are ALL, LOCOF, FIXOF, TOOLOF, ORGOF, MEASOF, MANOF.

1206 '"," expected'

DETECTION   During execution.

CAUSE   "," expected in the programmed instruction or function.

SOLUTION   Check the syntax of the block.

1209 'Axis index out of range'

DETECTION   During execution.

CAUSE   The possible causes are:
1. #CALL AX/#SET AX: The position for an axis is not correct. Maybe, the index indicated exceeds the maximum or there is no room for the axis beyond the last one when no position is indicated.
2. An axis name has been programmed with the wrong wild character.

SOLUTION   The solutions are:
1. #CALL AX/#SET AX: An axis may be placed in any unoccupied position between 1 and a number equal to the maximum number of axes allowed by the system plus the maximum number of spindles.
2. The possible wild characters are @1 to @6 and @SM.

1210 '#CALL AX/#SET AX: repeated axis name'

DETECTION   During execution.

CAUSE   The same axis has been programmed twice.

SOLUTION   Check the names, the same axis cannot be repeated in a channel.

1211 '#CALL AX/#SET AX: repeated axis index'

DETECTION   During execution.

CAUSE   An attempt has been made to place two axes in the same position.

SOLUTION   Check the configuration of the axes desired for the channel, two axes cannot be in the same position.

1213 '#CALL AX/#SET AX: not allowed when G63 is active'

DETECTION   During execution.

CAUSE   #CALL AX cannot be programmed if function G63 is active.

SOLUTION   Deactivate the threading G63 before modifying the configuration of the axes.

1214 '#CALL AX/#SET AX: too many axes required'

DETECTION   During execution.

CAUSE   Maybe, too many axes have been requested or they are being placed in higher positions than the maximum for a channel.

SOLUTION   Do not try to exceed in a channel the maximum number of axes of the system, the valid positions for the axes are between 1 and a value equal to the maximum number of axes allowed by the system plus the maximum number of spindles.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 28 of 128

1215 **'The #CALL AX/#SET AX instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Only the block number of a label may be programmed in the same block as a "#" instruction.
SOLUTION	Check the program.

1216 **'#CALL AX/#CAX: axis name being used'**

DETECTION	During execution.
CAUSE	The name programmed in the instruction for the spindle working as C axis is already being used by another axis of the channel.
SOLUTION	

1217 **'#CALL AX: index being used'**

DETECTION	During execution.
CAUSE	The position indicated for an axis in #CALL AX is occupied by another axis.
SOLUTION	Check the configuration of the axes desired for the channel, two axes cannot be in the same position. An axis may be placed in any unoccupied position between 1 and a number equal to the maximum number of axes plus the maximum number of spindles allowed by the system.

1218 **'The #FREE AX instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Only the block number of a label may be programmed in the same block as a "#" instruction.
SOLUTION	Check the syntax of the instruction in the manual.

1219 **'"," or "]" expected'**

DETECTION	During execution.
CAUSE	Syntax error in the instruction.
SOLUTION	Check the syntax of the instruction in the programming manual.

1220 **'#FREE AX: an active axis cannot be eliminated in manual mode'**

DETECTION	During execution.
CAUSE	The axes involved in the active function G201 (additive manual) cannot be eliminated from the channel.
SOLUTION	To eliminate one of these axes, first cancel the G201 or when activating it previously do not include this axis.

1221 **'The #SET AX instruction must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	Only the block number of a label may be programmed in the same block as a "#" instruction.
SOLUTION	Check the syntax of the instruction in the manual.

1222 **'#COMMENT END not expected'**

DETECTION	During execution.
CAUSE	#COMMENT END has been programmed without programming #COMMENT BEGIN first.
SOLUTION	Check the program.

1223 **'End-of-file character inside the comment block'**

DETECTION	During execution.
CAUSE	There is a #COMMENT BEGIN block without its corresponding #COMMENT END.
SOLUTION	Verify that each #COMMENT BEGIN block has its corresponding #COMMENT END in the program.

1224 **'Operator unknown or missing'**

DETECTION	During execution.
CAUSE	In the command to write a variable or parameter, an operator is unknown or missing.
SOLUTION	The valid assignment operators are "=", "+=", "-=", "*=", "/=".

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 29 of 128

1225 'Division by zero'

DETECTION	During execution.
CAUSE	An operation has been programmed whose execution involves dividing by zero.
SOLUTION	It is only possible to divide by numbers other than zero. When working with parameters, in the program history, that parameter may have taken the value of zero. Verify that the parameter does not reach the operation with that value (0).

1226 'Face axis (FACEAXIS) missing in the active plane for radius compensation'

DETECTION	During execution.
CAUSE	One of the axes of the active plane must be defined in the machine parameters as FACEAXIS type.
SOLUTION	In the machine parameters, define one of the axes as FACEAXIS.

1227 'Longitudinal axis (LONGAXIS) missing in the active plane for radius compensation'

DETECTION	During execution.
CAUSE	One of the axes of the active plane must be defined in the machine parameters as LONGAXIS type.
SOLUTION	In the machine parameters, define one of the axes as LONGAXIS.

1233 'The programmed zero offset exceeds the data range'

DETECTION	During execution.
CAUSE	A zero offset has been programmed with a value greater than the maximum established.
SOLUTION	Check the program.

1236 'Macro name too long'

DETECTION	During execution.
CAUSE	The number of characters of the name of the macro exceeds the maximum allowed.
SOLUTION	The maximum is 30 characters.

1237 '"\" expected in the text associated with the macro'

DETECTION	During execution.
CAUSE	In the replacement text of the macro, a text has been written between quotes without being preceded by the "\" character.
SOLUTION	The quotes must be preceded by the "\" character within the replacement text of the macro. A clear case of joining macros is "macro" = " \"macro1\" \"macro2\" ".

1238 'Replacement text of the macro too long'

DETECTION	During execution.
CAUSE	The number of characters of the replacement text of the macro exceeds the maximum allowed.
SOLUTION	The maximum is 140 characters.

1239 'Too many macros'

DETECTION	During execution.
CAUSE	The maximum number of macros that may be defined in a program has been exceeded.
SOLUTION	The maximum is 50 macros.

1240 'Nonexistent axis'

DETECTION	During execution.
CAUSE	An attempt has been made to use a macro that has not been defined previously.
SOLUTION	Define the macro as "macro" = "replacement text".
	Check the syntax of the macros in the programming manual.

1241 'The replacement text of the macro is missing'

DETECTION	During execution.
CAUSE	It informs that the macro has been assigned an empty string of characters.
SOLUTION	Associate the proper replacement text with the macro according to the functionality it should have. The replacement text must be written between quotes in the definition of the macro.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 30 of 128

1244   'Face axis close to the center: the spindle speed in G96 has been limited'

DETECTION	During execution.
CAUSE	While working at Constant Surface Speed, the spindle speed has been limited. The limitation is due to the proximity of the face axis to the rotating center.
SOLUTION	Do not bring the axis so close to the center, increase the maximum speed allowed or accept this limitation.

1245   'G96: no face axis has been defined in the active plane'

DETECTION	During execution.
CAUSE	None of the axes of the active plane has been defined as face axis.
SOLUTION	Set machine parameter of the face axis: FACEAXIS = Yes

1246   'Threading is not possible with feedrate in G95'

DETECTION	During execution.
CAUSE	An attempt has been made to execute a rigid tapping (G63) while function G95 is active (feedrate in mm/rev. or inch/rev.).
SOLUTION	Program the axis feedrate using function G94 (mm/min. or inches/min.).

1247   'Threading is not possible while G96 is active'

DETECTION	During execution.
CAUSE	An attempt has been made to execute a rigid tapping (G63) while function G96 is active (constant surface speed).
SOLUTION	Cancel function G96 with function G97.

1248   'Threading and G192 not allowed in the same block'

DETECTION	During execution.
CAUSE	G63 (rigid tapping) and G192 (constant surface speed limitation) cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1249   'Gear change is not possible while G96 is active'

DETECTION	During execution.
CAUSE	An attempt has been made to change the spindle gear using function G112 while function G96 is active.
SOLUTION	Cancel function G96 to make the spindle gear change.

1251   'Manual mode is not possible while G96 is active'

DETECTION	During execution.
CAUSE	An attempt has been made to switch to manual mode while function G96 is active.
SOLUTION	Cancel function G96 to be able switch to manual mode.

1252   '#FREE AX: The face turning axis cannot be eliminated while G96 is active'

DETECTION	During execution.
CAUSE	Using the #FREE AX command and being constant surface speed active, the face axis (FACEAXIS) cannot be eliminated from the channel.
SOLUTION	Cancel the constant surface speed using function G97 to eliminate the axis.

1254   'G192 and M19 not allowed in the same block'

DETECTION	During execution.
CAUSE	M19 (spindle orientation) and G192 (constant surface speed limitation) cannot be programmed in the same block.
SOLUTION	Program them in different blocks.

1255   'Negative acceleration percentage'

DETECTION	During execution.
CAUSE	Using function G130, a negative acceleration percentage has been programmed.
SOLUTION	The percentage of acceleration must be equal to or greater than zero.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 31 of 128

1256   'Acceleration percentage out of range'

DETECTION	During execution.
CAUSE	The acceleration percentage programmed with function G130 is too high.
SOLUTION	The maximum value allowed for the acceleration percentage is 2147483646.

1257   'The leadscrew pitch has been programmed twice'

DETECTION	During execution.
CAUSE	Using function G33, the thread pitch has been programmed more than once in the block.
SOLUTION	Define the thread pitch only once in the block.

1258   'Leadscrew pitch equal to zero'

DETECTION	During execution.
CAUSE	The thread pitch programmed with function G33 is zero.
SOLUTION	Program the thread pitch with letters I, J, K being each one associated with the axes X, Y, Z respectively.
	Check the syntax of the instruction in the programming manual.

1259   'Leadscrew pitch out of range'

DETECTION	During execution.
CAUSE	The thread pitch value programmed with function G33 is too high.
SOLUTION	Program a smaller value for the thread pitch.

1261   'Unknown kinematics type'

DETECTION	During execution.
CAUSE	An attempt has been made to activate #RTCP, or #TLC, or #CS/ACS in mode 6 without having a kinematics active.
SOLUTION	First activate the kinematics and then the transformation.
	#KIN ID [ ]
	#RTCP ON or #TLC ON [ ] or #CS/ACS ON [MODE6, , , ,]

1262   'The group of the axes is not enough for the transformation'

DETECTION	During execution.
CAUSE	The coordinate transformation #RTCP, or #TLC, or #CS/ACS needs a number of active axes and they must be the first ones of the channel. This number of active axes must be between 3 and 5 depending on the type of kinematics active.
SOLUTION	Activate the axes on to which to apply the transformation in the first positions of the channel using a #SET AX [, , , ,]

1263   'Rotary axis (axes) missing for the transformation'

DETECTION	During execution.
CAUSE	The instruction #TOOL ORI has been programmed, but there is no rotary axis to place the tool perpendicular to the defined incline plane.
SOLUTION	Either do not program the instruction #TOOL ORI or activate the kinematics that allows placing the tool perpendicular to the defined incline plane.

1264   'Programming not allowed while CS/ACS is active'

DETECTION	During execution.
CAUSE	An attempt has been made to execute one of the following functions while CS or ACS is active.
	• Home search (G74).
	• Modify software limits (G198 - G199).
	• Execute the probing cycle #PROBE1.
	• #OSC instruction.
	• #LINK instruction.
	• #ECS instruction.
SOLUTION	First cancel the active incline plane and then execute the desired instructions. Finally, activate the incline plane again.

FAGOR

CNC8070
REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 32 of 128

1265   'Programming not allowed while RTCP/TLC is active'

DETECTION	During execution.
CAUSE	An attempt has been made to execute one of the following functions while RTCP or TLC is active.

  • Home search (G74).
  • Modify software limits (G198 - G199).
  • #OSC instruction.
  • #KIN ID instruction.

SOLUTION	First cancel the active incline plane, then execute the desired instructions and finally activate the incline plane again.

1266   'The TLC feature is deactivated using the instruction #TLC OFF'

DETECTION	During execution.
CAUSE	While #TLC is active, a #TLC instruction other than #TLC OFF has been programmed.
SOLUTION	To program #TLC ON again to change the value programmed earlier, it must be canceled first using #TLC OFF.

1268   '#CS ON/#ACS ON: syntax error'

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the syntax of the instruction in the programming manual.

1269   '#CS ON/#ACS ON: the programmed angle is not valid'

DETECTION	During execution.
CAUSE	Programmed angle is not valid.
SOLUTION	The angle programmed in the instruction must be within ± 360º.

1270   'The coordinate transformation cannot be calculated'

DETECTION	During execution.
CAUSE	An attempt has been made to make a coordinate transformation from part to machine or from machine to part that is not solved.
SOLUTION	Cancel the transformation, change the position to be accessed and activate the transformation again.

1271   'The axes of the active kinematics can neither be excluded nor modified'

DETECTION	During execution.
CAUSE	Having an active kinematics, an attempt has been made to change the axes that affect that kinematics using the instructions #CALL AX, #SET AX, #FREE AX.
SOLUTION	Cancel the kinematics before changing the axes of the channel. #KIN ID [0].

1272   'The axes of the active transformation can neither be excluded nor modified'

DETECTION	During execution.
CAUSE	Having an active incline plane, an attempt has been made to change one of the first three axes that affect that transformation using the instructions #CALL AX[ ], #SET AX[ ], #FREE AX[ ].
SOLUTION	Cancel the transformation of the incline plane before changing the axes of the channel. #CS/ACS OFF ALL

1277   'The resulting zero offset exceeds the data range'

DETECTION	During execution.
CAUSE	The zero offset calculated from the coordinates programmed with function G92 is too large.
SOLUTION	Check the program.

1278   **'G131/G133: wrong value'**

DETECTION   During execution.

CAUSE   The possible causes are:

- Function G131 has been used to program the wrong value for the percentage of acceleration to be applied to the axes.
- Function G133 has been used to program the wrong value for the percentage of jerek to be applied to the axes.

SOLUTION   The percentage of acceleration or jerk to apply to the axes must be positive and less than or equal to 100%.

1279   **'" expected'**

DETECTION   During execution.

CAUSE   Quote marks are missing in the programmed expression or instruction.

SOLUTION   Check the syntax of the programmed block.

1281   **'The number of parameters and format indicators do not match'**

DETECTION   During execution.

CAUSE   The number of format indicators (%D or %d) appearing in the #MSG or #WARNING instruction does not match the number of parameters to be displayed in the message.

SOLUTION   Check the syntax of the instruction in the programming manual.

1282   **'Message too long'**

DETECTION   During execution.

CAUSE   A message of more than 69 characters has been programmed in the instruction #MSG, #ERROR, #WARNING.

SOLUTION   Reduce the number of characters in the message considering that the maximum is 69. This limit also includes the characters that would replace the possible indicators.

1283   **'Too many format indicators'**

DETECTION   During execution.

CAUSE   More than 5 format indicators (%D ó %d) have been programmed in the instruction #MSG, #ERROR, #WARNING.

SOLUTION   Reduce the number of format indicators programmed.

1284   **'Arithmetic expression expected'**

DETECTION   During execution.

CAUSE   Format indicators (%d - %D) have been programmed in the instruction #MSG, #WARNING, #ERROR, but the list of values to be displayed has not been programmed.

SOLUTION   Program the list of values to be displayed in the instruction.

Check the syntax of the instruction in the programming manual.

1285   **'Tool radius written twice'**

DETECTION   During execution.

CAUSE   The writing of the tool radius has been programmed more than once in the same block.

SOLUTION   Program only once in the block the writing of the tool radius.

1286   **'Tool length written twice'**

DETECTION   During execution.

CAUSE   The writing of the tool length has been programmed more than once in the same block.

SOLUTION   Program only once in the block the writing of the tool length.

1287   **'"[" expected'**

DETECTION   During execution.

CAUSE   The opening bracket is missing in the programmed expression or instruction.

SOLUTION   Check the syntax of the programmed block.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 34 of 128

1288 **'Too many parameters programmed in the instruction'**

DETECTION   During execution.
CAUSE   The instruction has not been programmed correctly.
SOLUTION   Check the syntax of the instruction in the programming manual.

1290 **'I, J, K coordinates programmed wrong'**

DETECTION   During execution.
CAUSE   The probable causes are:
              • The programmed values for the center of the circular interpolation, Polar origin or center of rotation of the coordinate system are too high.
              • Wrong values have been programmed for the coordinates of the center of the circular interpolation being G264 active.
SOLUTION   Check the program.

1291 **'No more S functions allowed'**

DETECTION   During execution.
CAUSE   The maximum number of S functions allowed in the same block has been exceeded.
SOLUTION   The maximum number of S functions allowed in the same block is 4.

1292 **'M function programmed twice'**

DETECTION   During execution.
CAUSE   An M function that is not related with the spindle has been programmed more than once in the same block.
SOLUTION   Program them in different blocks.

1293 **'H function programmed twice'**

DETECTION   During execution.
CAUSE   An H function has been programmed more than once in the same block.
SOLUTION   Program them in different blocks.

1301 **'The tool length transformation exceeds the valid numeric format'**

DETECTION   During execution.
CAUSE   The maximum numeric format has been exceeded in the tool length transformation.
SOLUTION   Modify the values of the transformation or those of the tool.

1302 **'Wrong character in the name'**

DETECTION   During execution.
CAUSE   The possible causes are:
              1. Wrong character in the name of a label.
              2. Wrong character in the name of a subroutine.
              3. Wrong character in the name of an external variable.
SOLUTION   Valid characters in each one of the previous cases:
              1. All except quotes, "]" and tab.
              2. All except quotes, blank space and tab. All characters after the "(" will be considered as part of the comment.
              3. letters, digits and the "_" character.

1303 **'Variable name too long'**

DETECTION   During execution.
CAUSE   Defining "part" as each set of characters separated by "." in the name of a variable, the error comes up when a part has more than 13 characters.
SOLUTION   Shorten the name of the variable.

1304 **'Wrong spindle speed'**

DETECTION   During execution.
CAUSE   The programmed spindle speed value is too low.
SOLUTION   The value of the spindle speed must be higher than 1.6 10E-7 rpm.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 35 of 128

1305 'Programming not allowed while #MCS is active'

DETECTION	During execution.
CAUSE	One of the following functions has been programmed while #MCS is active:

- Zero offsets (G54-G59, G159, G92, G158, G101, G102, G53) on/off
- Fixtures on/off ("V.G.FIX" variable).
- Mirror (G11/G12/G13/G14) image on/off.
- Programming in radius/diameters (G151/G152).
- Activate incremental programming (G91).
- Programming in mm/inches (G70/G71).
- Scaling factor (G72).
- Movement in G0, G1, G2, G3, G8 or G9 in Polar coordinates.
- Threading G63 or G33 in Polar coordinates.
- Polar origin (G30).
- Pattern rotation (G73).
- Instructions #FACE, #CYL and #RTCP.

SOLUTION	Check the program.

1306 'The kinematics cannot be changed while tool radius compensation is active'

DETECTION	During execution.
CAUSE	An attempt has been made to change the active kinematics while tool radius compensation (G41/G42) is active.
SOLUTION	Cancel tool radius compensation before changing the active kinematics.

1308 'An axis of the active transformation cannot be a slave'

DETECTION	During execution.
CAUSE	The #LINK instruction has been used to define an axis involved in the active kinematics transformation as a slave of a coupling.
SOLUTION	Cancel the kinematics in order to be able to activate the coupling with this axis as a slave.
	The axis involved in the active kinematics can be master of a coupling.

1309 'File name expected'

DETECTION	During execution.
CAUSE	It informs that no program has been selected for execution.
SOLUTION	Select the program to be executed.

1310 'Program line too long'

DETECTION	During execution.
CAUSE	The number of characters of the block programmed in the #EXBLK instruction exceeds the maximum allowed.
SOLUTION	The maximum number of characters allowed for the block programmed with the #EXBLK instruction is 128.

1311 'Measurement offset not included in programmed axis (axes)'

DETECTION	During execution.
CAUSE	An attempt has been made to exclude the measurement offset (G102) to an axis that has no measurement offset included (G101).
SOLUTION	It does not make sense to execute function G102 for an axis for which function G101 has not been executed previously.

1314 '#CS ON/#ACS ON: wrong identifier'

DETECTION	During execution.
CAUSE	The CS/ACS type identifier programmed in the instruction is a wrong value.
SOLUTION	The CS/ACS type identifier to be activated must be a positive value, other than zero and less than or equal to 5.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 36 of 128

1315 **'#CS ON/#ACS ON: undefined system'**

DETECTION | During execution.
CAUSE | The #CS ON/#ACS ON instruction has been programmed without parameters, but no transformation has been defined or activated previously.
SOLUTION | When programming the #CS ON/#ACS ON instruction without parameters, the CNC tries to activate the transformation stored last. In this case, it is not possible because there is no transformation stored.
 | Program #CS/#ACS with the proper parameters to activate the desired transformation.
 | Check the syntax of the instruction in the programming manual.

1316 **'#CS/#ACS DEF: parameters missing'**

DETECTION | During execution.
CAUSE | The parameters required for this instruction have not been programmed.
SOLUTION | The #CS/#ACS DEF instruction requires programming the mode, the translation vector and the rotation angles.
 | Check the syntax of the instruction in the programming manual.

1318 **'#CS ON/#ACS ON: No changes allowed with the active coordinate system'**

DETECTION | During execution.
CAUSE | An attempt has been made to change the parameters of a transformation that is active.
SOLUTION | Using the #CS/#ACS DEF instruction, it is not possible to change the parameters of transformations that have been defined earlier and have not been activated.
 | It is possible to modify the parameters of the ones already defined, but not activated yet.

1319 **'Nesting of #CS ON/#ACS ON instructions exceeded'**

DETECTION | During execution.
CAUSE | The maximum number of transformation that could be overlapped has been exceeded.
SOLUTION | Up to a maximum of 10 #CS/#ACS transformation may be overlapped.

1320 **'Too many labels'**

DETECTION | It does not come up in the current version. Check the program.
CAUSE | More than 128 labels of the "Nxxx:" type have been defined in the program. or of the " [xxx]" type.
SOLUTION | Remove labels until the number of each type is lower than 128.

1321 **'Label name too long'**

DETECTION | During execution.
CAUSE | A label name has been written with more than 15 characters.
SOLUTION | Reduce the number of characters in the label names.

1322 **'Label defined several times'**

DETECTION | During execution.
CAUSE | The same label has been defined several times in different points of the program.
SOLUTION | Eliminate the repeated labels.

1323 **'$GOTO: Wrong label'**

DETECTION | During execution.
CAUSE | A label can only be defined with a string of characters between brackets or with the "N" character followed by a positive number smaller than 2147483646.
SOLUTION | Remove labels until the number of each type is lower than 128.

1324 **'Undefined label'**

DETECTION | During execution.
CAUSE | An undefined label has been programmed in a $GOTO or #RPT command.
SOLUTION | Define the jump label in some point of the program.

1325 'Block number defined several times'

DETECTION   During execution.
CAUSE   The same block number "N" has been defined several times in different points of the program.
SOLUTION   Do not repeat the block number.

1326 'Wrong value to be assigned to a variable'

DETECTION   During execution.
CAUSE   The value assigned to a variable is too high.
SOLUTION   Check the program.

1327 'Spindle positioning (orienting) speed programmed twice'

DETECTION   During execution.
CAUSE   The spindle orientation speed (M19) has been programmed more than once in the block.
SOLUTION   Program the positioning speed only once in the block.

1328 '$FOR instruction without $ENDFOR'

DETECTION   During execution.
CAUSE   The $FOR instruction has been programmed in a block, but no $ENDFOR has been programmed afterwards.
SOLUTION   Check the program.

1330 'Mirror image programmed wrong'

DETECTION   During execution.
CAUSE   Function G14 (mirror image) has been programmed wrong.
SOLUTION   Function G14 can activate (factor -1) or cancel (factor 1) mirror imaging for each axis.

1331 '#TANGFEED RMIN :negative radius not allowed'

DETECTION   During execution.
CAUSE   #TANGFEED RMIN requires a radius greater than 0.
SOLUTION   Check the program.

1332 '#TOOL AX: orientation +/- expected after designating the axis'

DETECTION   During execution.
CAUSE   The tool orientation has not been programmed.
SOLUTION   Check the syntax of the instruction in the programming manual.

1333 'Change of the first and/or second axis of the plane while tool radius compensation is active'

DETECTION   During execution.
CAUSE   The possible causes are:
            • While tool radius compensation was active, an attempt has been made to change the first or second axis of the plane with the instruction #CALL AX or #SET AX.
            • While tool radius compensation was active, an attempt has been made to free the first or second axis of the plane with the instruction #FREE AX.
SOLUTION   Cancel the tool radius compensation to execute these actions on the first or second axis of the plane.

1334 'G200: does not allow movement in the same block'

DETECTION   During execution.
CAUSE   An axis movement has been programmed in the same line as function G200.
SOLUTION   Programming in different lines.

1336 'Wrong configuration: two CAXIS axis'

DETECTION   During execution.
CAUSE   Both axes programmed in the #FACE/#CYL instruction are C axes.
SOLUTION   Only one of the two axes may be a C axis. In other words, have machine parameter CAXIS = Yes.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 38 of 128

1337 'No CAXIS has been defined'

DETECTION	During execution.
CAUSE	None of the axes programmed in the #FACE/#CYL instruction is a C axis.
SOLUTION	One of the two axes programmed in the instruction must be a C axis. In other words, have machine parameter CAXIS = Yes.

1339 'The selection has no effect'

DETECTION	During execution.
CAUSE	It informs that the programmed instruction has no effect because it was programmed exactly like the previous one and, therefore, it is already active.
SOLUTION	Check the program.

1340 'The deselection has no effect'

DETECTION	During execution.
CAUSE	It informs that a #CAX OFF has been programmed when the spindle is not working as C axis. In other words, when a #CAX has not been programmed previously.
SOLUTION	Check the program.

1342 '#CAX OFF not allowed if a transformation is active'

DETECTION	During execution.
CAUSE	The C axis cannot be deactivated while #RTCP or #TLC is active.
SOLUTION	Check the program.

1343 '#FACE OFF not allowed with the type of kinematics active'

DETECTION	During execution.
CAUSE	The #FACE OFF instruction has been programmed without previously activating the machining operation on the face of the part using the #FACE instruction.
SOLUTION	Check the program.

1344 'No plane change allowed while machining the side of the part'

DETECTION	During execution.
CAUSE	A function G17-G20 has been programmed while side machining is active.
SOLUTION	Check the program.

1345 'G20: Axes programmed wrong'

DETECTION	During execution.
CAUSE	Using function G20 (plane change) the first two axes of the plane (parameters 1 and 2) have been programmed wrong.
SOLUTION	The two axes must be different and they must be the main axes of the configuration.

1347 '#CYL OFF not allowed with the type of kinematics active'

DETECTION	During execution.
CAUSE	The #CYL OFF instruction has been programmed without previously activating the machining operation on the side of the part using the #CYL instruction.
SOLUTION	Check the program.

1348 '#CYL: wrong radius'

DETECTION	During execution.
CAUSE	The possible causes are: 1. A negative or zero radius has been programmed in the #CYL instruction. 2. In the case of a variable radius, an attempt has been made to go through the developing cylinder generating a zero radius.
SOLUTION	The solutions are: 1. Program a positive radius other than zero in the #CYL instruction. 2. Do not go through the center of the developing cylinder.

1349 'Negative axis coordinate when activating #FACE'

DETECTION	During execution.
CAUSE	The linear axis that is part of the face C axis transformation is positioned in the negative portion with respect to the rotating axis.
SOLUTION	The possible solutions are: • Position the axis in the positive portion with respect to the rotation center when activating the face C axis. • Define the type of transformation for the face C axis where it is possible to go through the rotation center by changing the general parameter of the channel ALINGC = NO.

1350 'Wrong character between the #VAR/#ENDVAR instructions'

DETECTION	During execution.
CAUSE	A wrong character has been programmed in some block between these instructions.
SOLUTION	Between these instructions, only the declaration of user variables (separated by commas if there are several in the same line) or the programming of the block number are allowed.

1351 '#VAR/#ENDVAR/#DELETE: variable type not allowed'

DETECTION	During execution.
CAUSE	An attempt has been made to define or delete a variable that is not a user variable, V.P or V.S.
SOLUTION	Define or delete user variables only.

1352 '#VAR/#ENDVAR: the variable defined already exists'

DETECTION	During execution.
CAUSE	A user variable has been defined that has been defined before.
SOLUTION	Check the program.

1353 'Too many values to initialize the array'

DETECTION	During execution.
CAUSE	When initializing an user array variable, it initializes more positions than it has.
SOLUTION	Check the program.

1354 'Error when reading the variable'

DETECTION	During execution.
CAUSE	An error occurred when reading a variable.
SOLUTION	Check the program.

1355 'The variable cannot be deleted'

DETECTION	During execution.
CAUSE	An attempt has been made to delete a CNC variable.
SOLUTION	Only user defined variables may be deleted (prefixes P and S).

1356 'Variable or parameter expected'

DETECTION	During execution.
CAUSE	$IF EXIST instruction programmed wrong.
SOLUTION	The $IF EXIST control instruction only allows arithmetic parameters or variables.

1357 '#DELETE: wrong character'

DETECTION	During execution.
CAUSE	A wrong character has been programmed in the same block as the instruction.
SOLUTION	The instruction must be programmed alone in the block or next to the block number. Only user variables may be deleted , V.P or V.S (separated by commas if there are several in the same line) that have been defined before.

1358 '#DELETE: the variable to be deleted does not exist'

DETECTION	During execution.
CAUSE	An attempt has been made to delete a user variable that has been deleted before.
SOLUTION	Check the program.

1360   'G33/G63/G95/G96/G97 not allowed while the C axis is active'

DETECTION	During execution.
CAUSE	A G33/G63/G95/G96/G97 function has been programmed while the C axis was active.
SOLUTION	The C axis may be deactivated with the #CAX OFF instruction.

1363   'Wrong declaration of array variables'

DETECTION	During execution.
CAUSE	User variables (V.P, V.S) must be declared between #VAR/#ENDVAR instructions.
SOLUTION	Check the program.

1364   'Too many array variable indexes'

DETECTION	During execution.
CAUSE	A multi-dimensional array user variable has been defined with more than 4 dimensions.
SOLUTION	Check the program.

1365   'Negative spindle speed not allowed'

DETECTION	During execution.
CAUSE	A negative spindle speed has been programmed in the block.
SOLUTION	The programmed spindle speed must always be positive, except with function G63.

1367   'A gear change and a spindle movements cannot be simultaneous'

DETECTION	During execution.
CAUSE	An M function for spindle movement and function G112 (parameter set change) have been programmed in the same block.
SOLUTION	Program them in different blocks.

1368   'The circle's center and radius cannot be programmed at the same time'

DETECTION	During execution.
CAUSE	The center and the radius of a circular interpolation has been programmed.
SOLUTION	In a circular interpolation, one must program the coordinate of the last point and the radius or the center of the circle.

1369   '#HSC: programming not allowed'

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the syntax of the instruction in the programming manual.

1370   '#HSC: double programming'

DETECTION	During execution.
CAUSE	The activation or deactivation of HSC has been programmed twice in the block.
SOLUTION	Program only #HSC ON or #HSC OFF in the block.

1374   'M02/M30 expected'

DETECTION	During execution.
CAUSE	M02 or M30 has not been programmed at the end of the main program.
SOLUTION	Check the program.

1375   'M17/M29/#RET expected'

DETECTION	During execution.
CAUSE	M17, M29 or #RET has not been programmed at the end of the subroutine.
SOLUTION	Check the program.

1376   'No default name has been defined for the C axis'

DETECTION	During execution.
CAUSE	The #CAX instruction has been programmed with no parameters, the name given to the master spindle of the channel to work as C axis is the one indicated by machine parameter CAXIS. The error occurs when that machine parameter does not indicate any name.
SOLUTION	Indicate in machine parameter CAXIS the name used for the spindle when it works as C axis.

1377   'Parameter written with wrong index'

DETECTION	During execution.
CAUSE	An attempt has been made to write an arithmetic parameter that does not exist or is write protected.
SOLUTION	The number of arithmetic parameters available at the CNC is set by the following machine parameters: • MINLOCP-MAXLOCP for local parameters. • MINGLBP-MAXGLBP for global parameters. • MINCOMP-MAXCOMP for common parameters. The write-protected global parameters are those defined by machine parameters ROPARMIN - ROPARMAX.

1378   'Parameter read with wrong index'

DETECTION	During execution.
CAUSE	An attempt has been made to read a nonexistent arithmetic parameter.
SOLUTION	The number of arithmetic parameters available at the CNC is set by the following machine parameters. • MINLOCP-MAXLOCP for local parameters. • MINGLBP-MAXGLBP for global parameters. • MINCOMP-MAXCOMP for common parameters.

1380   'Canned cycle programmed wrong'

DETECTION	During execution.
CAUSE	In a block, nothing can be programmed after the parameters of a canned cycle.
SOLUTION	Program them in different blocks.

1381   'Nonexistent canned cycle'

DETECTION	During execution.
CAUSE	The programmed canned cycle does not exist.
SOLUTION	Refer to the canned cycles in the programming manual.

1382   'Parameter not allowed in canned cycle'

DETECTION	During execution.
CAUSE	One of the programmed parameters is not allowed for that canned cycle.
SOLUTION	Check the syntax of the cycle in the programming manual.

1383   'Mandatory parameter not programmed in canned cycle'

DETECTION	During execution.
CAUSE	One of the parameters required for that canned cycle has not been programmed.
SOLUTION	Check the syntax of the cycle in the programming manual.

1384   'M function not allowed with movement'

DETECTION	During execution.
CAUSE	A movement and an M function with associated subroutine and execution before the movement have been programmed in the same block. The subroutines are always executed at the end of the block; therefore, the M function will never be executed before the programmed movement.
SOLUTION	The possible solutions are: • Program the M function without an associated subroutine. • Set the M function in the machine parameter table to be executed after the movement.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 42 of 128

1385   'D and the tool length cannot be modified in the same block'

DETECTION   During execution.

CAUSE   An attempt has been made to write the "V.G.TOL" variable in the same block where a tool change or tool offset change is programmed.

SOLUTION   Programming in different lines.

1386   'D and the tool offsets cannot be modified in the same block'

DETECTION   During execution.

CAUSE   An attempt has been made to write the "V.A.TOFL.axis" variable in the same block where a tool change or tool offset change is programmed.

SOLUTION   Check the program.

1387   'Too many M functions in the same block'

DETECTION   During execution.

CAUSE   The maximum number of M functions that may be programmed in the same block has been exceeded.

SOLUTION   The maximum number of M functions that may be programmed in the same block depends on the software version:
- Up to version V1.10, up to 7 M functions are allowed.
- From version V2.00 on, up to 14 M functions are allowed.

1388   'No more H functions allowed'

DETECTION   During execution.

CAUSE   The maximum number of H functions that may be programmed in the same block has been exceeded. This maximum is 7.

SOLUTION   Program them in different blocks.

1389   'Incompatible G functions (G10/G11/G12/G13/G14)'

DETECTION   During execution.

CAUSE   Two or more G functions of the same group cannot be programmed in the same block.

SOLUTION   Program them in different blocks.

1390   'Incompatible G functions (G98/G99)'

DETECTION   During execution.

CAUSE   Two or more G functions of the same group cannot be programmed in the same block.

SOLUTION   Program them in different blocks.

1392   'Parameter programmed twice'

DETECTION   During execution.

CAUSE   The possible causes are:
- A particular parameter has been programmed more than once in a call #PCALL, G180-G189 or canned cycle.
- A particular parameter has been programmed more than once in the #POLY instruction.

SOLUTION   Eliminate the repeated parameter.

1393   'The current tool and tool offset do not match the ones programmed'

DETECTION   In the block search mode.

CAUSE   The tool with offset D located in the spindle does not match the tool with offset D that has been programmed before reaching the stop point. The CNC has motion blocks prepared that will be used to machine the part after repositioning with the programmed tool radius. If the tool that is in the spindle is different and the tool radius is compensated in the program, it will machine a different part.

SOLUTION   Change the tool of the spindle so it matches the one programmed.

1394   'The subroutine associated with the G function does not exist'

DETECTION   During execution.

CAUSE   The possible causes are:
- Function G74 has been programmed alone in the block, but it has no subroutine associated with machine parameter REFPSUB.
- A function G180-G189 has been programmed alone in the block, but it has no subroutine associated with machine parameter OEMSUB.

SOLUTION   Define in the corresponding parameter the name of the subroutine to be executed and validated.

1395   'Function G74 with associated subroutine must be programmed alone in the block'

DETECTION   During execution.

CAUSE   Only the block number or a label may be programmed in the same block as function G74 in order to execute the associated subroutine.

SOLUTION   Program them in different blocks.

1396   'Programming not allowed in MDI'

DETECTION   During execution.

CAUSE   That command cannot be executed in MDI.

SOLUTION   Execute that command within the program.

1397   'The position programmed for the Hirth axis is wrong'

DETECTION   During execution.

CAUSE   The coordinate programmed for the Hirth axis does not correspond to a whole step.

SOLUTION   It is only possible to program coordinates from REFVALUE plus "n" times HPITCH.

1398   'An axis with parameter HIRTH = NO cannot be activated as Hirth axis'

DETECTION   During execution.

CAUSE   An attempt has been made to activate or deactivate (G171 and G170) a Hirth axis while its machine parameter HIRTH = NO.

SOLUTION   Set machine parameter of the axis: HIRTH = Yes.

1399   'The axis cannot be activated as Hirth'

DETECTION   During execution.

CAUSE   An attempt has been made to activate a Hirth axis when:
- It is the first or second axis of the plane and tool radius compensation is active.
- It is the first or second axis of the plane and collision detection is active.
- It is one of the first three axes and #CS or #ACS transformations are active.
- It is one of the first five axes and 5-axis transformations like #RTCP, #TLC or #TOOL ORI are active.

SOLUTION   Check the program.

1400   'Tool length change is not possible while RTCP is active'

DETECTION   During execution. This error would come up in the first unofficial versions.

CAUSE   When trying to make a tool change while RTCP was active.

SOLUTION   Cancel the RTCP, make the tool change and activate the RTCP function again.

1401   '#TLC ON not allowed without prior deselection'

DETECTION   During execution.

CAUSE   An attempt has been made to activate #TLC ON when it was already active.

SOLUTION   Check the program.

1402   '#LINK: The master and slave axes must have the same parameters HIRTH and HPITCH'

DETECTION   During execution.

CAUSE   An attempt has been made to activate a coupling where:
- The master is a Hirth axis but not the slave or viceversa.
- Both are Hirth axes, but the HPITCH is different.

SOLUTION   Coupling (slaving) is only possible if both or none of the axes is a Hirth axis The Hirth pitch of the axes may be modified in the machine parameter table, HPITCH.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 44 of 128

1403 **'#LINK: a coupling (slaving) cannot be defined with a Hirth axis deactivated'**

DETECTION	During execution.
CAUSE	An attempt has been made to activate a coupling between Hirth axis while one of them is deactivated.
SOLUTION	Activate both Hirth axes to be able to activate the coupling.

1404 **'The gear associated with the programmed M does not exist'**

DETECTION	During execution.
CAUSE	A spindle gear change has been programmed with function M4x (M41 through M44), but the "x" gear does not exist.
SOLUTION	The number of spindle gears available is indicated by its machine parameter NPARSETS and, therefore, the CNC only accepts function M41 through M4x with "x" less than or equal to NPARSETS.

1405 **'The S value exceeds the maximum gear'**

DETECTION	During execution.
CAUSE	The error comes up, having an AUTOGEAR spindle, when programming a speed greater than the maximum of any of the gears existing for that spindle.
SOLUTION	The possible solutions are: • Program a lower spindle speed that may be reached with one of the gears existing for that spindle. • Increase, in the machine parameter table, the maximum speed that can be reached in any of the gears existing for that spindle.

1406 **'#CALL: It does not allow to program parameters'**

DETECTION	During execution.
CAUSE	The #CALL instructiont does not allow to program parameters.
SOLUTION	Program the call to the subroutine with #PCALL.

1407 **'Error when reading the pocket data'**

DETECTION	During execution.
CAUSE	When executing a 2D or 3D pocket, the CNC has not been able to decode some of them. This can happen when the pocket data has been edited manually and a numerical value has been replaced with a variable or the last bracket of instructions #DATAP2D, #DATAP3D,… has been eliminated.
SOLUTION	Edit the pocket again with the cycle editor.

1408 **'Programming not allowed without a spindle being controlled in position'**

DETECTION	During execution.
CAUSE	The programmed function or instruction cannot be executed if the spindle is not controlled in position.
SOLUTION	The spindle must have an encoder.

1409 **'Nesting of T functions with subroutine not allowed'**

DETECTION	During execution.
CAUSE	A T has been executed that has a subroutine associated by machine parameter (TOOLSUB) where another T has been programmed.
SOLUTION	T cannot be programmed within the subroutine associated withT.

1411 **'#CD: wrong number of blocks'**

DETECTION	During execution.
CAUSE	The number of blocks programmed in the instruction is wrong.
SOLUTION	The maximum number of blocks to analyze is 200.

1412 **'#DGWZ: graphics display area defined wrong'**

DETECTION	During execution.
CAUSE	The axis limits have been defined wrong in the instruction.
SOLUTION	Both limits may be positive or negative, but the lower limits of an axis must always be lower than its upper limits.

1413 'The spindle positioning speed cannot be zero'

DETECTION During execution.

CAUSE The programmed positioning speed for function M19 is zero.

SOLUTION Program a positioning speed greater than zero using the syntax: "S.POS =".

1414 '#PARK: this instruction only admits one axis'

DETECTION During execution.

CAUSE The #PARK instruction can only park one axis.

SOLUTION Program a #PARK block for each axis to be parked.

1417 'File path too long'

DETECTION During execution.

CAUSE The maximum number of characters allowed for the path of a program or subroutine has been exceeded.

The path of a program or subroutine may have a maximum of 120 characters.

SOLUTION Move the program or subroutine to another directory to reduce the number of characters of the path.

1418 'An axis of the main plane or of the active transformation cannot be parked'

DETECTION During execution.

CAUSE An attempt has been made to park (#PARK) an axis that is part of the main work plane or of the active kinematics.

SOLUTION Check the program.

1419 'An axis of a gantry axis or a coupling (slaving) cannot be parked'

DETECTION During execution.

CAUSE An attempt has been made to park (#PARK) an axis that is part of a gantry axis or of an active coupling (#LINK).

SOLUTION Check the program.

1420 'Open control blocks at the end of the program'

DETECTION During execution.

CAUSE Some "$" control block does not have its corresponding closing instruction.

SOLUTION Check all the "$" control blocks of the program and subroutines: $IF - $ENDIF, $FOR - $ENFOR, $SWITCH - $ENDSWITCH, …

1421 'The axes of the active transformation can neither be slaves nor parked'

DETECTION During execution.

CAUSE One of the axes involved in the programmed transformation is parked, is a slave of a gantry pair or is a slave of an active coupling.

SOLUTION Unpark, deactivate the active coupling or undo the gantry pair in order to use the axis in the transformation.

1422 '#CS ON/#ACS ON: wrong programmed mode'

DETECTION During execution.

CAUSE The programmed MODE parameter is wrong.

SOLUTION The value of the MODE parameter must be between 1 and 6.

1423 '#CS ON/#ACS ON: the parameter of an aligned axis must be 0 or 1'

DETECTION During execution.

CAUSE The value programmed for the parameter of an aligned axis is wrong.

SOLUTION The value of the parameter of an aligned axis must be 0 or 1.

1424 'G function not allowed while MCS is active'

DETECTION During execution.

CAUSE The programmed G function cannot be executed while #MCS is active.

SOLUTION Check the program.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 46 of 128

1425   'Block skip is only admitted at the beginning of the line'

DETECTION   During execution.
CAUSE   The "/" character is only admitted at the beginning of the program line.
SOLUTION   Check the program.

1426   'The pocket was resolved with a different tool radius'

DETECTION   Check the program.
CAUSE   The pocket was resolved with a different tool radius and it must be generated again.
SOLUTION   Generate the pocket again.

1427   'Axis programmed wrong'

DETECTION   During execution.
CAUSE   Syntax error when programming the instruction or G function, for example:
- G0 <Z>Z20 (axis name repeated).
- G74 X Y (The home searching order for the axes is missing).
- G20 X Y Z (the order of the axes in the system is missing).
SOLUTION   Refer to the manual for the syntax of the instruction or G function.

1428   'The movement in the main plane must be programmed before the G function of the cycle'

DETECTION   During execution.
CAUSE   The coordinates of the axes that define the position of the plane where the cycle must be executed have been defined after the G function of the cycle; therefore, they are considered its parameters.
SOLUTION   Program the coordinates of the axes before the G function of the cycle.

1429   'Too many subroutines in the same block'

DETECTION   During execution.
CAUSE   The maximum number of subroutines that may be executed in the same block has been exceeded. This maximum is 5.
SOLUTION   Program the subroutines in different blocks or use subroutine nesting as necessary.

1430   'Numeric format exceeded'

DETECTION   During execution.
CAUSE   A value has been assigned to a data, variable or parameter that exceeds the established format.
SOLUTION   Check the program.

1431   'Wrong spindle position in M19'

DETECTION   During execution.
CAUSE   The value programmed for the spindle position in M19 is too high.
SOLUTION   Program a smaller value.

1432   'A slave axis of a Gantry or coupling cannot be programmed'

DETECTION   During execution.
CAUSE   A slave axis of an active coupling (#LINK) or a Gantry pair has been programmed in the instruction #CALL AX, #SET AX, #FREE AX or #RENAME AX.
SOLUTION   In order to operate with the axis in those instructions, deactivate the active coupling (#UNLINK) or undo the Gantry pair (by machine parameter).

1433   'A parked axis cannot be part of the main plane'

DETECTION   During execution.
CAUSE   The instruction #CALL AX or #SET AX has been used to try to configure a parked axis as one of the main three axes of the system.
SOLUTION   In order to configure the axis as one of the main three axes of the system, you must unpark it first (#UNPARK).

1434 'An associated slave axis could not be included in the configuration'

DETECTION	During execution.
CAUSE	The instruction #CALL AX or #SET AX has been used to include in the system configuration the master axis of an active coupling (#LINK) or that of a Gantry pair. When including the master axis, the slave axis is also included automatically and it can never occupy one of the main three positions of the channel. The error is issued because there is no free position other than the main three for the slave axis.
SOLUTION	In order to only include the master axis, the active coupling must be deactivated first (#UNLINK) or undo the Gantry pair.
	In order to include both the master and the slave axis, first eliminate another axis from the channel or increase the number of axes of the channel.

1435 'An associated slave axis could not be assigned a name because it was repeated'

DETECTION	During execution.
CAUSE	The instruction #CALL AX or #SET AX has been used to include in the system configuration the master axis of an active coupling (#LINK) or that of a Gantry pair. When including the master, the slave axis is also included automatically. The error comes up because the name of the slave axis is already occupied by another axis of the current configuration of the channel.
SOLUTION	Rename either the slave axis to be included or the one that already exists in the configuration.

1436 'The stop block has not been programmed in the block search'

DETECTION	During execution.
CAUSE	The option to execute the program in block search has been selected, but the final block of that execution has not been indicated.
SOLUTION	Once the block search option has been selected, it shows the stop block option. Select the block where the block search will end.

1439 'The axes of the active transformation cannot be Hirth'

DETECTION	During execution.
CAUSE	An attempt has been made to activate an axis transformation (#TLC, #RTCP, #TOOL ORI, #CS or #ACS), but one of the axes involved in the transformation is a Hirth axis.
SOLUTION	For the axis to be able to be part of the transformation, it must not be a Hirth axis; therefore, set its machine parameter HIRTH = NO.

1441 'The kinematics has not been activated'

DETECTION	During CNC power-up or parameter validation.
CAUSE	There is a kinematics that must activate, but it doesn't either because it is an unknown kinematics or because the axes required for that kinematics are not the right ones.
SOLUTION	Make the value of the type of kinematics to be activated a valid value and make sure that the axes involved in it are properly defined. The axes must always be the first ones of the channel, they must not be Hirth, they must not be parked and they must not be slaves of a coupling or a gantry pair.

1442 'The kinematics has been deactivated'

DETECTION	Warning during reset or at the beginning of the program.
CAUSE	There is a kinematics that has been deactivated, either because it is an unknown kinematics or because the axes required for that kinematics are not the right ones.
SOLUTION	Make the value of the type of kinematics to be activated a valid value and make sure that the axes involved in it are properly defined. The axes must always be the first ones of the channel, they must not be Hirth, they must not be parked and they must not be slaves of a coupling or a gantry pair.

1443 '#CS/ACS has been deactivated'

DETECTION	Warning during reset or at the beginning of the program.
CAUSE	The axes required for the transformation of an incline plane are not the right ones.
SOLUTION	Make sure that the main three axes of the channel are defined, linear, unparked and not slaves of a coupling or a gantry pair.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 48 of 128

1444 'The main axes of the transformation must be linear'

DETECTION	During execution.
CAUSE	One of the main three axes involved in the programmed transformation or kinematics is neither a linear axis nor a C axis.
SOLUTION	The main three axes of the transformation or kinematics must be linear (machine parameter AXISTYPE) or a C axis (machine parameter CAXIS).

1445 'Wrong parameter value'

DETECTION	During execution.
CAUSE	In a canned cycle a parameter has been assigned a wrong value.
SOLUTION	Check the program.

1446 'Starting block not allowed in a local subroutine'

DETECTION	During execution.
CAUSE	The starting block cannot be a block of a local subroutine.
SOLUTION	Select another starting block nearby

1447 'Software option not allowed'

DETECTION	During execution.
CAUSE	The CNC does not have the software option required to execute the programmed command.
SOLUTION	In diagnosis, it is possible to check the software options offered by the CNC.

1448 'The tool cannot be placed perpendicular to the active plane'

DETECTION	During execution.
CAUSE	The possible causes are: • The spindle type does not allow positioning the tool in that position as it may be the case of angular spindles. • Positioning the spindle perpendicular to the active plane is out of limits.
SOLUTION	Define another active plane or change the spindle.

1449 '#PATH instruction programmed wrong'

DETECTION	During execution.
CAUSE	The syntax of the #PATH instruction is wrong.
SOLUTION	Refer to the programming manual.

1450 'Wrong solution for placing spindle perpendicular to the active plane'

DETECTION	During execution.
CAUSE	An attempt has been made to read the TOOLORIF1 or TOOLORIF2 variable, but its value is wrong because the tool could not be positioned perpendicular to the active plane.
SOLUTION	There are different causes for the tool not being able to position perpendicular to the active plane. Depending on the cause, the solution may be defining a new plane or changing the spindle (see error 1448).

1451 'Nonexistent variable for the type of axis'

DETECTION	During execution.
CAUSE	The variable does not exit for the requested axis type (linear, rotary or spindle).
SOLUTION	Check the program.

1452 'Nonexistent variable for the type of drive'

DETECTION	During execution.
CAUSE	The variable does not exit for the requested drive type (analog, simulated or sercos).
SOLUTION	Check the program.

1453 'Axis name too long'

DETECTION	During execution.
CAUSE	The axis name has more than two characters.
SOLUTION	The valid axis names are: X, X1 … X9, Y, Y1, … Y9, …, Z, Z1, … Z9 A, A1 … A9, B, B1, … B9, …, C, C1, … C9 U, U1 … U9, V, V1, … V9, …, W, W1, … W9 S, S1 … S9

1455 'PROFILE: Null profile'

DETECTION	During execution.
CAUSE	The possible causes are: • In the profile cycle of the cycle editor, the file containing the profile cycle is missing. • The file indicated in the profile cycle of the cycle editor is empty.
SOLUTION	The profile cycle of the cycle editor must indicate the file containing the profile.

1456 '#POLY: Parameters missing'

DETECTION	During execution.
CAUSE	When programming the #POLY instruction, some mandatory parameter "EP" or "SP" is missing
SOLUTION	Check the syntax of the instruction in the programming manual.

1457 '#POLY: parameter value'

DETECTION	During execution.
CAUSE	The possible causes are: • The interpolation parameters of the polynomial are wrong. • The curvature radius is less than or equal to zero.
SOLUTION	The interpolation parameters of the polynomial must be positive, SP < EP and the curvature radius must be greater than zero.

1458 '#POLY: Too many axes have been programmed'

DETECTION	During execution.
CAUSE	More than three axes have been programmed in the polynomial.
SOLUTION	A polynomial interpolation can only be programmed for three axes.

1459 '#POLY: Wrong starting point'

DETECTION	During execution.
CAUSE	The starting point of the polynomial is not the same as the current position.
SOLUTION	Modify the independent term of the polynomial for each axis make it the same as the end position of the previous block.

1461 'G9: The arc's intermediate point has been programmed wrong'

DETECTION	During execution.
CAUSE	One or both coordinates of the arc's intermediate point have not been programmed.
SOLUTION	Function G9 requires the programming of both coordinates of the arc's intermediate point.

1462 'G8: the tangent path cannot be calculated'

DETECTION	During execution.
CAUSE	An arc tangent to the previous path cannot be done with the programmed radius and end point.
SOLUTION	Check the values of the programmed radius and end point.

1463 'G9: Circular path programmed wrong'

DETECTION	During execution.
CAUSE	No arc can go through those three points.
SOLUTION	Define two points in the block that define an arc with the end point of the previous movement. Bear in mind that all three points must be different and cannot be in line.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 50 of 128

1464   'Programmed rotary axis out of the range of the module'

DETECTION	During execution.
CAUSE	The absolute coordinate (G90) programmed for the MODULE type rotary axis is wrong.
SOLUTION	The coordinate programmed for the axis must be between the limits set by its machine parameters MODUPLIM and MODLOWLIM.

1465   'Functions RTCP and TLC are incompatible with each other'

DETECTION	During execution.
CAUSE	While one of the two functions was active, an attempt has been made to activate the other one.
SOLUTION	Both functions cannot be active at the same time.

1466   'G8 cannot be programmed as second motion block for G36/G37/G38/G39'

DETECTION	During execution.
CAUSE	A G8 block cannot be the second motion block of one of the functions G36/G37/G38/G39. On one hand, these functions do not have an intermediate block to make the path joining two blocks and, on the other hand, function G8 does not have a previous block to be tangent to.
SOLUTION	The second motion block of a G36/G37/G38/G39 function must be G0/G1/G2/G3.

1467   'POSLIMIT/NEGLIMIT cannot exceed the value of the machine parameter'

DETECTION	During execution.
CAUSE	The value to be written in variables A.POSLIMIT.axis and A.NEGLIMIT.axis cannot exceed the value of machine parameters POSLIMIT and NEGLIMIT for that axis.
SOLUTION	Check the program.

1468   'G30: Polar origin programmed wrong'

DETECTION	During execution.
CAUSE	Only one of the coordinates of the Polar origin has been programmed.
SOLUTION	The coordinates of the Polar origin must be programmed in both main axes.

1469   'Negative or zero radius not allowed'

DETECTION	During execution.
CAUSE	A movement has been programmed in Polar coordinates, but the Polar radius is negative or zero.
SOLUTION	The Polar radius must always be greater than zero. When programming in incremental mode, the programmed value may be negative or zero, but not the absolute Polar radius.

1470   'UNIDIR rotary axis programmed wrong in incremental mode'

DETECTION	During execution.
CAUSE	The incremental coordinate programmed for the MODULE and UNIDIR type rotary axis is wrong.
SOLUTION	If machine parameter UNIDIR of the rotary axis is POSITIVE, the incremental coordinate must be programmed with a positive or zero value. If machine parameter UNIDIR of the rotary axis is NEGATIVE, the incremental coordinate must be programmed with a negative or zero value.

1471   'G73: Rotation center programmed wrong'

DETECTION	During execution.
CAUSE	Only one of the two coordinates of the pattern (coordinate system) rotation has been programmed.
SOLUTION	The coordinates of the rotation center must be programmed in both main axes.

1472   'G73: The rotation angle has not been programmed'

DETECTION	During execution.
CAUSE	The coordinates of the rotation center have been programmed, but not the rotation angle.
SOLUTION	The coordinate system rotation angle must be programmed together with the coordinates of the rotation center in both main axes.

1473   '#POLY cannot be programmed while pattern rotation is active'

DETECTION	During execution.
CAUSE	The #POLY instruction has been programmed while the coordinate system rotation (G73) was active.
SOLUTION	Check the program.

1475   'Radius programmed twice'

DETECTION	During execution.
CAUSE	The radius "R" or "R1" has been programmed more than one in the same block.
SOLUTION	Define only one radius in the block.

1476   'The pocket was resolved with a different tool nose (tip) radius'

DETECTION	During execution.
CAUSE	The pocket was generated with a tool radius other than the current one. To execute it, it must be generated again.
SOLUTION	Generate the pocket again.

1477   'The pocket was resolved with a different tool's cutting length'

DETECTION	During execution.
CAUSE	The pocket was generated with a tool's cutting length other than the current one. To execute it, it must be generated again.
SOLUTION	Generate the pocket again.

1478   'The pocket was resolved with a different tool entry angle'

DETECTION	During execution.
CAUSE	The pocket was generated with a tool entry angle other than the current one. To execute it, it must be generated again.
SOLUTION	Generate the pocket again.

1479   'G74: A subroutine has not been associated'

DETECTION	During execution.
CAUSE	Function G74 has been programmed alone in the block, but no subroutine associated by machine parameter has been defined.
SOLUTION	Associate a subroutine to G74 using machine parameter REFSUB.

1480   'Program: #EXEC ["path+program",channel]'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

1481   'Wrong channel number'

DETECTION	During execution.
CAUSE	A wrong channel number has been programmed in the instruction #EXEC, #MEET or #WAIT.
SOLUTION	The channel number must be between 1 and 4.

1483   'Program: #WAIT/#MEET [signal, channel, channel, ...]'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 52 of 128

1484 'Signal number out of range'

DETECTION	During execution.
CAUSE	The signal number programmed in the instruction #WAIT, #MEET or #SIGNAL is wrong.
SOLUTION	The signal programmed in the instructions #WAIT, #MEET and #SIGNAL must be between 1 and 10.

1485 '#WAIT/#MEET not effective'

DETECTION	During execution.
CAUSE	It informs the user that the instruction #WAIT or #MEET does not generate any wait because the delayed activation of a signal in the same channel has been programmed in it.
SOLUTION	Check the program.

1486 'Program: #SIGNAL [signal, signal, signal, ...]'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

1487 'Program: #CLEAR [signal, signal, signal, ...]'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

1489 'Axis name repeated in the resulting group'

DETECTION	During execution.
CAUSE	Using the instruction #RENAME AX, more than one axis have been renamed with the same name.
SOLUTION	Rename each axis with a different name so the resulting group of axes in the channel does not have two with the same name.

1490 'G63 requires programming M19 before'

DETECTION	During execution.
CAUSE	An attempt has been made to do a threading G63 with a SERCOS spindle without previously orienting it with M19.
SOLUTION	Program M19 before doing the threading.

1491 'Wrong probe number'

DETECTION	During execution.
CAUSE	The probe number selected with function #SELECT PROBE is wrong.
SOLUTION	The selected probe number must be either 1 or 2.

1492 'There is no digital input associated with the probe (PRBDI1/2)'

DETECTION	During execution.
CAUSE	The possible causes are: • A measurement has been taken with function G100, but there is no digital probe input associated with the probe. • A probe has been selected using the instruction #SELECT PROBE, but there is no digital input associated with that probe.
SOLUTION	Associate a digital input with the probe using machine parameter PRBDI1/2.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 53 of 128

1493 **'#SPLINE ON, G41/G42 and G136 cannot be programmed at the same time'**

DETECTION	During execution.
CAUSE	An attempt has been made to activate functions #SPLINE ON, G41, G42 and G136 at the same time at the CNC.
SOLUTION	Cancel some of these functions.

1494 **'Wrong index'**

DETECTION	During execution.
CAUSE	The error number indicated in the #ERROR or #WARNING instruction does not exist.
SOLUTION	Check the program.

1495 **'#PROBE1: wrong axis for probing cycle'**

DETECTION	During execution.
CAUSE	The PROBE1 cycle issues the error because the main three axes of the current configuration are not the same as the main three axes of the initial configuration.
SOLUTION	Go back to the initial configuration for the first three axes of the channel.

1496 **'#PROBE1: programming not allowed while #TOOL AX[-] is active'**

DETECTION	During execution.
CAUSE	An attempt has been made to execute the PROBE1 cycle while the tool is oriented in the negative direction of the axis.
SOLUTION	Program #TOOL AX[+] before executing the PROBE1 cycle to orient the tool in the positive direction of the axis.

1497 **'Wrong operator for the type of variable'**

DETECTION	During execution.
CAUSE	An attempt has been made to change the value of the variable AXISNAMEi, GAXISNAMEi, SPDLNAMEi or GSPDLNAMEi using the operators "+=", "-=", "*=" y "/=".
SOLUTION	Check the program.

1499 **'Too many nesting levels of #RPT and subroutines'**

DETECTION	During execution.
CAUSE	It is considered nesting when the #RPT instruction is programmed between two labels that defined the influence area of another #RPT Bearing this in mind, the error comes up because the number of nesting levels of RPT instructions and subroutines, if any, has been exceeded.
SOLUTION	The maximum number of nesting levels allowed is 20.

1500 **'#EXEC: the program cannot be executed in the indicated channel'**

DETECTION	During execution.
CAUSE	An attempt has been made to execute a program with #EXEC in a channel that: • Where another program is being executed. • That is in error. • Was in manual and cannot switch to automatic mode.
SOLUTION	Wait for the program to finish in the other channel or reset it.

1501 **'Labels repeated in #RPT'**

DETECTION	During execution.
CAUSE	In the #RPT instruction, the starting and finishing labels are the same.
SOLUTION	Define different starting and finishing labels.

1502 **'The variable requires programming an array index'**

DETECTION	During execution.
CAUSE	A variable has been programmed, but no index has been indicated for which to read or write a variable.
SOLUTION	Refer to the programming manual for control variables.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 54 of 128

1503   'The variable requires programming an axis'

DETECTION   During execution.

CAUSE   An axis variable has been programmed, but no axis has been indicated for which to read or write a variable.

SOLUTION   Refer to the programming manual for control variables.

1504   'The variable does not allow programming an array index'

DETECTION   During execution.

CAUSE   An array index has been programmed when reading or writing a variable that is not an array variable.

SOLUTION   Refer to the programming manual for control variables.

1505   'The variable does not allow programming an axis'

DETECTION   During execution.

CAUSE   An axis has been programmed when reading or writing a variable that is not an axis variable.

SOLUTION   Refer to the programming manual for control variables.

1506   'Undefined label or #RPT command labels swapped'

DETECTION   During execution.

CAUSE   The possible causes are:
              • The second label has not been defined.
              • M30 has been programmed between the first and the second labels.
              • In the #RPT instruction, the second label has been programmed first and then the first one.

SOLUTION   Define the labels and the instruction correctly.

1507   'The second label of the #RPT must be programmed alone in the block'

DETECTION   During execution.

CAUSE   When programming the #RPT[N1,N2] instruction, the second label must be programmed alone in the block without any other type of command. This error would come up if N2 :X10 were programmed.

SOLUTION   The last label must be programmed alone in the block. Program the command in the previous line to execute it with #RPT or in the next line not to execute it with the #RPT.

1508   'G201 and active C axis are not allowed on the axes of the main plane'

DETECTION   During execution.

CAUSE   The possible causes are:
              1. G201 has been programmed for a C axis that is also one of the first three axes of the channel.
              2. One of the axes programmed in the #FACE instruction is in additive manual mode, G201.

SOLUTION   The possible solutions are:
              1. Program #FACE OFF so the axis stops being a C axis.
              2. Program G202 #AXIS [axis] so the axis stops being in additive manual mode.

1509   '#SET AX/#CALL AX: Offset programming has no effect'

DETECTION   During execution.

CAUSE   All the axes programmed in the instruction already belonged to the current configuration and, therefore, the instruction only means a change of order (sequence). In this case, the programmed offset option has no effect.

SOLUTION   Check the program.

1510 'The axis does not exist or is not available in the channel'

DETECTION   During execution.

CAUSE   The possible causes are:
- An attempt has been made to move an axis that does not exist or is not available in the channel.
- An attempt has been made to execute an instruction that involves an axis that does not exist or is not available in the channel.
- An attempt has been made to read or write an axis variable for an axis that does not exist or is not available in the channel.

SOLUTION   Indicate, in any of the previous actions, axes that belong to the channel.

An new axis may be included in the channel by modifying the machine parameters of the channel CHNAXIS and CHAXISNAME or using the instructions #SET AX or #CALL AX.

1511 'The tool is not in the magazine'

DETECTION   During execution.

CAUSE   An attempt has been made to use the TM.P[ ] variable to read the magazine position of a tool that is not there.

SOLUTION   It is only possible to read the position of the tools that are in the magazine.

1512 'Free or nonexistent magazine position'

DETECTION   During execution.

CAUSE   An attempt has been made to use the TM.T[ ] variable to read the number of a tool located in a nonexistent magazine position.

SOLUTION   It is only possible to read the tool that is a in valid magazine position.

1513 'Error when writing the variable'

DETECTION   During execution.

CAUSE   An attempt has been made to write a variable and:
- The variable does not exist.
- It is an axis variable, but the axis does not exist.
- It is an axis variable, but it does not exist for the type of axis.
- It is an axis variable, but it does not exist for the type of drive of the axis.
- The combined operator (+=, ...) is not allowed for that variable.
- The value to write is not valid for that variable.

SOLUTION   Check the syntax of the variable and the value to write in it.

1514 'A 2D-3D pocket cannot be executed while G72 is active'

DETECTION   During execution.

CAUSE   A scaling factor (G72 or #SCALE[n] command) is already active when trying to execute a pocket.

SOLUTION   Cancel the scaling factor.

1515 'One or several axes of the original configuration are not available'

DETECTION   During execution.

CAUSE   An axis of the temporary exchange has been released and cannot be restored with a reset or beginning of a program because the channel that took it has not released (freed) it.

SOLUTION   The other channel will release the axis with reset or with the beginning of another program. The axis release can also be programmed with the instruction #FREE AX.

1516 'Value expected'

DETECTION   During execution.

CAUSE   The list of subroutine calling parameters (with #PCALL or G180-189) has been programmed wrong.

SOLUTION   The list of parameters to call a subroutine may consist of:
- Local arithmetic parameters: P0 = 30.
- letters: A = 30 or A30.

Both can be mixed on the list.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 56 of 128

1518 'NR requires programming a movement in the block'

DETECTION	During execution.
CAUSE	The repetition (NR) of a block has been programmed that does not involve a movement.
SOLUTION	Programming block repetition with NR is only valid with blocks that involve a movement. The repetition of other types of blocks must be programmed using flow controlling instructions offered by the CNC.

1519 'NR: M/T/D/H cannot be programmed in the same block'

DETECTION	During execution.
CAUSE	An M, T, D or H function has been programmed in the same block as the number of repetitions (NR).
SOLUTION	Programming block repetition with NR is only valid with blocks that involve a movement. The repetition of other types of blocks must be programmed using flow controlling instructions offered by the CNC.

1520 'NR: $GOTO cannot be programmed in the same block'

DETECTION	During execution.
CAUSE	The $GOTO instruction has been programmed in the same block as the number of repetitions (NR).
SOLUTION	Programming block repetition with NR is only valid with blocks that involve a movement. The repetition of other types of blocks must be programmed using flow controlling instructions offered by the CNC.

1521 'NR: A call to a subroutine cannot be programmed in the same block'

DETECTION	During execution.
CAUSE	A call to a subroutine (L, LL, #CALL, #PCALL, #MCALL or G180-G189) has been programmed in the same block as the number of repetitions (NR).
SOLUTION	Programming block repetition with NR is only valid with blocks that involve a movement. The repetition of other types of blocks must be programmed using flow controlling instructions offered by the CNC.

1522 'Negative value not allowed'

DETECTION	During execution.
CAUSE	A negative number of block repetitions (NR) has been programmed.
SOLUTION	Program a zero or positive value.

1523 'POS and T must be programmed in the same block'

DETECTION	During execution.
CAUSE	The tool T and the position POS has not been programmed in the same block.
SOLUTION	The tool T and the position it must be occupied in the magazine must be programmed in the same block.

1525 'The axis cannot be exchanged'

DETECTION	During execution.
CAUSE	An attempt has been made to exchange an axis during the instruction #CALL AX, #SET AX or #FREE AX while its machine parameter "AXISEXCH" = NO.
SOLUTION	In order to be able to exchange axes between channels, machine parameter "AXISEXCH" of those axes must set to "temporary" or "maintained".

1526 '#EXEC: the indicated channel is not a CNC channel'

DETECTION	During execution.
CAUSE	An attempt has been made to use the #EXEC instruction to execute a program in a channel that is not a CNC channel, but a PLC channel.
SOLUTION	The type of channel may be changed using machine parameter CHTYPE.

1527 'Wrong identifier after %'

DETECTION	During execution.
CAUSE	In the instruction #MSG, #ERROR or #WARNING , an invalid identifier has been programmed after "%".
SOLUTION	The valid identifiers are: • To display a number: %D or %d. • To display the "%" character: %%.

1529 'Expecting list of identifiers or "]" after end-quotes'

DETECTION	During execution.
CAUSE	In the instruction #MSG, #ERROR or #WARNING , an invalid character has been programmed after the message to be displayed.
SOLUTION	After the end-quotes of the message to be displayed, it only admits the list of variables separated by commas and the closing bracket. Check the syntax of this instruction in the programming manual.

1530 'G53 cannot be programmed with polar coordinates'

DETECTION	During execution.
CAUSE	The final point has been defined with Polar or Cylindrical coordinates in function G53.
SOLUTION	When programming referred to machine zero (home), the points must be defined only with Cartesian coordinates.

1531 'Program: #EXBLK [block, channel]'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

1532 'Program: #MASTER <spindle name>'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

1533 'Program: #FREESP [sp1, sp2, ..]'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

1534 'Program: #CALLSP [sp1, sp2, ..]'

DETECTION	During execution.
CAUSE	The possible causes are: • The syntax of the instruction is wrong. • Only the block number of a label may be programmed in the same block as the instruction.
SOLUTION	In the first case, refer to the programming manual. In the second case, program in different blocks.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 58 of 128

1535 'Program: #SETSP [sp1, sp2, ..]'

DETECTION   During execution.

CAUSE   The possible causes are:
- The syntax of the instruction is wrong.
- Only the block number of a label may be programmed in the same block as the instruction.

SOLUTION   In the first case, refer to the programming manual. In the second case, program in different blocks.

1538 'A spindle cannot be restored from the channel'

DETECTION   During execution.

CAUSE   If a spindle that belongs to a channel has been released and it is one that must be restored with a RESET or beginning of a program (AXISEXCH = TEMPORARY), it may not be possible if it is being used by another channel.

SOLUTION   The channel that now has the spindle must release it. Otherwise, the original channel that the spindle belongs to will not be able to use it and it will issue a warning at each RESET or at the beginning of a program.

1539 'Spindle name repeated in the resulting group'

DETECTION   During execution.

CAUSE   Using the instruction #RENAME SP, more than one spindle have been renamed with the same name.

SOLUTION   Rename each spindle with a different name so the resulting group of axes in the channel does not have two with the same name.

1540 'Programming not allowed without a master spindle in the channel'

DETECTION   During execution.

CAUSE   The possible causes are:
- An attempt has been made to read or write a variable for the master spindle, but it does not exist in the channel.
- The G function or instruction cannot be executed if there is no master spindle in the channel.

SOLUTION   Define a master spindle for the channel using the #MASTER instruction.

1541 'The spindle cannot be eliminated while the C axis is active'

DETECTION   During execution.

CAUSE   Using the #FREE SP or #SET SP instruction, an attempt has been made to eliminate the spindle from the channel that is working as C axis.

SOLUTION   Before removing the spindle from the channel, cancel the C axis with the #CAX OFF instruction or activate another spindle as C axis.

1542 'The spindle cannot be exchanged'

DETECTION   During execution.

CAUSE   Using the instruction #CALL SP, #SET SP or #FREE SP, an attempt has been made to change the spindle from one channel to another, but the spindle is not exchangeable.

SOLUTION   In order for the spindle to be exchangeable, set its machine parameter AXISEXCH to TEMPORARY or MAINTAINED.

1544 'G63 and M3/M4/M5/M19/M41-M44 not allowed in the same block'

DETECTION   During execution.

CAUSE   No M function associated with the spindle may be programmed in the same block that contains a rigid tapping G63.

SOLUTION   It may be programmed either in the previous or next block depending on the result to be obtained. If programmed in the next block, these M functions cancel the modal tapping G63 in such way that G63 must be programmed again in the next movement in order to keep tapping.

1545 'Spindle gear change is not possible while G63 or #CAX is active'

DETECTION	During execution.
CAUSE	The spindle gear change is not possible while tapping G63 is active or while working as C axis.
SOLUTION	Cancel G63 or #CAX, change the gear and activate functions G63 or #CAX again if so wished.

1546 'G63 not allowed without prior gear at the spindle'

DETECTION	During execution.
CAUSE	This situation is only possible if after a system power-up or a reset, the PLC does not indicate any gear with GEAR1/2/3/4 for that particular spindle. Then, there is no spindle gear and if the spindle is not used before programming G63, it won't be generated automatically, either.
SOLUTION	Before starting a G63 tapping, the spindle must have a gear activated. Generate or program a spindle gear before activating G63.
	• Directly using functions M41-M44.
	• If the spindle is "AUTOGEAR" the gear is also generated when programming a speed (S).
	• If the spindle is not "AUTOGEAR", the gear may be generated by programming a speed (S) and M3 or M4.

1547 '#CAX not allowed while G63 is active'

DETECTION	During execution.
CAUSE	A spindle cannot be activated as C axis if it is being used by function G63.
SOLUTION	Cancel G63 before programming #CAX or use another spindle of the channel for the C axis.

1548 'The master spindle cannot be changed while G33/G63/G95/G96 are active'

DETECTION	During execution.
CAUSE	Function G63 and G96 use the master spindle of the channel. This one cannot be removed from the channel while these functions are active.
SOLUTION	Cancel G63 or G96 before releasing the spindle.

1549 'Null or wrong pocket name'

DETECTION	During execution.
CAUSE	When a 2D or 3D pocket has been defined with the cycle editor, the pocket name has not been defined or its name is wrong. The pocket name cannot be #DATAP2D or #DATAP3D generated by the cycle editor.
SOLUTION	Assign a different name to the pocket.

1550 'Two C axes cannot be programmed in the same channel'

DETECTION	During execution.
CAUSE	The C axis feature may be activating using any spindle of the channel, but not simultaneously with more than one of them.
SOLUTION	Cancel #CAX on a spindle before activating it for another one.

1551 'Too many local parameter nesting levels'

DETECTION	During execution.
CAUSE	The nesting of subroutine calls with #PCALL or with function G1-G190 increases the nesting level of the local parameters. The error comes up because the maximum nesting level (7) of local parameters has been exceeded.
SOLUTION	Decrease the nesting of subroutines or use the instructions #CALL, L or LL to call them; they do not increase the nesting level of local parameters.

1552 'The name of the variable must begin with a "V."'

DETECTION	During execution.
CAUSE	The name of a variable has been written from a part program or MDI without the starting prefix "V."
SOLUTION	Add the prefix "V." to the name of the variable.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 60 of 128

1553 'Too many user variables'

DETECTION	During execution.
CAUSE	The maximum number of user variables (V.P, V.S) that can be defined has been exceeded.
SOLUTION	The maximum number of user variables that can be defined is 20.

1554 'The PLC has not recognized the START in a #EXEC instruction'

DETECTION	Execution of #EXEC instruction.
CAUSE	When giving the order to execute a program in another channel, the PLC must give the "go ahead" to the program start-up (START) If it doesn't give this go-ahead, it will issue this error.
SOLUTION	Check everything that could prevent, via PLC, the execution of a program in that channel.

1700 'CENTER PUNCHING: F = 0'

DETECTION	Executing the center-punching cycle.
CAUSE	F programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1701 'CENTER PUNCHING: S = 0'

DETECTION	Executing the center-punching cycle.
CAUSE	S programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1702 'CENTER PUNCHING: T = 0'

DETECTION	Executing the center-punching cycle.
CAUSE	No tool T has been programmed in the cycle.
SOLUTION	Assign a tool T.

1703 'CENTER PUNCHING: P = 0'

DETECTION	Executing the center-punching cycle.
CAUSE	P (depth) programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1704 'CENTER PUNCHING: ALPHA = 0'

DETECTION	Executing the center-punching cycle.
CAUSE	Angle alpha programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1705 'CENTER PUNCHING: DIAMETER = 0'

DETECTION	Executing the center-punching cycle.
CAUSE	Diameter programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1706 'DRILLING 1: F = 0'

DETECTION	Executing drilling cycle 1.
CAUSE	F programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1707 'DRILLING 1: S = 0'

DETECTION	Executing drilling cycle 1.
CAUSE	S programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1708 'DRILLING 1: T = 0'

DETECTION	Executing drilling cycle 1.
CAUSE	No tool T has been programmed in the cycle.
SOLUTION	Assign a tool T.

1709   **'DRILLING 1: P = 0'**

DETECTION   Executing drilling cycle 1.
CAUSE   P (depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1710   **'DRILLING 2: F = 0'**

DETECTION   Executing drilling cycle 2.
CAUSE   F programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1711   **'DRILLING 2: S = 0'**

DETECTION   Executing drilling cycle 2.
CAUSE   S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1712   **'DRILLING 2: T = 0'**

DETECTION   Executing drilling cycle 2.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a tool T.

1713   **'DRILLING 2: P = 0'**

DETECTION   Executing drilling cycle 2.
CAUSE   P (depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1714   **'DRILLING 2: B = 0'**

DETECTION   Executing drilling cycle 2.
CAUSE   B (drilling peck) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1715   **'THREADING: F = 0'**

DETECTION   Executing the threading cycle.
CAUSE   F programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1716   **'THREADING: S = 0'**

DETECTION   Executing the threading cycle.
CAUSE   S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1717   **'THREADING: T = 0'**

DETECTION   Executing the threading cycle.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a tool T.

1718   **'THREADING: P = 0'**

DETECTION   Executing the threading cycle.
CAUSE   P (depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1719   **'REAMING: F = 0'**

DETECTION   Executing the reaming cycle.
CAUSE   F programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1720   **'REAMING: S = 0'**

DETECTION   Executing the reaming cycle.
CAUSE   S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 62 of 128

1721   'REAMING: T = 0'

DETECTION   Executing the reaming cycle.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a tool T.

1722   'REAMING: P = 0'

DETECTION   Executing the reaming cycle.
CAUSE   P (depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1723   'BORING 1: F = 0'

DETECTION   Executing boring cycle 1.
CAUSE   F programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1724   'BORING 1: S = 0'

DETECTION   Executing boring cycle 1.
CAUSE   S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1725   'BORING 1: T = 0'

DETECTION   Executing boring cycle 1.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a tool T.

1726   'BORING 1: P = 0'

DETECTION   Executing boring cycle 1.
CAUSE   P (depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1727   'DRILLING 3: F = 0'

DETECTION   Executing drilling cycle 3.
CAUSE   F programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1728   'DRILLING 3: S = 0'

DETECTION   Executing drilling cycle 3.
CAUSE   S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1729   'DRILLING 3: T = 0'

DETECTION   Executing drilling cycle 3.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a tool T.

1730   'DRILLING 3: P = 0'

DETECTION   Executing drilling cycle 3.
CAUSE   P (depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1731   'BORING 2: F = 0'

DETECTION   Executing boring cycle 1.
CAUSE   F programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1732   'BORING 2: S = 0'

DETECTION   Executing boring cycle 1.
CAUSE   S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1733 'BORING 2: T = 0'

DETECTION   Executing boring cycle 1.

CAUSE   No tool T has been programmed in the cycle.

SOLUTION   Assign a tool T.

1734 'BORING 2: P = 0'

DETECTION   Executing boring cycle 1.

CAUSE   P (depth) programmed in the cycle equal to 0.

SOLUTION   Assign a value greater than 0.

1735 'RECTANGULAR POCKET: F = 0'

DETECTION   Executing rectangular pocket cycle.

CAUSE   F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.

SOLUTION   Assign a value greater than 0.

1736 'RECTANGULAR POCKET: S = 0'

DETECTION   Executing rectangular pocket cycle.

CAUSE   Roughing or finishing S programmed in the cycle equal to 0.

SOLUTION   Assign a value greater than 0.

1737 'RECTANGULAR POCKET: T = 0'

DETECTION   Executing rectangular pocket cycle.

CAUSE   No tool T has been programmed in the cycle.

SOLUTION   Assign a roughing or finishing tool T.

1738 'RECTANGULAR POCKET: P = 0'

DETECTION   Executing rectangular pocket cycle.

CAUSE   P (pocket depth) programmed in the cycle equal to 0.

SOLUTION   Assign a value greater than 0.

1739 'RECTANGULAR POCKET: tool diameter smaller than DELTA'

DETECTION   Executing rectangular pocket cycle.

CAUSE   The diameter of the roughing or finishing tool is smaller than the roughing or finishing pass in the main plane.

SOLUTION   Decrease the size of the milling pass in XY (DELTA) or machine with a tool that has a larger diameter.

1740 'RECTANGULAR POCKET: tool diameter greater than the pocket'

DETECTION   Executing rectangular pocket cycle.

CAUSE   The diameter of the roughing or finishing tool is greater than the pocket dimensions L or H.

SOLUTION   Choose a tool of a smaller diameter to make the pocket.

1741 'RECTANGULAR POCKET: Finishing tool diameter smaller than delta'

DETECTION   Executing rectangular pocket cycle.

CAUSE   The diameter of the finishing tool is smaller than the finishing stock in the main plane (delta).

SOLUTION   Choose a tool of a larger diameter to make the pocket.

1742 'PRE-EMPTIED POCKET: F = 0'

DETECTION   Executing pre-emptied pocket cycle.

CAUSE   F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.

SOLUTION   Assign a value greater than 0.

1743 'PRE-EMPTIED POCKET: S = 0'

DETECTION   Executing pre-emptied pocket cycle.

CAUSE   Roughing or finishing S programmed in the cycle equal to 0.

SOLUTION   Assign a value greater than 0.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 64 of 128

1744   **'PRE-EMPTIED POCKET: T = 0'**

DETECTION   Executing pre-emptied pocket cycle.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a roughing or finishing tool T.

1745   **'PRE-EMPTIED POCKET: P = 0'**

DETECTION   Executing pre-emptied pocket cycle.
CAUSE   P (pocket depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1746   **'PRE-EMPTIED POCKET: tool diameter smaller than DELTA'**

DETECTION   Executing pre-emptied pocket cycle.
CAUSE   The diameter of the roughing or finishing tool is smaller than the roughing or finishing pass in the main plane.
SOLUTION   Decrease the size of the milling pass in XY (DELTA) or machine with a tool that has a larger diameter.

1747   **'PRE-EMPTIED POCKET: tool diameter greater than the pocket'**

DETECTION   Executing pre-emptied pocket cycle.
CAUSE   The diameter of the roughing or finishing tool is greater than the pocket dimensions L or H.
SOLUTION   Choose a tool of a smaller diameter to make the pocket.

1748   **'PRE-EMPTIED POCKET: Finishing tool diameter smaller than delta'**

DETECTION   Executing pre-emptied pocket cycle.
CAUSE   The diameter of the finishing tool is smaller than the finishing stock in the main plane (delta).
SOLUTION   Choose a tool of a larger diameter to make the pocket.

1749   **'PRE-EMPTIED POCKET: R <r'**

DETECTION   Executing pre-emptied pocket cycle.
CAUSE   The outside diameter of the pocket (R) is smaller than the inside radius (r).
SOLUTION   Set parameters R and r with values so R >r

1750   **'RECTANGULAR BOSS :F = 0'**

DETECTION   Executing rectangular boss cycle.
CAUSE   F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1751   **'RECTANGULAR BOSS:S = 0'**

DETECTION   Executing rectangular boss cycle.
CAUSE   Roughing or finishing S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1752   **'RECTANGULAR BOSS :T = 0'**

DETECTION   Executing rectangular boss cycle.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a roughing or finishing tool T.

1753   **'RECTANGULAR BOSS:P = 0'**

DETECTION   Executing rectangular boss cycle.
CAUSE   P (boss depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1754   **'RECTANGULAR BOSS :tool diameter smaller than DELTA'**

DETECTION   Executing rectangular boss cycle.
CAUSE   The diameter of the roughing or finishing tool is smaller than the roughing or finishing pass in the main plane.
SOLUTION   Decrease the size of the milling pass in XY (DELTA) or machine with a tool that has a larger diameter.

1755   'RECTANGULAR BOSS :Finishing tool diameter smaller than delta'

DETECTION	Executing rectangular boss cycle.
CAUSE	The diameter of the finishing tool is smaller than the finishing stock in the main plane (delta).
SOLUTION	Choose a tool of a larger diameter to make the rectangular boss.

1756   'CIRCULAR BOSS :F = 0'

DETECTION	Executing circular boss cycle.
CAUSE	F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1757   'CIRCULAR BOSS:S = 0'

DETECTION	Executing circular boss cycle.
CAUSE	Roughing or finishing S programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1758   'CIRCULAR BOSS :T = 0'

DETECTION	Executing circular boss cycle.
CAUSE	No tool T has been programmed in the cycle.
SOLUTION	Assign a roughing or finishing tool T.

1759   'CIRCULAR BOSS:P = 0'

DETECTION	Executing circular boss cycle.
CAUSE	P (boss depth) programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1760   'CIRCULAR BOSS :R = 0'

DETECTION	Executing circular boss cycle.
CAUSE	R (boss radius) programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1761   'CIRCULAR BOSS :tool diameter smaller than DELTA'

DETECTION	Executing circular boss cycle.
CAUSE	The diameter of the roughing or finishing tool is smaller than the roughing or finishing pass in the main plane.
SOLUTION	Decrease the size of the milling pass in XY (DELTA) or machine with a tool that has a larger diameter.

1762   'CIRCULAR BOSS :Finishing tool diameter smaller than delta'

DETECTION	Executing circular boss cycle.
CAUSE	The diameter of the finishing tool is smaller than the finishing stock in the XY plane (delta).
SOLUTION	Choose a tool of a larger diameter to make the circular boss.

1763   'SURFACE MILLING: F = 0'

DETECTION	Executing the surface milling cycle.
CAUSE	F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1764   'SURFACE MILLING: S = 0'

DETECTION	Executing the surface milling cycle.
CAUSE	Roughing or finishing S programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1765   'SURFACE MILLING: T = 0'

DETECTION	Executing the surface milling cycle.
CAUSE	No tool T has been programmed in the cycle.
SOLUTION	Assign a roughing or finishing tool T.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 66 of 128

1766   **'SURFACE MILLING: P = 0'**

DETECTION   Executing the surface milling cycle.
CAUSE   P (surface milling depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1767   **'SURFACE MILLING: L = 0 and H = 0'**

DETECTION   Executing the surface milling cycle.
CAUSE   The two dimensions of surface milling (length and width) programmed in the cycle are 0.
SOLUTION   Set at least one of the dimensions L or H to a value greater than 0.

1768   **'POINT-TO-POINT PROFILE: F = 0'**

DETECTION   Executing point-to-point profiling cycle.
CAUSE   F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1769   **'POINT-TO-POINT PROFILE: S = 0'**

DETECTION   Executing point-to-point profiling cycle.
CAUSE   Roughing or finishing S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1770   **'POINT-TO-POINT PROFILE: T = 0'**

DETECTION   Executing point-to-point profiling cycle.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a roughing or finishing tool T.

1771   **'POINT-TO-POINT PROFILE: P = 0'**

DETECTION   Executing point-to-point profiling cycle.
CAUSE   P (point-to-point profile depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1772   **'POINT-TO-POINT PROFILE: Null profile'**

DETECTION   Executing point-to-point profiling cycle.
CAUSE   The first two points of the profile (P1 and P2) are the same and the profile is considered to be empty.
SOLUTION   Assign the coordinates of the desired points in the profile (from P1 to P12). The last point of the profile will be the one having the next one the same.

1773   **'POINT-TO-POINT PROFILE: Tool radius equal to or greater than the tangential entry/exit radius'**

DETECTION   Executing point-to-point profiling cycle.
CAUSE   The radius of one of the tools used in the cycle is greater than the tangential entry radius (Radius P1) or exit radius (Radius P12) programmed in the cycle.
SOLUTION   Increase the radius of the tangential entry/exit or execute the cycle with a tool of smaller radius.

1774   **'PROFILE: F = 0'**

DETECTION   Executing the profiling cycle.
CAUSE   F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1775   **'PROFILE: S = 0'**

DETECTION   Executing the profiling cycle.
CAUSE   Roughing or finishing S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1776   **'PROFILE: T = 0'**

DETECTION   Executing the profiling cycle.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a roughing or finishing tool T.

1777 'PROFILE: P = 0'

DETECTION	Executing the profiling cycle.
CAUSE	P (profile depth) programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1778 'PROFILE: (FINISHING): Tool's cutting length < P'

DETECTION	Executing the profiling cycle.
CAUSE	The cutting length of the finishing tool defined in the tool table is smaller than the profile depth (P) defined in the cycle.
SOLUTION	Choose as the finishing tool, a tool with greater cutting length.

1779 'SLOT MILLING: F = 0'

DETECTION	Executing the slot milling cycle.
CAUSE	F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1780 'SLOT MILLING: S = 0'

DETECTION	Executing the slot milling cycle.
CAUSE	Roughing or finishing S programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1781 'SLOT MILLING: T = 0'

DETECTION	Executing the slot milling cycle.
CAUSE	No tool T has been programmed in the cycle.
SOLUTION	Assign a roughing or finishing tool T.

1782 'SLOT MILLING: P = 0'

DETECTION	Executing the slot milling cycle.
CAUSE	P (profile depth) programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1783 'SLOT MILLING: L = 0'

DETECTION	Executing the slot milling cycle.
CAUSE	L (groove length) programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1784 'SLOT MILLING: tool diameter smaller than DELTA'

DETECTION	Executing the slot milling cycle.
CAUSE	The diameter of the roughing or finishing tool is smaller than the roughing or finishing pass in the XY plane.
SOLUTION	Decrease the size of the milling pass in XY (DELTA) or machine with a tool that has a larger diameter.

1785 'SLOT MILLING: tool diameter greater than the pocket'

DETECTION	Executing the slot milling cycle.
CAUSE	The diameter of the roughing or finishing tool is greater than the groove dimensions L or H.
SOLUTION	Choose a tool of a smaller diameter to make the groove.

1786 'SLOT MILLING: Finishing tool diameter smaller than delta'

DETECTION	Executing the slot milling cycle.
CAUSE	The diameter of the finishing tool is smaller than the finishing stock in the XY plane (delta).
SOLUTION	Choose a tool of a greater diameter to make the slot.

1787 'CIRCULAR POCKET: F = 0'

DETECTION	Executing circular pocket cycle.
CAUSE	F for roughing, finishing or penetrating in Z programmed in the cycle equal to 0.
SOLUTION	Assign a value greater than 0.

1788   'CIRCULAR POCKET: S = 0'

DETECTION   Executing circular pocket cycle.
CAUSE   Roughing or finishing S programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1789   'CIRCULAR POCKET: T = 0'

DETECTION   Executing circular pocket cycle.
CAUSE   No tool T has been programmed in the cycle.
SOLUTION   Assign a roughing or finishing tool T.

1790   'CIRCULAR POCKET: P = 0'

DETECTION   Executing circular pocket cycle.
CAUSE   P (pocket depth) programmed in the cycle equal to 0.
SOLUTION   Assign a value greater than 0.

1791   'CIRCULAR POCKET: tool diameter smaller than DELTA'

DETECTION   Executing circular pocket cycle.
CAUSE   The diameter of the roughing or finishing tool is smaller than the roughing or finishing pass in the XY plane.
SOLUTION   Decrease the size of the milling pass in XY (DELTA) or machine with a tool that has a larger diameter.

1792   'CIRCULAR POCKET: tool diameter greater than the pocket'

DETECTION   Executing circular pocket cycle.
CAUSE   The diameter of the roughing or finishing tool is greater than the R radius of the pocket.
SOLUTION   Choose a tool of a smaller diameter to make the pocket.

1793   'CIRCULAR POCKET: Finishing tool diameter smaller than delta'

DETECTION   Executing circular pocket cycle.
CAUSE   The diameter of the finishing tool is smaller than the finishing stock in the XY plane (delta).
SOLUTION   Choose a tool of a larger diameter to make the pocket.

1794-1800'Tool diameter = 0'

DETECTION   Executing a canned cycle.
CAUSE   The radius of the roughing or finishing tool is 0.
SOLUTION   Correct the value of the tool radius in the tool table.

1801-1803'Beta or theta < 0 or Beta or theta > 90'

DETECTION   Executing a pocket.
CAUSE   In a pocket, a lateral penetration angle has programmed for roughing (beta) or for finishing (theta) that is not between 0º and 90º.
SOLUTION   Correct the values of beta or theta.

1804   'G87: Depth = 0'

DETECTION   Executing a G87 pocket.
CAUSE   The programmed I coordinate is the same as the Z coordinate plus the D distance. If Z has not been programmed, the I coordinate is the same as the current position plus the D distance. This means that the pocket has no depth.
SOLUTION   Modify the parameters involved I, Z and/or D.

1805   'G87: tool diameter greater than the pocket'

DETECTION   Executing a G87 pocket.
CAUSE   The tool radius is greater than the J and/or K dimensions of the pocket.
SOLUTION   Select a tool with a smaller radius.

1806   'G87: tool diameter smaller than L'

DETECTION   Executing a G87 pocket.
CAUSE   The tool diameter is smaller than the finishing stock L.
SOLUTION   Select a tool with a larger radius.

1807 'G87: Tool diameter = 0'

DETECTION	Executing a G87 pocket.
CAUSE	Tool with radius 0.
SOLUTION	Modify the value of the tool radius in the tool table.

1808 'G87: Tool missing'

DETECTION	Executing a G87 pocket.
CAUSE	Tool missing at the spindle.
SOLUTION	Load a tool in the spindle before calling the cycle.

1809 'G87: tool diameter smaller than C'

DETECTION	Executing a G87 pocket.
CAUSE	The tool diameter is smaller than the machining pass (C) in the main plane.
SOLUTION	Select a tool of larger diameter or reduce C.

1810 'G88: Depth = 0'

DETECTION	Executing a G88 pocket.
CAUSE	The programmed I coordinate is the same as the Z coordinate plus the D distance. If Z has not been programmed, the I coordinate is the same as the current position plus the D distance. This means that the pocket has no depth.
SOLUTION	Modify the parameters involved I, Z and/or D.

1811 'G88: tool diameter greater than the pocket'

DETECTION	Executing a G88 pocket.
CAUSE	The tool radius is greater than the J dimension of the pocket.
SOLUTION	Select a tool with a smaller radius.

1812 'G88: tool diameter smaller than L'

DETECTION	Executing a G88 pocket.
CAUSE	The tool diameter is smaller than the finishing stock L.
SOLUTION	Select a tool with a larger radius.

1813 'G88: Tool diameter = 0'

DETECTION	Executing a G88 pocket.
CAUSE	Tool with radius 0.
SOLUTION	Modify the value of the tool radius in the tool table.

1814 'G88: Tool missing'

DETECTION	Executing a G88 pocket.
CAUSE	Tool missing at the spindle.
SOLUTION	Load a tool in the spindle before calling the cycle.

1815 'G88: tool diameter smaller than C'

DETECTION	Executing a G88 pocket.
CAUSE	The tool diameter is smaller than the machining pass (C) in the main plane.
SOLUTION	Select a tool of larger diameter or reduce C.

1816 'The X parameter must be multiple of the I parameter'

DETECTION	Executing a multiple machining cycle (G160-G165).
CAUSE	The X parameter must be multiple of the I parameter.
SOLUTION	Assign a correct value to the cycle parameter.

1817 'One of parameters I and K must be programmed'

DETECTION	Executing a multiple machining cycle (G160-G165).
CAUSE	One of the parameters I (machining pass), K (number of machining operations) has not been programmed in a multiple machining cycle.
SOLUTION	Assign a correct value to the cycle parameter.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 70 of 128

1818   'J = 0'

DETECTION   Executing a multiple machining cycle (G160-G165).

CAUSE   The J parameter (ordinate pass between machining operations) has been programmed with a 0 value in a multiple machining cycle.

SOLUTION   Assign a correct value to the cycle parameter.

1819   'K = 0'

DETECTION   Executing a multiple machining cycle (G160-G165).

CAUSE   The K parameter (number of machining operations) has been programmed with a 0 value in a multiple machining cycle.

SOLUTION   Assign a correct value to the cycle parameter.

1820   'Two of parameters X, I and K must be programmed'

DETECTION   Executing a multiple machining cycle (G160-G165).

CAUSE   In a multiple machining cycle, two of the following parameters have not been programmed: X (length), I (machining pass), K (number of machining operations).

SOLUTION   Assign a correct value to the cycle parameter.

1821   'B must be multiple of I'

DETECTION   Executing a multiple machining cycle (G164).

CAUSE   The I parameter (machining pass) must be contained an entire number of times in B (angular travel).

SOLUTION   Assign a correct value to the cycle parameter.

1822   'The Y parameter must be multiple of the J parameter'

DETECTION   Executing a multiple machining cycle (G160-G165).

CAUSE   The J parameter (machining pass) must be contained an entire number of times in Y (length in ordinates).

SOLUTION   Assign a correct value to the cycle parameter.

1823   'Two of parameters Y, J and D must be programmed'

DETECTION   Executing a multiple machining cycle (G160-G165).

CAUSE   In cycles G161 and G162, two of the following parameters have not been programmed: Y (length in ordinates), J (machining pass), D (number of machining operations).

SOLUTION   Assign a correct value to the cycle parameter.

1824   'One of parameters I and K must be programmed'

DETECTION   Executing a multiple machining cycle (G160-G165).

CAUSE   One of the these two parameters have not been programmed, I (machining pass), K (number of machining operations).

SOLUTION   Assign a correct value to the cycle parameter.

1825   'One of parameters I and A must be programmed'

DETECTION   Executing the multiple machining cycle (G165).

CAUSE   One of the these two parameters have not been programmed, I (angular pass), A (number of machining operations).

SOLUTION   Assign a correct value to the cycle parameter.

1826   'Parameter I multiplied by K must be equal to 360'

DETECTION   Executing a multiple machining cycle G163.

CAUSE   In a multiple machining in circle (G163), the pass (I) multiplied by the number of machining operations (K) is not equal to 360º.

SOLUTION   Assign a correct value to the cycle parameter.

1827   'Parameters X and Y are equal to 0'

DETECTION   Executing a multiple machining cycle (G163-G165).

CAUSE   The parameters defining the center of the arc of the multiple machining have been programmed with a value of 0.

SOLUTION   Assign a correct value to the cycle parameter.

1828 '360 must be multiple of parameter I'

DETECTION	Executing a multiple machining cycle in circle (G163).
CAUSE	The angular pass (I) must be multiple of 360º.
SOLUTION	Assign a correct value to the cycle parameter.

1829 'I = 0'

DETECTION	Executing a multiple machining canned cycle (G160-G165).
CAUSE	Parameter I (distance between consecutive machining passes) has been assigned a value of 0.
SOLUTION	Assign a value other than 0.

1830 'Tool diameter = 0'

DETECTION	It does not exist.
CAUSE	The diameter of the tool used to execute the cycle is 0.
SOLUTION	Assign a value other than 0 to the tool diameter.

1831 'G82: C = 0'

DETECTION	Execution of the canned cycle G82.
CAUSE	The C parameter of that cycle has been programmed with a value of 0. The C parameter indicates up to what distance of the previous drilling peck it will move in rapid before doing the next peck.
SOLUTION	Assign to C a value other than 0.

1832 'ROUGHING: I = 0'

DETECTION	Executing a canned cycle.
CAUSE	The I parameter (maximum penetrating pass) has been defined with a value of 0.
SOLUTION	Assign to I a value other than 0 and less than the roughing tool's cutting length.

1833 'FINISHING: N = 0 and undefined tool's cutting length.

DETECTION	Executing a canned cycle.
CAUSE	N (number of finishing passes) has been programmed with a value of 0 and the cutting length of the finishing tool has not been defined.
SOLUTION	Program the N value.

1834 'FINISHING: Z delta greater than the tool's cutting length'

DETECTION	Executing a canned cycle.
CAUSE	The cycle parameter dz (penetrating stock or excess material) has been programmed with a value greater than the cutting length of the finishing tool.
SOLUTION	Program a smaller dz or use another tool.

1835 'Pocket with islands: Wrong safety Z'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The safety Z (Zs) is programmed inside the part.
SOLUTION	Program it outside the part.

1836 'Pocket with islands: Wrong Z profile'

DETECTION	Execution of a 3D pocket cycle with islands.
CAUSE	One of the programmed profiles in Z cannot be machined.
SOLUTION	Check that the profiles in Z can be machined.

1837 'Pocket with islands: Profile in XY intersects itself'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	One of the profiles in XY forms more than one closed profile.
SOLUTION	Check that the profiles in XY only have one point in common, that is precisely its starting point.

1838 'Pocket with islands: The XY profile is not closed'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	One of the XY profiles is not a closed profile; in other words, its initial and final points are not the same.
SOLUTION	Check that all the profiles in the plane are closed.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 72 of 128

1839 'Pocket with islands: Wrong intersection of XY profiles'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	Two XY profiles have at least some straight or arc section in common.
SOLUTION	Check that the intersections of all the XY profiles only take place on points, never on sections.

1840 'Pocket with islands: Not enough memory to resolve it'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	There is no system memory left to continue resolving the pocket.
SOLUTION	

1841 'Wrong tool position before the canned cycle'

DETECTION	Executing a canned cycle.
CAUSE	Before executing the canned cycle, the coordinate of the tool tip along the longitudinal axis (Z) is located between the reference plane and the surface of the part.
SOLUTION	Move the tool along the longitudinal axis (Z) to get it out of the safety area delimited by the reference plane and the part.

1842 'Tool diameter smaller than DELTA'

DETECTION	Executing a canned cycle.
CAUSE	The diameter of the finishing tool is smaller than the finishing pass (DELTA).
SOLUTION	Choose a tool whose diameter is greater than the finishing pass or decrease the finishing pass (DELTA).

1843 'Pocket with islands: Wrong roughing tool radius'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The radius of the tool programmed in the roughing operations or roughing at the bottom has a value of 0. Or the radius of the tool is too large for the geometry of the pocket, taking into account the lateral finishing stock to be left.
SOLUTION	If the tool radius is not 0, check that the value programmed in the finishing stock is correct. In this case, a tool must be programmed with a smaller radius than the current one.

1844 'Pocket with islands: Wrong finishing tool radius'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The tool radius programmed in the finishing pass has a value of 0 or it is too large for the geometry of the pocket.
SOLUTION	If the tool radius is no t0, program a tool whose radius is smaller than the current one.

1845 'G165: Chord length (I) greater than the diameter'

DETECTION	Execution of the canned cycle G165.
CAUSE	The length of the chord defined in parameter I is greater than the diameter of the circle.
SOLUTION	Assign a correct value to the I parameter.

1846 'Zero arc radius'

DETECTION	Execution of multiple machining in an arc.
CAUSE	In the cycle editor, an arc has been programmed with a zero radius or an arc whose center coordinates Xa and Ya are the same as the XY coordinates of the cycle that is going to be repeated.
SOLUTION	Program an arc whose radius is not zero; in other words, assign other values to parameters Xa, Ya or R.

1847 'RECTANGULAR BOSS :Q = 0'

DETECTION	Execution of rectangular boss canned cycle.
CAUSE	The material to be removed in the work plane to make the boss has not been defined.
SOLUTION	Assign to Q a value other than 0.

1848 'CIRCULAR BOS S :Q = 0'

DETECTION	Execution of circular boss canned cycle.
CAUSE	The material to be removed in the work plane to make the boss has not been defined.
SOLUTION	Assign to Q a value other than 0.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 73 of 128

1849 'RECTANGULAR POCKET: Finishing stock DELTA greater than the pocket'

DETECTION	Executing a rectangular pocket.
CAUSE	The defined pocket cannot be executed because half the shorter length of the pocket minus the tool radius is smaller than the finishing stock in the work plane (DELTA).
SOLUTION	Decrease the value of DELTA.

1850 'Lx must be multiple of Ix'

DETECTION	Executing cycles G160 and G161.
CAUSE	The length of the straight line or the rectangle must be multiple of the distance between machining operations Ix. Otherwise, the machining operations will not be equidistant.
SOLUTION	Modify the values of Lx and Ix.

1851 'Ly must be multiple of Iy'

DETECTION	Executing cycles G160 and G161.
CAUSE	The length of the straight line or the rectangle must be multiple of the distance between machining operations Iy. Otherwise, the machining operations will not be equidistant.
SOLUTION	Modify the values of Ly and Iy.

1852 'No tool has been programmed'

DETECTION	Executing a calibration cycle (#PROBE1/2) or measuring cycle (#PROBE 3-9).
CAUSE	No tool has been loaded into the spindle before executing the cycle.
SOLUTION	Execute the cycle with a tool in the spindle (#PROBE1) or with a probe (#PROBE 2-9).

1853 'The probe signal has not been received'

DETECTION	Executing a #PROBE cycle.
CAUSE	While executing a #PROBE cycle, no point has been probed during a probing move.
SOLUTION	Modify the geometric parameters of the cycle so the probe actually touches the tool (#PROBE1) or the part (#PROBE 2-9).

1854 'The diameter J must be greater than zero'

DETECTION	Executing a #PROBE 2 or a #PROBE 9 cycle
CAUSE	A negative diameter has been programmed.
SOLUTION	Assign a positive value.

1855 'Tool diameter greater than that of the hole'

DETECTION	Executing a #PROBE 2 or a #PROBE 9 cycle
CAUSE	The programmed diameter of the hole to be measured or that of the hole used to calibrate the probe is smaller than the diameter of the probe used in measurement.
SOLUTION	Assign a proper value.

1856 'Wrong withdraw distance E'

DETECTION	Executing a #PROBE 2 or a #PROBE 9 cycle
CAUSE	The withdraw distance cannot be negative and cannot exceed the diameter of the hole.
SOLUTION	Assign to E a value between 0 and the diameter of the hole.

1857 'There is no tool offset selected'

DETECTION	Execution of the calibration cycle (#PROBE1/2).
CAUSE	A tool calibration cycle or probing cycle has been executed without previously loading a tool offset (D).
SOLUTION	Program the tool offset D before calling the cycle.

1859 'The active plane for calibrating the probe must be G17, G18 or G19'

DETECTION	Executing the #PROBE 2 cycle.
CAUSE	When starting the probe calibration cycle, G20 is active.
SOLUTION	Program G17 (XY plane), G18 (ZX plane) or G19 (YZ plane) before calling the cycle.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 74 of 128

1860 'Program K = 0/1/2'

DETECTION	Executing the #PROBE 3 cycle (surface measuring cycle).
CAUSE	Parameter K only admits the values 0, 1, 2.
SOLUTION	Parameter K defines the axis to be used to measure the surface. Assign to K one of the following values:

- 0 for the abscissa axis.
- 1 for the ordinate axis.
- 2 for the longitudinal axis.

1861 'The safety distance B must be greater than zero'

DETECTION	Execution of the calibration or measuring cycle (#PROBE).
CAUSE	Parameter B has been programmed with a negative value.
SOLUTION	Assign a positive value.

1862 '#PROBE 1: not allowed when G20 is active'

DETECTION	Executing the #PROBE 1 cycle.
CAUSE	When starting the probe calibration cycle, G20 is active.
SOLUTION	Program G17 (XY plane), G18 (ZX plane) or G19 (YZ plane) before calling the cycle.

1863 'Tool worn out'

DETECTION	Execution of the tool calibration cycle #PROBE1.
CAUSE	The cycle has detected a tool that is worn out. The measured tool wear is greater than the allowed tolerance (parameter L or M).
SOLUTION	Change the tool and execute the cycle again.

1864 '#PROBE 1: Parameters U, V, W must be greater than X, Y, Z'

DETECTION	Execution of the tool calibration cycle #PROBE1.
CAUSE	In the cycle, the coordinates defining the position of the probe have been defined wrong. UVW are the top coordinates and XYZ are the bottom coordinates.
SOLUTION	Assign the parameters so U > X, V > Y, W > Z.

1865 'Pocket with islands: Roughing T missing'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The tool number (T) programmed in the roughing operation does not exist in the tool table.
SOLUTION	Insert the tool in the table or program another tool number that is in the table.

1866 'Pocket with islands: Wrong roughing F'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The machining feedrate (F) programmed for the roughing operation has a value of 0.
SOLUTION	Program F with a value other than 0.

1867 'Pocket with islands: Wrong roughing S'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The spindle speed (S) programmed for the roughing operation has a value of 0.
SOLUTION	Program S with a value other than 0.

1868 'Pocket with islands: Finishing T missing'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The tool number (T) programmed in the finishing operation does not exist in the tool table.
SOLUTION	Insert the tool in the table or program another tool number that is in the table.

1869 'Pocket with islands: Wrong finishing F'

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The machining feedrate (F) programmed for the finishing operation has a value of 0.
SOLUTION	Program F with a value other than 0.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 75 of 128

1870   **'Pocket with islands: Wrong finishing S'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The spindle speed (S) programmed for the finishing operation has a value of 0.
SOLUTION	Program S with a value other than 0.

1871   **'Pocket with islands: Wrong roughing pass'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The machining pass (D) programmed in the roughing operation is greater than the tool diameter.
SOLUTION	Program a roughing pass value less than or equal to the tool diameter.

1872   **'Pocket with islands: Wrong finishing pass'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The machining pass (D) programmed in the finishing operation at the bottom of the 2D pocket is greater than the tool diameter. Or the finishing pass (e) in 3D pockets has been programmed with a value of 0.
SOLUTION	Program the correct value in the corresponding case.

1873   **'Pocket with islands: Wrong side stock (excess material)'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The value of the side stock (d) programmed for the finishing operation is greater than the diameter of the tool to be used for that operation.
SOLUTION	Program a side stock less than or equal to the tool diameter.

1874   **'Pocket with islands: Wrong depth'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The value of the programmed pocket depth (P) is 0.
SOLUTION	Program it with a value other than 0.

1875   **'Pocket with islands: XY profile does not exist'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The programmed XY profile does not exist.
SOLUTION	Change the name of the profile with an existing one or generate the programmed profile.

1876   **'Pocket with islands: Wrong XY profile'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	Geometry of the XY profile programmed wrong.
SOLUTION	Enter the profile editor to edit the profile and check what part of the geometry is programmed wrong.

1877   **'Pocket with islands: Z profile does not exist'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The programmed Z profile does not exist.
SOLUTION	Change the name of the profile with an existing one or generate the programmed profile.

1878   **'Pocket with islands: Wrong finishing tool tip (nose) radius'**

DETECTION	Execution of a 3D pocket cycle with islands.
CAUSE	The tip radius (Rp) of the finishing tool is greater than its radius (R).
SOLUTION	Program a tool tip radius smaller than or equal to the radius.

1879   **'Pocket with islands: Wrong semi-finishing tool radius'**

DETECTION	Execution of a 2D or 3D pocket cycle with islands.
CAUSE	The possible causes are: • The radius of the finishing tool for the bottom of 2D pockets has a value of 0 or it is too large for the geometry of the pocket. • The radius of the semi-finishing tool in 3D pockets has a value of 0.
SOLUTION	Program the correct value in the corresponding case.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 1000-1999

Page 76 of 128

1880 'Pocket with islands: Semi-finishing T missing'

DETECTION   Execution of a 3D pocket cycle with islands.

CAUSE   The tool number (T) programmed in the semi-finishing operation does not exist in the tool table.

SOLUTION   Insert the tool in the table or program another tool number that is in the table.

1881 'Pocket with islands: Wrong semi-finishing F'

DETECTION   Execution of a 3D pocket cycle with islands.

CAUSE   The machining feedrate (F) programmed for the semi-finishing operation has a value of 0.

SOLUTION   Program F with a value other than 0.

1882 'Pocket with islands: Wrong semi-finishing S'

DETECTION   Execution of a 2D or 3D pocket cycle with islands.

CAUSE   The spindle speed (S) programmed for the semi-finishing operation has a value of 0.

SOLUTION   Program S with a value other than 0.

1883 'Pocket with islands: Pocket already in execution'

DETECTION   Execution of a 2D or 3D pocket cycle with islands.

CAUSE   A 2D or 3D pocket cannot be simulated or executed because it is already in execution or simulation.

SOLUTION   Wait for it to end, to execute it again.

1884 'Pocket with islands: Arc programmed wrong in the profile'

DETECTION   Execution of a 2D or 3D pocket cycle with islands.

CAUSE   An arc of one of the profiles making up the geometry of the pocket is programmed wrong. This could be due to the following causes:
              • It has not been edited in the cycle itself.
              • The profile has been modified with a program other than the profile editor.
              • Machine parameter DIAMPROG of the axes of the profile is not properly set.

SOLUTION   Enter the cycle screen an edit each profile of the pocket to find the arc that is programmed wrong. Once the arc and the wrong profile have been found, go to the editor and modify the arc.

1885 'Pocket with islands: Wrong pocket geometry'

DETECTION   Execution of a 2D or 3D pocket cycle with islands.

CAUSE   Some of the profiles making up the pocket has some path programming error (rounding, chamfer, etc.).

SOLUTION   Enter the cycle screen an edit each profile of the pocket to find the error. Once found the error and which profile it's in, go to the editor and modify it.

1886 'Pocket with islands: Wrong roughing tool's penetration angle'

DETECTION   Execution of a 2D or 3D pocket cycle with islands.

CAUSE   The value of the penetration angle (Ae) of the tool programmed in the roughing operation is smaller than or equal to 0º or greater than 90º.

SOLUTION   Program a value greater than 0º and smaller than or equal to 90º.

1887 'Pocket with islands: Wrong penetration angle of the semi-finishing tool'

DETECTION   Execution of a 3D pocket cycle with islands.

CAUSE   The value of the penetration angle (Ae) of the tool programmed in the semi-finishing operation is smaller than or equal to 0º or greater than 90º.

SOLUTION   Program a value greater than 0º and smaller than or equal to 90º.

Errors 1000-1999

ERRORS 2000-2999

2000   **'Tool radius greater than the arc radius'**

DETECTION	During execution.
CAUSE	The tool radius is greater than the radius of the arc to be machined.
SOLUTION	Use a tool with a smaller radius.

2001   **'Profile damaged by tool radius compensation'**

DETECTION	During execution.
CAUSE	The tool radius is too large for the profile to be machined and, consequently, would damage the profile.
SOLUTION	Use a tool with a smaller radius.

2002   **'The first block of the linear compensation must be linear'**

DETECTION	During execution.
CAUSE	After activating tool radius compensation (G41 or G42), the next motion block is a circular block. Tool radius compensation cannot begin in a circular block.
SOLUTION	Tool radius compensation must begin in a linear block. Therefore, the motion block that goes after G41-G42 must be a linear motion block.

2003   **'Tool radius too large in consecutive arcs'**

DETECTION	During execution.
CAUSE	When machining two consecutive arcs that make up a loop (intersect each other) The tool radius is too large for machining the inside of the loop.
SOLUTION	Use a tool with a smaller radius.

2004   **'Too many motionless blocks between blocks that have tool radius compensation'**

DETECTION	During execution.
CAUSE	While tool radius compensation is active, too many motionless blocks (parameter assignments P, variables, etc.) have been programmed between motion blocks.
SOLUTION	Reduce the number of motionless blocks programmed. To do this, several of these blocks may be combined into a single block.

2005   **'The last block of the linear compensation must be linear'**

DETECTION	During execution.
CAUSE	After canceling tool radius compensation (G40), the next motion block is a circular block. Tool radius compensation cannot end in a circular block.
SOLUTION	Tool radius compensation must end in a linear block. Therefore, the motion block that goes after G40 must be a linear motion block.

2006   **'Tool radius compensation (G41/G42) must be changed on a linear path'**

DETECTION	During execution.
CAUSE	The tool radius compensation has been changed and the next motion block is a circular block. In other words, G42 has been programmed while G41is active or vice versa. The tool radius compensation cannot be changed if the next motion block is an arc.
SOLUTION	Program a linear motion block after (G41-G42).

2007 **'While G138 is active, G40 is not allowed after the first compensation block'**

DETECTION	During execution.
CAUSE	Tool radius cancellation (G40) has been programmed while the first compensation block is being processed in a direct selection (G138).
SOLUTION	Using direct compensation selection (G138) requires an additional block of motion in the plane in order to complete the beginning of the compensation and then be able to cancel it. Another option is to use the indirect compensation selection (G139) that does not need an additional block.

2008 **'While G138 is active, no G41/G42 change is allowed after the first compensation block'**

DETECTION	During execution.
CAUSE	While processing the first compensation block of a direct selection (G138), a change from G41 to G42 or viceversa has been programmed.
SOLUTION	Using direct compensation selection (G138) requires an additional block of motion in the plane in order to complete the beginning of the compensation. The type of compensation may be changed after this movement. Another option is to use the indirect compensation selection (G139) that does not need an additional block.

2010 **'Tool radius too large when calculating tool radius compensation between arcs'**

DETECTION	During execution.
CAUSE	The tool radius is too large for the programmed arc.
SOLUTION	The possible solutions are: • Program a smaller tool radius to machine the programmed arc. • Eliminate the arc from the programmed profile. • Program a larger arc.

2011 **'Compensation movement on circular path (profile damaged)'**

DETECTION	During execution.
CAUSE	The tool radius is too large for the programmed arc.
SOLUTION	The possible solutions are: • Program a smaller tool radius to machine the programmed arc. • Eliminate the arc from the programmed profile. • Program a larger arc.

2013 **'Different tool radius between previous path and an arc'**

DETECTION	During execution.
CAUSE	The error comes up because when compensating an arc, the tool radius has changed with respect to the previous block.
SOLUTION	Do not change the tool radius when compensating an arc.

2016 **'The profile has been changed to avoid a collision'**

DETECTION	During execution.
CAUSE	The process of detecting collisions or steps during tool compensation has detected a tool movement that could damage the programmed profile and it has eliminated it.
SOLUTION	Depending on the collision detected, it may be solved by using a tool of a smaller radius, programming the profile in a different way or simply accepting the change proposed by the collision detecting process.

2017 **'Programming not allowed while collision detection is active'**

DETECTION	During execution.
CAUSE	While detecting collisions, certain processes like home search, probing, etc. are not allowed.
SOLUTION	Do not activate collision detection if any of these processes has been programmed.

2100 **'Positive software limit overrun in G5-G60'**

DETECTION	During execution.
CAUSE	The path generated by the CNC to round the corner is out of the active travel limits.
SOLUTION	Program without G50 or G7 rounding .

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 2000-2999

Page 80 of 128

2101 **'Negative software limit overrun in G5-G60'**

DETECTION   During execution.
CAUSE   The path generated by the CNC to round the corner is out of the active travel limits.
SOLUTION   Program without G50 or G7 rounding .

2102 **'Programming instruction #ROUNDPAR with parameter too small'**

DETECTION   During execution.
CAUSE   The error parameter is too small to be processed.
SOLUTION   Program the instruction with a value greater than 20 microns.

2103 **'Programming instruction #ROUNDPAR with the wrong type of parameter'**

DETECTION   During execution.
CAUSE   The type programmed for rounding is wrong.
SOLUTION   Program valid values according to the manual.

2106 **'Internal error in HSC mode'**

DETECTION   During execution.
CAUSE   An error has occurred when calculating the path to work in HSC mode.
SOLUTION   Cancel the HSC mode in the blocks causing the error.

2108 **'HSC mode change not allowed'**

DETECTION   During execution.
CAUSE   It has been programmed in HSC mode while another one was active.
SOLUTION   Cancel the HSC mode before programming another one.

2109 **'HSC: Contour error too small'**

DETECTION   During execution.
CAUSE   The contour error is too small to be processed.
SOLUTION   Program the instruction with a value greater than 20 microns.

2110 **'Positive software limit overrun in HSC mode'**

DETECTION   During execution.
CAUSE   The path generated by the CNC to work in HSC mode is out of the active travel limits.
SOLUTION   Cancel the HSC mode in the block or blocks that cause the error.

2111 **'Negative software limit overrun in HSC mode'**

DETECTION   During execution.
CAUSE   The path generated by the CNC to work in HSC mode is out of the active travel limits.
SOLUTION   Cancel the HSC mode in the block or blocks that cause the error.

2112 **'Undetermined initial tangent in spline'**

DETECTION   During execution.
CAUSE   Either the starting point is not the same as the previous one, thus the tangent is undetermined or only one axis of the tangent has been programmed.
SOLUTION   Program a prior entry point to the spline.

2113 **'Undetermined final tangent in spline'**

DETECTION   During execution.
CAUSE   Either the final point is not the same as the previous one, thus the tangent is undetermined or only one axis of the tangent has been programmed.
SOLUTION   Program an exit point after the spline.

2114 **'Spline activating command not allowed'**

DETECTION   During execution.
CAUSE   The spline mode activation has been programmed without finishing a previous spline.
SOLUTION   Cancel a spline mode before defining a new one.

2115 **'Programming instruction #ASPLINE with wrong parameter'**

DETECTION   During execution.
CAUSE   The type of tangency is wrong.
SOLUTION   Program one of the types indicated in the manual.

2116   'Error when activating the spline'

DETECTION   During execution.
CAUSE   An attempt has been made to activate a spline mode without canceling the previous one.
SOLUTION   Before activating it, cancel the previous one.

2118   'SPLINE: wrong type of spline'

DETECTION   During execution.
CAUSE   The type of spline programmed is not allowed.
SOLUTION   Program the type of spline as indicated in the manual.

2119   'SPLINE: The type cannot be changed while splines are active'

DETECTION   During execution.
CAUSE   An attempt has been made to change the type of spline while another one was active.
SOLUTION   Cancel the spline before programming another one of another type.

2121   'Circular paths (arcs) are not allowed while splines are active'

DETECTION   During execution.
CAUSE   An arc has been programmed while splines were active.
SOLUTION   Cancel the spline mode before programming the arc.

2122   'Polynomial blocks are not allowed while splines are active'

DETECTION   During execution.
CAUSE   A polynomial movement has been programmed while splines were active.
SOLUTION   Cancel the spline mode before programming the polynomial.

2123   'Programming not allowed while splines are active'

DETECTION   During execution.
CAUSE   A change of the coordinate system has been programmed while splines were active.
SOLUTION   Cancel the spline mode before programming the change of coordinate system.

2124   'SPLINE: It requires at least two main axes'

DETECTION   During execution.
CAUSE   The command spline has been programmed while having only one axis in the channel.
SOLUTION   Program the movements without spline mode.

2125   'SPLINE: error when calculating the tangent'

DETECTION   During execution.
CAUSE   A point has been programmed more than once.
SOLUTION   Program different points.

2126   'Positive software limit overrun in the spline'

DETECTION   During execution.
CAUSE   The path generated by the CNC to work in SPLINE mode is out of the active travel limits.
SOLUTION   Cancel the SPLINE mode in the block or blocks that cause the error.

2127   'Negative software limit overrun in the spline'

DETECTION   During execution.
CAUSE   The path generated by the CNC to work in SPLINE mode is out of the active travel limits.
SOLUTION   Cancel the SPLINE mode in the block or blocks that cause the error.

2128   'Helical path programmed wrong'

DETECTION   During execution.
CAUSE   One of the parameters of the helical path is missing or is wrong. The pitch has been programmed, but not the depth, or both the pitch and then end point are missing or the depth, the pitch and the end point are not compatible with each other.
SOLUTION   Program the end point compatible with the pitch and the depth. If they are full turns, program the pitch and the depth.

2129 'Error in the RTCP transformation'

DETECTION   During execution.
CAUSE   RTCP has been programmed, but the channel does not have all the axes needed for the transformation.
SOLUTION   Before activating the RTCP, configure the channel with all the necessary axes.

2130 'Error in the incline plane transformation'

DETECTION   During execution.
CAUSE   An incline plane has been programmed and the channel does not have three axes. Three axes are required for the transformation.
SOLUTION   Before activating the incline plane, configure the channel with all the necessary axes.

2131 'Internal error in C axis mode'

DETECTION   During execution.
CAUSE   C axis transformation not possible for the block.
SOLUTION   Eliminate the block that caused the error. Send the information to FAGOR.

2133 'Master axis missing'

DETECTION   During execution.
CAUSE   The master axis of the coupling is not available in the channel.
SOLUTION   Program the coupling with axes from the channel.

2134 'Slave axis missing'

DETECTION   During execution.
CAUSE   The slave axis of the coupling is not available in the channel.
SOLUTION   Program the coupling with axes from the channel.

2135 'Zero maximum spindle speed programmed'

DETECTION   During execution.
CAUSE   In constant surface speed mode, the maximum spindle speed has not been programmed.
SOLUTION   Program the maximum rpm of the spindle.

2136 'Zero maximum cutting speed programmed'

DETECTION   During execution.
CAUSE   In constant surface speed mode, the cutting speed has not been programmed.
SOLUTION   Program the cutting speed.

2137 'FACEAXIS has not been defined'

DETECTION   During execution.
CAUSE   'No face axis has been defined'
SOLUTION   Set machine parameter FACEAXIS = TRUE for the face axis.

2138 'Threading axis missing'

DETECTION   During execution.
CAUSE   The threading axis is not in the channel.
SOLUTION   Do not program a threading operation if all the axes are not available or configure the axes of the channel to be able to make the thread.

2139 'Threading movement not programmed'

DETECTION   During execution.
CAUSE   No movement has been programmed for the threading axis.
SOLUTION   Program the movement for the threading axis.

2140 'Wrong parameter in programming instruction #SLOPE'

DETECTION   During execution.
CAUSE   A parameter of the programming instruction #SLOPE is wrong.
SOLUTION   Refer to the manual for the right values to program it:
             #SLOPE [type(0..2), jerk(0..2), accel(0,2), mov(0,1)]

2141   'Circular path not allowed (less than 2 axes)'

DETECTION   During execution.
CAUSE   Not possible if there aren't at least two axes in the channel.
SOLUTION   Configure the channel with at least two axes to be able to work with circular
interpolation.

2142   'Programming not allowed with Hirth axis (axes)'

DETECTION   During execution.
CAUSE   A command has been programmed that is incompatible with a Hirth axis.
SOLUTION   Check the program.

2143   'The tangent to the spline cannot be calculated'

DETECTION   During execution.
CAUSE   'The tangent to unselect the spline cannot be calculated'
SOLUTION   Unselect another point or program another tangent for the unselection.

2144   'Error when generating the spline'

DETECTION   During execution.
CAUSE   The first block has not been programmed when activating the tangent spline.
SOLUTION   Program a movement before activating the tangent spline.

2145   'End of the program without canceling splines'

DETECTION   During execution.
CAUSE   The end of the program has been reached while the spline mode was active.
SOLUTION   Program the instruction SPLINE OFF before ending the program.

2146   'The spline cannot be canceled'

DETECTION   During execution.
CAUSE   Enough blocks have not been read to cancel the spline.
SOLUTION   Do not program a spline when programming only one block.

2147   'Splines cannot be programmed in HSC mode'

DETECTION   During execution.
CAUSE   The command SPLINE ON has been programmed while the HSC mode was active.
SOLUTION   First cancel the HSC mode with #HSC OFF or do not program the spline mode.

2148   'Too many motionless blocks between blocks in spline mode'

DETECTION   During execution.
CAUSE   The motion blocks are too far apart.
SOLUTION   Compact the intermediate blocks by grouping as many as possible in the same block
or cancel the spline first and activate it again afterwards.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 2000-2999

Page 84 of 128

ERRORS 3000-3999

3000 'An arc cannot be programmed with a slave axis'

DETECTION   During execution.
CAUSE   The possible causes are:
             1. An attempt has been made to separately move an axis that is coupled to another axis.
             2. An attempt has been made to separately move an axis that is associated as Gantry to another axis.
SOLUTION   A slave axis cannot move separately.To move a slave axis, move the master axis it is associated with.

3001 'An axis cannot be moved in DRO mode'

DETECTION   During execution.
CAUSE   An attempt has been made to move an axis that is in DRO mode.
SOLUTION   Do not try to move a DRO axis or set the axis so it is no longer a DRO axis.

3002 'Positive software travel limit overrun'

DETECTION   While repositioning an axis.
CAUSE   An attempt has been made to reposition the axis at a point that is beyond the software limits. It is not possible to reach the point to resume the execution of the program.
SOLUTION   Keep the axes within the software limits.

3003 'Negative software travel limit overrun'

DETECTION   While repositioning an axis.
CAUSE   An attempt has been made to reposition the axis at a point that is beyond the software limits. It is not possible to reach the point to resume the execution of the program.
SOLUTION   Keep the axes within the software limits.

3005 'Error of the position control when initializing the probing process'

DETECTION   When initializing the probing process
CAUSE   It is an enabling safety feature of the probing process in all the programmed axes. The programmed axes are not valid or are not available.
SOLUTION   Check that the programmed axes are valid and available.

3007 'The probe signal has been received before the movement'

DETECTION   After detecting the probe signal.
CAUSE   Probing process enabled without having programmed G100.
SOLUTION   Contact your supplier.

3008 'Error in the probing process'

DETECTION   After detecting the probe signal.
CAUSE   It is an enabling safety feature of the probing process in all the programmed axes.
SOLUTION   Check that the programmed axes are valid and available.

3010 'The axis has not been defined as probe (PROBEAXIS)'

DETECTION   During execution.
CAUSE   An attempt has been made to probe with an axis that is not defined as an axis involved in the probing movement.
SOLUTION   Change the parameter PROBEAXIS of the axis to probe with.

Errors 3000-3999

3011 'Too many axis programmed as probe'

DETECTION   When executing a probing block.

CAUSE   The number of axes programmed in the probing process exceeds the maximum number of axes of the channel.

SOLUTION   Modify the probing block.

3013 'Braking distance greater than parameter PROBERANGE'

DETECTION   During execution.

CAUSE   The distance needed to brake at the current speed is greater than parameter PROBERANGE.

SOLUTION   Increase the braking distance (PROBERANGE), or decrease the probing feedrate.

3015 'During block search, the home search is left out'

DETECTION   During execution.

CAUSE   An attempt has been made to search home while executing in dry run.

SOLUTION   Search home outside the program.

3016 'During the block search, the manual mode is left out'

DETECTION   During execution.

CAUSE   A manual mode request has been received during block search.

SOLUTION   Do not try to make a manual mode request during block search or bear in mind the warning that the manual mode is not active.

3017 'Block search already activated'

DETECTION   During block search.

CAUSE   An attempt has been made to activate block search, but it was already activated.

SOLUTION   Contact your supplier.

3018 'The constant surface speed has not been reached yet'

DETECTION   During execution.

CAUSE   The spindle has not reached the programmed speed.

SOLUTION   The solutions may be:
- Decrease the constant surface speed S.
- Decrease the axis feedrate.
- Program S before and give the spindle some time to catch up speed.

3019 'The feedrate programmed in G95 is too low'

DETECTION   During execution.

CAUSE   The resulting feedrate is too low.

SOLUTION   Increase the feedrate or increase the spindle rpm.

3020 'S has not been programmed in G96'

DETECTION   During execution.

CAUSE   The spindle speed is zero.

SOLUTION   Program a speed.

3021 'The maximum constant surface speed is zero'

DETECTION   During execution.

CAUSE   The maximum spindle speed is zero.

SOLUTION   Program a speed.

3022 'Error when initializing the coordinates of the axes'

DETECTION   During execution.

CAUSE   The coordinates of an axis do not match.

SOLUTION   Contact your Fagor supplier.

3023 'Axes missing in the new coordinate system'

DETECTION   During execution.

CAUSE   While defining the new coordinate system, one of the first three axes of the channel is not active.

SOLUTION   Activate the desired first three axes of the channel using the instruction #SET AX[ ].

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 3000-3999

Page 86 of 128

3025 'Positive software travel limit overrun'

DETECTION   During execution.

CAUSE   An attempt has been made to move an axis to coordinate that is beyond the software limits. The limits may have been set by machine parameter, via program or in real time.

SOLUTION   Keep the axes within the software limits.

3026 'Negative software travel limit overrun'

DETECTION   During execution.

CAUSE   An attempt has been made to move an axis to coordinate that is beyond the software limits. The limits may have been set by machine parameter, via program or in real time.

SOLUTION   Keep the axes within the software limits.

3027 'Error when calculating the reverse RTCP transformation'

DETECTION   During execution.

CAUSE   The transformation from part coordinates to machine coordinates cannot be executed.

SOLUTION   Contact your Fagor supplier.

3029 'Error when calculating the direct RTCP transformation'

DETECTION   During execution.

CAUSE   The transformation from machine coordinates to part coordinates cannot be executed.

SOLUTION   Contact your Fagor supplier.

3030 'Simulated axes cannot be mixed with non-simulated axes'

DETECTION   During execution.

CAUSE   In the process of measuring with a probe, simulated axes are being mixed with non-simulated ones.

SOLUTION   Use the same type of axes in the process of measuring with a probe.

3031 '(RTCP/TLC) compensation not allowed in the current status.

DETECTION   During execution.

CAUSE   An attempt has been made to activate one of the RTCP or TLC compensations while another compensation was active.

SOLUTION   Execute #RTCP OFF or TLC OFF before trying to activate the desired transformation.

3032 'Home search not possible in DRO mode'

DETECTION   During execution.

CAUSE   An attempt has been made to home a DRO axis.

SOLUTION   Do not home a DRO axis.

3033 'It is not possible to switch to manual mode if the axis is at the rotation center'

DETECTION   During execution.

CAUSE   An attempt has been made to activate the manual mode while the face C axis transformation is active and the axes are in the center of rotation.

SOLUTION   Activate the manual mode with the axes in a different position from that of the rotation center.

3034 'Wrong set number

DETECTION   During execution.

CAUSE   An attempt has been made to access a nonexistent set of axis parameters.

SOLUTION   The set number must be between 1 and NPARSET.

3035 'The difference between the following errors of slaved (coupled) axes is too large'

DETECTION   During execution.

CAUSE   The following error difference between the master and the slave is greater than MAXCOUPLE for Gantry axes or than the error programmed for coupled axes

SOLUTION   Adjust similarly the dynamic performance of the axes being coupled or increase the maximum difference allowed between the following errors of both axes.

3036   'Wrong gear change process'

DETECTION	During execution.
CAUSE	The spindle gear in the CNC's history and the one indicated by the PLC as being active are not the same.
SOLUTION	Check the PLC program.

3037   'Axis locked'

DETECTION	During execution.
CAUSE	The axis tries to move, but the SERVO is not ON.
SOLUTION	Check the treatment of the SERVO signal in the PLC program or increase the value of the DWELL parameter of the axis.

3038   'Too many parameters pending to be reported'

DETECTION	During execution.
CAUSE	The system is overloaded.
SOLUTION	Close the Windows applications not related to the CNC. Contact Fagor.

3039   'The stop block has not been found in the block search'

DETECTION	While executing in dry run.
CAUSE	When executing the program in dry run, it does not go through the stop block.
SOLUTION	Change the stop block.

3040   'Hirth axis positioned wrong'

DETECTION	During execution.
CAUSE	The Hirth axis is not positioned in a multiple number of its step.
SOLUTION	Position the Hirth axis properly in a valid position or cancel it as Hirth axis.

3041   'No spindle has been defined for G33/G95'

DETECTION	During execution.
CAUSE	Functions G33 and G95 need a spindle to work. This spindle is by default the master spindle of the channel, but if PLC register SYNC indicates another spindle, it works with the latter.
	If at a particular moment, none of the two is defined, an error is issued.
SOLUTION	Assign a spindle to the PLC register or have a master spindle defined in the channel.

3043   'Spindle in M5 when activating electronic threading'

DETECTION	During execution.
CAUSE	The spindle involved in electronic threading (G33) is in M5 by default, or an M5 has been programmed in the same block as the G33.
SOLUTION	Program a spindle rotation first or in the same block as the G33 indicating the turning direction (M3/M4) or the speed (S).

3500   'Linear acceleration of section 1 smaller than or equal to zero'

DETECTION	On startup or during execution.
CAUSE	A zero acceleration has been programmed.
SOLUTION	Program a positive value for the acceleration.

3501   'Linear acceleration of section 1 greater than the maximum'

DETECTION	During startup or execution.
CAUSE	An acceleration greater than the maximum has been programmed.
SOLUTION	Program a smaller value than the maximum for the acceleration.

3502   'Linear acceleration of section 2 smaller than or equal to zero'

DETECTION	On startup or during execution.
CAUSE	A zero acceleration has been programmed.
SOLUTION	Program a positive value for the acceleration.

3503   'Linear acceleration of section 2 greater than the maximum'

DETECTION	During startup or execution.
CAUSE	An acceleration greater than the maximum has been programmed.
SOLUTION	Program a smaller value than the maximum for the acceleration.

3504 'Acceleration changing speed greater than the maximum'

DETECTION	During startup or execution.
CAUSE	An acceleration changing speed greater than the maximum has been programmed.
SOLUTION	Program a smaller value than the maximum for the speed.

3505 'Jerk limit overshoot'

DETECTION	During execution.
CAUSE	Jerk overshoot on this path.
SOLUTION	Contact your Fagor supplier.

3506 'The Jerk limit will be exceeded'

DETECTION	During the execution of the command to analyze the frequencies.
CAUSE	The frequency is too high for the programmed amplitude.
SOLUTION	Decrease the maximum frequency or decrease the amplitude.

3507 'Acceleration limit exceeded'

DETECTION	During execution.
CAUSE	Too much acceleration on this path.
SOLUTION	Contact your Fagor supplier.

3508 'Frequency too high for the programmed speed'

DETECTION	During the execution of the command to analyze the frequencies.
CAUSE	The additive speed is lower than the one resulting for the maximum frequency.
SOLUTION	Decrease the maximum frequency or program a higher additive speed.

3600 'Zero spindle speed'

DETECTION	During execution.
CAUSE	No spindle positioning speed has been programmed.
SOLUTION	Check the programming and make sure that parameters REFFEED2 and G00FEED of the active spindle gear are not zero.

3601 'Programmed spindle speed greater than the maximum limit'

DETECTION	During execution.
CAUSE	A spindle speed has been programmed that is greater than the maximum set by machine parameter for the gear being used.
SOLUTION	Check spindle parameter G00FEED.

3602 'The spindle cannot be moved in DRO mode'

DETECTION	During execution.
CAUSE	Spindle in DRO mode via PLC. DROS = 1 and SERVOSON = 0.
SOLUTION	Check spindle marks DRO and SERVO in the PLC program.

3603 'The commanded spindle position exceeds the module range'

DETECTION	During execution.
CAUSE	An attempt has been made to position the spindle in a coordinate that is out of the range set by machine parameters MODUPLIM and MODLOWLIM.
SOLUTION	Check parameters MODUPLIM and MODLOWLIM

3604 'Spindle positioning direction contrary to the setting of the machine parameter'

DETECTION	During execution.
CAUSE	A positioning has been programmed that requires a spindle move in the opposite direction to what has been set by machine parameter UNIDIR.
SOLUTION	Check spindle machine parameter UNIDIR.

3605 'Spindle turning direction contrary to the setting of the machine parameter'

DETECTION	During execution.
CAUSE	A rotation has been programmed that requires a spindle move in the opposite direction to what has been set by machine parameter UNIDIR.
SOLUTION	Check spindle parameter UNIDIR .

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 3000-3999

Page 89 of 128

3606 'The spindle positioning requires an absolute coordinate'

DETECTION	During execution.
CAUSE	After a spindle turn in open loop, the positioning must always be in absolute coordinates.
SOLUTION	Program the positioning in absolute coordinates.

3700 'Axis travel limit overrun'

DETECTION	During the movement of the axis.
CAUSE	An attempt has been made to overrun the physical travel limits (PLC).
SOLUTION	Check the program.

3701 'The reference position is beyond the software limits'

DETECTION	During the validation of machine parameters.
CAUSE	The value of parameter REFVALUE exceeds the range of the axis software limits.
SOLUTION	Change the parameters involved.

3702 'Axis following error out of limit'

DETECTION	During execution.
CAUSE	The values allowed for the following error and set by machine parameter have been exceeded. Possible causes: Improper axis adjustment, some enable is missing, failures in the motor, drive, feedback system and/or mechanical.
SOLUTION	Check parameters, adjustment, connections, status of the drive, feedback system, motor,

3703 'Positive software travel limit overrun'

DETECTION	During the movement of the axis.
CAUSE	The coordinate set by machine parameter LIMITPOS or by variable RTLIMITPOS has been exceeded.
SOLUTION	Check the program.

3704 'Negative software travel limit overrun'

DETECTION	During the movement of the axis.
CAUSE	The coordinate set by machine parameter LIMITPOS or by variable RTLIMITPOS has been exceeded.
SOLUTION	Check the program.

3705 'Home search required'

DETECTION	When positioning the spindle if it has not been homed.
CAUSE	The spindle reference (home) got lost when switching from open loop to close loop.
SOLUTION	Contact your supplier.

3706 'Error when operating with a probe'

DETECTION	Error when initializing the probing process.
CAUSE	The digital input assigned to the probe is not valid.
SOLUTION	Check the probe parameters.

3707 'Error when searching home'

DETECTION	In the home search process for analog and sercos axes.
CAUSE	Reported by the CAN counter and the Sercos drive when the home search process fails. It may be due to wrong parameter settings or a faulty counter, sercos drive, feedback system, connections, etc.
SOLUTION	Check the parameters related with home search, status of the counter module or sercos drive, feedback system, connections, etc.

3708 'Maximum time to get into the in-position zone exceeded'

DETECTION	During the movement of an axis.
CAUSE	The time to reach position is greater than the value set by parameter INPOMAX.
SOLUTION	Adjust parameter INPOMAX.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 3000-3999

Page 90 of 128

3709 'Error when refreshing Analog Inputs'

DETECTION	When reading analog inputs.
CAUSE	Failure in the cyclic reading process for analog intputs. This could probably be due to problems in the COMPCI, CAN bus, analog input module, etc.
SOLUTION	Check the status of the CAN bus, analog input modules, connections, etc.

3710 'Position command values out of range'

DETECTION	During execution.
CAUSE	Overflow in the position command variable. The maximum value for SGN32 has been exceeded.
SOLUTION	Check the parameter settings of the axes (Gain, ServoOff, Resolution, etc.

3711 'Error when refreshing CAN feedback readings'

DETECTION	From the position loop, when reading the feedback of the analog axes.
CAUSE	The feedback reading of the analog axes was not completed due to an error at the CAN bus, failure at one of the feedback readings, feedback module or saturation at the CAN bus.
SOLUTION	Solve the possible error at the CAN bus or feedback module or increase the LOOPTIME if there is saturation at the bus.

3712 'Home search error with Gantry axes'

DETECTION	During execution. While searching home on Gantry axes.
CAUSE	It detected the home switch of the master axis before that of the slave.
SOLUTION	Adjust the position of the home switches of the Gantry axes. The first home switch to be pressed must be the slave one.

3800 'Continuous jog feedrate equal to zero'

DETECTION	During execution or when changing to continuous jog mode.
CAUSE	The feedrate for continuous jog is zero. No manual feedrate V.G.FMAN has been programmed and machine parameter JOGFEED is zero.
SOLUTION	Check axis machine parameter JOGFEED.

3801 'Distance or feedrate in incremental jog equal to zero'

DETECTION	During execution.
CAUSE	The feedrate or distance for incremental jog is zero. No manual feedrate V.G.FMAN has been programmed and machine parameter INCJOGFEED is zero or machine parameter INCJOGDIST is zero.
SOLUTION	Check axis machine parameters INCJOGFEED and INCJOGDIST.

3803 'The handwheel resolution cannot be zero'

DETECTION	While changing handwheel resolution.
CAUSE	The index selected from the jog panel or via PLC for parameter MPGRESOL[i] is zero.
SOLUTION	Check axis machine parameter MPGRESOL.

3804 'Handwheel index out of range (switch positions 1-3)'

DETECTION	While changing handwheel resolution.
CAUSE	An attempt has been made to set via PLC a handwheel position that is out of the permitted range.
SOLUTION	Check the writing of PLC.MPGIDX from the PLC.

3805 'Zero feedrate or distance in incremental jog'

DETECTION	While changing the distances or feedrates of incremental jog.
CAUSE	The index of incremental jog has been changed using a PLC program or the jog panel. The index selected for parameter INCJOGDIST or INCJOGFEED is zero.
SOLUTION	Check axis machine parameters INCJOGDIST and INCJOGFEED.

3806 'Incremental jog feedrate greater than the maximum value'

DETECTION	During execution.
CAUSE	The feedrate set by machine parameter for incremental jog is greater than the maximum allowed.
SOLUTION	Check machine parameters MAXMANFEED, INCJOGFEED and G00FEED.

3807 'Incremental jog index out of range (switch positions 1-5)'

DETECTION	During execution.
CAUSE	An attempt has been made to set via PLC an incremental jog position that is out of the permitted range.
SOLUTION	Check the writing of PLC.INCJOGIDX from the PLC.

3808 'The axis does not exist or is not available'

DETECTION	During execution.
CAUSE	The possible causes are: 1. The axis was not in jog mode when quitting that mode. 2. A G101 has been programmed on a slave axis of a Gantry pair. 3. A G102 has been programmed on a slave axis of a Gantry pair.
SOLUTION	The first case is solved with a Reset.

3809 'Zero spindle speed programmed in G95'

DETECTION	During execution.
CAUSE	An attempt has been made to move an axis in continuous or incremental jog while G95 is active and the speed programmed for the spindle to be synchronized is zero.
SOLUTION	Program a speed for the spindle involved in the synchronization, either selected by the PLC in the SYNC register or the master spindle.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 3000-3999

Page 92 of 128

ERRORS 4000-4999

4000   **'Error when initializing the Sercos ring'**

CAUSE   Failed Sercos ring initializing process due to fiber optics connection problems, wrong CNC and drive parameter settings, problems on Sercos or drive boards, etc.

Class

0	Error when initializing the chip. The Sercos board is not detected or overflow at the DPRAM of the SERCON due to too many axes and data of the cyclic channel to be trasmitted.
1	Error when initializing the ring up to phase 4.
2	Error when resetting the errors.

Transition

It indicates the point in the initializing sequence where the error occurs:

0	Timeout.
1	Error when changing to Phase 0 (Optical Fiber)
2	Error when changing to Phase 1. (A drive does not respond: Hardware failure or Wrong ID in selector).
3	Error when changing to Phase 2.
4	Error when reading the OEM version. (Wrong drive ID at the CNC).
5	Error when reading T1min.
6	Error when reading Tatmt.
7	Error when reading T4min.
8	Error when reading Tmtsy.
9	Error when reading Tmtsc.
10	Error when reading SlaveNr.
11	Error when reading Tatat.
12	Error when calculating times.
13	Error when writing the OEM Password.
14	Error when writing T1.
15	Error when writing T4.
16	Error when writing T3.
17	Error when writing T2.
18	Error when writing Tncyc.
19	Error when writing Tscyc.
20	Error when writing MDTlen.
21	Error when writing TelegramType.
22	Error when writing MDT List.
23	Error when writing AT List.
24	Error when writing MDT Offset.
25	Error when writing RealTime Bit 2.
26	Error when writing OpMode.
27	Reset command error.
28	Park command error.
29	Phase 3 command error.

30	Error when changing to Phase 3.
31	Phase 4 command error.
32	Error when changing to Phase 4.
33	Error when reading Class Diagnostics 1.
34	Default error.
35	Error when reading Tncyc.
36	Error when reading OpMode.
37	Error when reading AxisType.
38	Error when reading G00Feed.
39	Error when reading Monit Window.
40	Error when reading SP100.
41	Error when reading KV.
42	Error when reading Checksum.
43	Error when reading DV33.
44	Error when writing DV33.

Wrong value 1

PDU code received with an error (must be 5).

Wrong value 2

Sercos drive error code:

0	SERC_NO_ERROR
0x8001	ERROR_HSTIMEOUT
0x8002	ERROR_SCHSTIMEOUT
0x8004	SERC_ERROR_TIMEOUT
0x4000	ERROR_CALCULATE_T1
0x4001	ERROR_CALCULATE_T2
0x4002	ERROR_CALCULATE_T3
0x4004	ERROR_CALCULATE_T4
0x4008	ERROR_CALCULATE_TEND
0x2001	ERROR_WRONGPHASE
0x2002	ERROR_WRONGADDRESS
0x2004	ERROR_WRONGATNUMBER
0x2008	ERROR_SCTRANSNOTREADY
0x1002	ERROR_DPRAMOVERFLOW
0x1004	ERROR_SCNOTINIT
0x1008	ERROR_WRONGCHANNELNUMBER
0x0801	ERROR_SCTRANSNODATA
0x0802	ERROR_SCNODATA
0x0803	NOT_READY_FOR_SCDATA
0x0400	ERROR_SCTRANS
0x0201	ERROR_ATMISS
0x0202	ERROR_NERR
0x0203	ERROR_MSTMISS
0x0204	ERROR_DISTORSION
0x0205	ERROR_FIBRA_ROTA
0x0101	NOT_READY
0x0102	BUSYTIMEOUT
0x0080	ERROR_DEFAULT
0x0040	READY_FOR_SCDATA

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 4000-4999

Page 94 of 128

Wrong value 2+3

Other Sercos drive error codes.

5+0x10	Request of Abort/Suspend/Resume of a command that is not active.
7+0x7	Request to a busy service channel.
7+0x20	Wrong logic axis number.
8+0xFE0	Request to a busy service channel.

SOLUTION Check the following:
- Analyze the error codes to try to determine the error source.
- Verify that the optical fiber is properly connected and in good condition.
- Check the parameter settings at the CNC and at the drives. Drive selector thumb-wheel at the drives (DriveID), parameters LOOPTIME, SERCPOWSEL, SERCBRATE, OPMODE, etc.

Contact our technical service department.

4001 'Parameter LOOPTIME different at the CNC and at the drive'

CAUSE Value of the parameter LOOPTIME different at the CNC and at the drive.

SOLUTION Set it to the same at the CNC and at the drive.

4002 'Parameter OPMODE different at the CNC and at the drive'

CAUSE Value of the parameter OPMODE different at the CNC and at the drive.

SOLUTION Set it to the same at the CNC and at the drive.

4003 'Parameter AXISMODE different at the CNC and at the drive'

CAUSE Value of the parameter AXISTYPE different at the CNC and at the drive.

SOLUTION Set it to the same at the CNC and at the drive.

4005 'Following error monitoring not active at the drive'

CAUSE 'Following error monitoring not active at the drive'

SOLUTION Activate parameter PP159 at the drive.

4006 'Drive parameter SP100 must be 0'

CAUSE The drive does not have an additional command enabled.

SOLUTION Set drive parameter SP100 to 0.

4007 'The Sercos spindle needs a gain other than 0'

CAUSE The spindle KV is 0.

SOLUTION Set the drive's KV to a value other than 0.

4008 'Sercos ring error because the drive has been reset'

CAUSE Warning that the drive has been reset.

SOLUTION Ignore voluntary resets of the drive with the reset button, WinDDS (recording the version, soft reset) or hardware problems at the drive.

Contact our technical service department.

4200 'Error in the Sercos cyclic channel'

CAUSE An error has occurred when reading or writing the cyclic channel.

SOLUTION Contact our technical service department.

4201 'Error in the Sercos service channel'

CAUSE An error has occurred in the Sercos service channel in the processes launched from the loop:
- Reading of variables: List of errors.
- Writing of variables: FF, ACF, set presetting, KV.
- Execution of commands: Park, SET change.

SOLUTION Analyze the error codes to try to determine the error source.

Contact our technical service department.

4202 'Drive Enable (DRENA) missing'

CAUSE While moving an axis, the DRENA signal (enable) of the PLC drops.

SOLUTION Analyze the PLC maneuver to determine what causes the DRENA signal to drop.

4203 **'Speed Enable (SPENA) missing'**

CAUSE   While moving an axis, the SPENA signal (enable) of the PLC drops.

SOLUTION   Analyze the PLC maneuver to determine what causes the SPENA signal to drop.

4204 **'Error when resetting the Sercos drive'**

CAUSE   An error occurred when executing the command to reset the errors of a drive (ID 99). This command will be executed from the loop while processing a CNC reset if the drive is reporting errors.

SOLUTION   Analyze the error codes to try to determine the error source.

              Contact our technical service department.

4205 **'SERCOS ring error'**

Value

Error codes that identify the source or sources of the problem:

0x00000002 Broken optic fiber.

0x00000100 AT loss.

0x00000200 MST loss.

0x00008000 AT transmission error. (Optic fiber or drive reset)

0xFFFF0000 Failed access to the common RAM of the SERCON.

CAUSE   An error occurred at the Sercos bus that causes a loss of Phase 4.

SOLUTION   Analyze the error codes to try to determine the error source.

              Contact our technical service department.

4206 **'SERCOS drive error'**

CAUSE   The drive reports an error.

SOLUTION   Analyze error codes. Refer to the drive manual.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 4000-4999

Page 96 of 128

ERRORS 5000-5999

5000   'PLC error: The timer does not exist'

DETECTION   PLC timer data reading processes.
CAUSE   Request to read a timer that does not exist.
SOLUTION   Valid timers: T1 - T256.

5001   'PLC error: The counter does not exist'

DETECTION   PLC counter data reading processes.
CAUSE   Request to read a counter that does not exist.
SOLUTION   Valid counters: C1 - C256.

5002   'CNCRD: unidentified variable'

DETECTION   When executing CNCRD instructions.
CAUSE   The variable does not exist.
SOLUTION   Check the syntax of the variable to be read.

5003   'CNCWR: unidentified variable'

DETECTION   When executing CNCWR instructions.
CAUSE   The variable does not exist.
SOLUTION   Check the syntax of the variable to be written.

5004   'Variable without reading permission for PLC'

DETECTION   When executing CNCRD instructions.
CAUSE   CNCRD of a variable that has no reading permission for the PLC.
SOLUTION   Refer to the manual to check the permissions of the variable.

5005   'Variable without writing permission for PLC'

DETECTION   When executing CNCRD instructions.
CAUSE   CNCRD of a variable that has no writing permission for the PLC.
SOLUTION   Refer to the manual to check the permissions of the variable.

5006   'Variable writing from the PLC out of range'

DETECTION   When executing CNCWR instructions.
CAUSE   The value to be written is out of range.
SOLUTION   Check the syntax of the variable to be written.

5007   'Syntax error when writing the variable from the PLC'

DETECTION   When executing CNCWR instructions.
CAUSE   The variable does not exist or it does not have writing permission.
SOLUTION   Check the syntax of the variable to be written.

5008   'The variable could not be written from the PLC'

DETECTION   Of GUP/LUP/LUPACT values from the CNCWR instruction .
CAUSE   A request has been received to write unavailable GUP/LUP/LUPACT values.
SOLUTION   Check the settings of global and local parameters as well as the PLC program.

5009 'Syntax error when reading the variable from the PLC'

DETECTION	When executing CNCRD instructions.
CAUSE	The variable does not exist or it does not have reading permission.
SOLUTION	Check the syntax of the variable to be written.

5010 'Division by zero at the PLC'

DETECTION	When executing DVS/MDS instructions from the PLC program.
CAUSE	If the denominator of a DVS/MDS operation is 0
SOLUTION	Check the PLC program to make sure that the denominator is not 0.

5013 'Error when reading the PLC digital inputs'

DETECTION	When reading the digital inputs of the PLC.
CAUSE	The digital I/O configuration table is wrong.
SOLUTION	Check the digital I/O configuration table.
	Contact our technical service department.

5014 'Error when writing the PLC digital outputs'

DETECTION	When writing the digital outputs of the PLC.
CAUSE	A request has been made to write the digital outputs without finishing the previous one or the digital I/O configuration table is not valid.
SOLUTION	Make the relevant checks to assure the integrity of the BUS CAN. Increase the PLCFREQ cycle times. Check the digital I/O configuration table.
	Contact our technical service department.

5015 'The variable could not be read from the PLC'

DETECTION	Of GUP/LUP/LUPACT values from the CNCRD instruction .
CAUSE	A request has been received to read unavailable GUP/LUP/LUPACT values.
SOLUTION	Check the settings of global and local parameters as well as the PLC program.

5016 'Reading value out of range'

DETECTION	From the CNCRD instruction.
CAUSE	When the data that has been read is out of range.
SOLUTION	Check the syntax of the instruction.

5017 'Null CNCEX block'

DETECTION	From the CNCEX instruction.
CAUSE	The block is null or the channel it is addressed to is not available.
SOLUTION	Check the block to be executed and the status of the channel.

5018 ''CNCEX has not been executed because the communication mark is set to "1"'

DETECTION	When executing a CNCEX instruction.
CAUSE	The control mark of the CNCEX block is set to 1 when beginning to execute a new CNCEX block. It may be due to incorrect programming or because the channel is busy with a previous CNCEX.
SOLUTION	Check, in the PLC program, the logic of the CNCEX's and of the control marks.

5020 'CNCEX: execution incomplete'

DETECTION	When executing a CNCEX instruction.
CAUSE	The block cannot be executed in the requested channel.
SOLUTION	Check the status of the channel where the CNCEX is to be executed.

5021 '#'CNCEX: the indicated channel is not a PLC channel'

DETECTION	When executing a CNCEX instruction.
CAUSE	The channel the CNCEX block is addressed to is not a PLC channel.
SOLUTION	Check the machine parameters for the type of channel and the PLC program.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 5000-5999

Page 98 of 128

ERRORS 6000-6999

6000 'Feedback alarm'

DETECTION   During execution.

CAUSE   Feedback error on analog axes. It is enabled by means of machine parameter FBACKAL.

DIFFERENTIAL TTL:
- One or more cables broken (A, B signals or their inverted signals).
- Feedback input disconnected at the counter.

DIFFERENTIAL SINUSOIDAL:
- One or more cables broken (A, B signals or their inverted signals).
- Feedback input disconnected at the counter.
- Input signal amplitude above 1.45 Vpp (approx.)
- Input signal amplitude under 0.4 Vpp (approx.)
- Too much phase shift between A and B signals (in quadrature, in theory)

For non-differential signals, the feedback alarm (FBACKAL) must be disabled because it cannot be determined for sure.

SOLUTION   Contact our technical service department.

6001 'Tendency test alarm activated'

DETECTION   When an axis runs away and the tendency test monitoring is activated.

CAUSE   Axis runaway. Positive feedback on the axis for a longer period of time than what is set by parameter ESTDELAY.

SOLUTION   Check the sign of the velocity command and of the feedback. Set parameters ESTDELAY, command sign AXISCHG and feedback LOOPCHG.

6002 'The PLC program is not running'

DETECTION   Continuous monitoring of PLC status.

CAUSE   The PLC program is not running.
- Installation of a new software version.
- The PLC has been stopped and has not been resumed.

SOLUTION   Set the PLC program running. Compile the PLC program if necessary.

6003 'External emergency activated'

DETECTION   Continuous monitoring from each interpolation cycle.

CAUSE   The PLC mark _EMERGEN has been set to zero.

SOLUTION   Check the logic of the _EMERGEN signal in the PLC program.

6004 'Error when initializing the BUS CAN'

CAUSE   The drive reports an error.

SOLUTION   Analyze error codes. Refer to the drive manual.

6005 'Error when selecting the work frequency of the BUS CAN'

DETECTION   During system startup.

CAUSE   One or several nodes are not tuned in to the frequency set by parameter.

SOLUTION   Check the following:
- The CAN cable distance matches the frequency parameter.
- See if all the modules are recognized in diagnosis mode.

Contact our technical service department.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 6000-6999

6006   'One or several CAN nodes do not respond'

CAUSE   One or several remote nodes do not respond due to a reset, shortcircuit, poor performance, etc.

SOLUTION   See if all the modules are recognized in diagnosis mode.

Contact our technical service department.

6007   'Watchdog in the COMPCI'

CAUSE   The COMPCI's do not respond due to poor performance, contact, etc.

SOLUTION   Contact our technical service department.

6008   'Watchdog in the PC + COMPCI'

CAUSE   The COMPCI and the PC do not respond due to poor performance, contact, etc.

SOLUTION   Contact our technical service department.

6009   'Watchdog at the PC'

CAUSE   The PC does not respond.

SOLUTION   Contact our technical service department.

6010   'Error in COMPCI processes'

CAUSE   Various errors in the processes managed by the COMPCI.

SOLUTION   Contact our technical service department.

6011   'CAN error in remote node'

CAUSE   Error reported by a node.

Error code

1   CAN controller overrun. Possible loss of received messaged.

2   BusOFF. The node detected that the BUS dropped.

3   Reception fifo overrun. Possible loss of received messaged.

4   Warning. The error counter exceeds level 1 (96).

11   The node informs that the COMPCI does not respond to the presence controls.

12   Node reset (power supply problems, shortcircuit, watchdog, etc.)

13   Message transmission failure.

Type of module

1   Analog outputs.

2   Counter.

3   Digital outputs.

4   Digital inputs.

5   Analog inputs.

7   PT100 inputs.

8   CAN keyboard handwheel.

9   Probes.

10   Jog keyboard.

11   Alphanumeric keyboard.

Node logic number

SOLUTION   Contact our technical service department.

6012   'CAN controller error'.

CAUSE   The COMPCI detects BUSOFF. The Bus dropped.

SOLUTION   Verify in diagnosis mode that all the modules and the CAN error counter are recognized. In case of error, make the relevant checks to assure the integrity of the BUS CAN.

Contact our technical service department.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 6000-6999

Page 100 of 128

6013   'Timeout when initializing the CAN'

DETECTION   During startup.

CAUSE   Failed initialization of the BUS CAN due to BUS problems. Off/On sequence too fast.

SOLUTION   Make the following checks and take the following actions to assure the integrity of the BUS CAN:

- There are no CAN groups with the same address selected with the thumb-wheel.
- The CAN thumb-wheel of the COMPCI must be 0.
- Line terminators.
- Grounds.
- CAN cable continuity.
- CAN cable connectors (even unplug them and plug them back in).
- Ribbon (flat) cable connection between the nodes and the power supply.
- Power supplies of the CAN groups (levels, possible resets, etc.).
- See if all the modules are recognized in diagnosis mode.

Contact our technical service department.

6014   'Failed access to DPRAM of the COMPCI'

DETECTION   During startup.

CAUSE   Failed initialization of the BUS CAN due to BUS problems. Off/On sequence too fast.

SOLUTION   Make the following checks and take the following actions to assure the integrity of the BUS CAN:

- There are no CAN groups with the same address selected with the thumb-wheel.
- The CAN thumb-wheel of the COMPCI must be 0.
- Line terminators.
- Grounds.
- CAN cable continuity.
- CAN cable connectors (even unplug them and plug them back in).
- Ribbon (flat) cable connection between the nodes and the power supply.
- Power supplies of the CAN groups (levels, possible resets, etc.).
- See if all the modules are recognized in diagnosis mode.

Contact our technical service department.

6015   'CAN error counter exceeded level 1'

CAUSE   The (Rx/Tx) error counter exceeds level 1 (96).

SOLUTION   Make the relevant checks to assure the integrity of the BUS CAN.

Contact our technical service department.

6016   'CAN error counter at a critical level'

CAUSE   The (Rx/Tx) error counter exceeds the critical level (127).

SOLUTION   Make the relevant checks to assure the integrity of the BUS CAN.

Contact our technical service department.

6017   'CAN controller's FIFO overrun'

CAUSE   Overflow at the CAN controller's receiving FIFO. Possible loss of received messaged.

SOLUTION   Contact our technical service department.

6018   'COMPCI's FIFO CAN overrun'

CAUSE   Overflow at the COMPCI's receiving FIFO. Possible loss of received messaged.

SOLUTION   Contact our technical service department.

6019   'CAN reading timeout'.

CAUSE   Failed reading of digital and analog inputs, counters and keyboard handwheels. A node did not send the message in time.

SOLUTION   Verify in diagnosis mode that all the modules and the CAN error counter are recognized. In case of error, make the relevant checks to assure the integrity of the BUS CAN. If necessary, increase the cycle times (LOOPTIME, PLCFREQ).

Contact our technical service department.

6020 'CAN cycle overlap'

CAUSE   Failed reading of digital and analog inputs, counters and keyboard handwheels. A node did not send the message in time.

SOLUTION   Verify in diagnosis mode that all the modules and the CAN error counter are recognized. In case of error, make the relevant checks to assure the integrity of the BUS CAN. If necessary, increase the cycle times (LOOPTIME, PLCFREQ).

Contact our technical service department.

6021 'Problems in CAN transmission'

CAUSE   Failed transmission of digital and analog outputs, etc. Possible BUS collapse.

SOLUTION   Verify in diagnosis mode that all the modules and the CAN error counter are recognized. In case of error, make the relevant checks to assure the integrity of the BUS CAN. If necessary, increase the cycle times (LOOPTIME, PLCFREQ).

Contact our technical service department.

6022 'Error when initializing the Axis counter'

DETECTION   During startup.

CAUSE   The counter does not exist.

SOLUTION   Check the parameters.

Contact our technical service department.

6023 'Error when initializing the Handwheel counter'

DETECTION   During startup.

CAUSE   The counter does not exist.

SOLUTION   Check the parameters.

Contact our technical service department.

6024 'Error when initializing the Keyboard Handwheel'

DETECTION   During startup.

CAUSE   The handwheel input does not exist.

SOLUTION   Check the parameters.

Contact our technical service department.

6025 'Checksum error at the COMPCI'

CAUSE   Error in the checksum control of the data of the digital or analog outputs sent from the PC to the COMPCI through the Common RAM.

Type of module

1   Analog outputs.

3   Digital outputs.

SOLUTION   Contact our technical service department.

6026 'Hardware error at the COMPCI'

DETECTION   During system startup.

CAUSE   Error during the hardware test of the COMPCI.

SOLUTION   The diagnoses mode displays detailed information on this type of error. Verify that the software version of the COMPCI is the right one.

Contact our technical service department.

6027 'Hardware error in remote CAN node'

DETECTION   During system startup.

CAUSE   Error during the hardware test of the remote nodes.

SOLUTION   The diagnoses mode displays detailed information on this type of error. Verify that the software version of the node is the right one.

Contact our technical service department.

6028 'Error when accessing the RAM of the SERCON'

DETECTION   During startup.

CAUSE   RAM test failure.

SOLUTION   Contact our technical service department.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 6000-6999

Page 102 of 128

6029 **'Position increment limit exceeded at the CNC'**

    CAUSE   Excessive position increment of an analog axis. Hardware failure at the counter node,
                                Accesses to the COMPCI (connections), etc.

    SOLUTION   Contact our technical service department.

6030 **'Position increment limit exceeded at the CAN counter node'**

    CAUSE   Excessive position increment of an analog axis. Hardware failure of the counter node.

    SOLUTION   Contact our technical service department.

6031 **'Incompatible software version at COMPCI/Remote nodes'**

    DETECTION   During startup.

    CAUSE   Incompatible software version at CNC and COMPCI or remote modules.

    SOLUTION   Update the software of the COMPCI and remote modules. (itfcboot.exe).

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 6000-6999

Page 103 of 128

Errors 6000-6999

ERRORS 7000-7999

7001 'Two consecutive T's in cyclic magazine (M6 required)'

DETECTION	During execution.
CAUSE	Cyclic magazine and two consecutive T's have been programmed.
SOLUTION	Program an M6 after each T.

7002 'The tool is not in the magazine and ground tools are not allowed'

DETECTION	During execution.
CAUSE	A tool has been programmed that is in the tool table, but it is not in the magazine table. In this case the tool is assumed to be a ground tool, but the machine parameter indicates that ground tools are not allowed.
SOLUTION	Change the machine parameter to accept ground tools.

7003 'T not defined in the table'

DETECTION	During execution.
CAUSE	A tool has been programmed that is not defined in the tool table.
SOLUTION	Define a tool.

7004 'Tool rejected or worn out (expired)'

DETECTION	During execution.
CAUSE	A ground tool has been programmed but it is either expired (worn out) or invalid.
SOLUTION	Program an equivalent tool that is neither expired nor rejected.

7005 'Tool rejected or expired and without replacement'

DETECTION	During execution.
CAUSE	A tool has been programmed that is either worn out (expired) or invalid. When searching for a tool of the same family, there is none available.
SOLUTION	Enable a tool of the same family so it can be used.

7006 'D not allowed for this tool'

DETECTION	During execution.
CAUSE	An invalid edge has been programmed. The tool has less edges than the one programmed.
SOLUTION	Define the necessary edges on the tool.

7007 'In load mode: T0 not admitted'

DETECTION	During execution.
CAUSE	T0 cannot be programmed when the magazine is in load mode.
SOLUTION	Program the T to be loaded.

7008 'In load mode: this tool is already loaded'

DETECTION	During execution.
CAUSE	While the tool is load mode, a tool has been programmed that is already in the magazine.
SOLUTION	Program a tool that is not loaded yet.

7009   'In load mode: M6 without T'

DETECTION	During execution.
CAUSE	In load mode, an M6 has been programmed without the corresponding T.
SOLUTION	First, program a T with the tool to be loaded.

7010   'In load mode: T with two M6'

DETECTION	During execution.
CAUSE	In load mode, a second M6 has been detected for the same T.
SOLUTION	Program the T's and the M6's in sequence.

7011   'In load mode: D alone not allowed'

DETECTION	During execution.
CAUSE	A "D" cannot be programmed alone in load mode.
SOLUTION	It cannot be programmed like this.

7012   'In load mode: The tool is either worn out (expired) or rejected'

DETECTION	During execution.
CAUSE	In load mode, the tool to be loaded is either worn out (expired) or rejected.
SOLUTION	Program another tool that is available.

7013   'In load mode: Wrong position or there is no room in the magazine'

DETECTION	During execution.
CAUSE	In load mode, it has been detected that either the magazine is full or the indicated position is occupied.
SOLUTION	If there is no room in the magazine, first unload a tool. If the indicated position is occupied, choose another one.

7014   'In unload mode: T is not in the magazine'

DETECTION	During execution.
CAUSE	While the magazine is in unload mode, a tool has been programmed that is not in the magazine.
SOLUTION	A tool that is not loaded cannot be unloaded.

7015   'In unload mode: D not allowed with T'

DETECTION	During execution.
CAUSE	The D cannot be programmed in unload mode.
SOLUTION	Remove the D from the programming instruction.

7016   'In unload mode: D alone not allowed'

DETECTION	During execution.
CAUSE	A "D" cannot be programmed alone in unload mode.
SOLUTION	It cannot be programmed like this.

7017   'In unload mode: M6 without T'

DETECTION	During execution.
CAUSE	In unload mode, an M6 has been programmed without the corresponding T.
SOLUTION	First, program a T with the tool to be unloaded.

7018   'In unload mode: T with two M6'

DETECTION	During execution.
CAUSE	In load mode, a second M6 has been detected for the same T.
SOLUTION	Program the T's and the M6's in sequence.

7019   'In Setting: T0 not admitted'

DETECTION	During execution.
CAUSE	In Setting mode, the tool T0 is not valid
SOLUTION	Program a tool other than 0.

7020   'In Setting: D required'

DETECTION   During execution.
CAUSE   In Setting mode, the edge of the tool must be specified.
SOLUTION   Program the edge.

7021   'In Setting: D alone not allowed'

DETECTION   During execution.
CAUSE   A "D" cannot be programmed alone in unload mode.
SOLUTION   It cannot be programmed like this.

7022   'In Setting: M6 not admitted'

DETECTION   During execution.
CAUSE   In Setting mode, the M6 is not programmed
SOLUTION   It cannot be programmed like this.

7023   'In load mode: M6 required'

DETECTION   During execution.
CAUSE   Two consecutive T's have been programmed in load mode.
SOLUTION   In this mode, an M6 must be programmed after the T.

7024   'In unload mode: T0 not admitted'

DETECTION   During execution.
CAUSE   T0 cannot be programmed when the magazine is in unload mode.
SOLUTION   Program the T to be unloaded.

7025   'In unload mode: M6 required'

DETECTION   During execution.
CAUSE   Two consecutive T's have been programmed in unload mode.
SOLUTION   In this mode, an M6 must be programmed after the T.

7026   'In unload mode: This tool is already loaded'

DETECTION   During execution.
CAUSE   In unload mode, a tool has been programmed that is not in the magazine.
SOLUTION   Program a T that is already in the magazine.

7027   M6 without T'

DETECTION   During execution.
CAUSE   An M6 has been detected without its corresponding T. There is machine parameter
                whose value indicates what to do in this situation. If = 0, no error is issued; if = 1 a
                warning is issued and if = 2 an error is issued.
SOLUTION   First program the T or change the parameter value.

7028   T0 not admitted'

DETECTION   During execution.
CAUSE   T0 not allowed in a turret type magazine. In this type of magazine, the tool changes
                are made by rotating the turret. Just program the T.
SOLUTION   Do not program T0.

7029   'The operation could not be carried out: Error at the tool manager'

DETECTION   During execution.
CAUSE   Emergency at the tool magazine. It may be because the PLC has activated an invalid
                maneuvering mark, because the PLC has activated SETTMEM or because there is
                an internal error in the logic.
SOLUTION   This error comes up due to an invalid maneuver when changing the tool. The solution
                in this case, is to correct the change maneuver in the PLC program. In the case of
                SETTMEM, check why the PLC has set the emergency signal.

7030   M6 without T'

DETECTION   During execution.

CAUSE   An M6 has been detected without its corresponding T. If parameter "M6 ALONE = NO", an error is issued.

SOLUTION   M6 cannot be programmed without the corresponding T.

7031   'T with two M6'

DETECTION   During execution.

CAUSE   Two consecutive M6 have been detected for the same T.

SOLUTION   Check the program.

7032   'Error in the PLC operation'

DETECTION   During execution.

CAUSE   Wrong sequence in the PLC program.

SOLUTION   Check the PLC program.

7033   'The PLC generated an emergency in the tool magazine'

DETECTION   When the PLC is running.

CAUSE   The PLC sets the emergency mark to the tool manager.

SOLUTION   Remove the emergency.

7034   'Operation error: No room could be found in the magazine'

DETECTION   During execution.

CAUSE   The PLC maneuver generates an error.

SOLUTION   Check the PLC program.

7035   'Error in the magazine table'

DETECTION   During execution.

CAUSE   Error at the magazine during a maneuver.

SOLUTION   Check the magazine status. Maybe there is no room in the magazine.

7036   'It is not possible to change the family of tools that are in the magazine or in the spindle'

DETECTION   While editing tables or during execution.

CAUSE   For safety reasons, it is not possible to change the family of a tool that is in the magazine.

SOLUTION   First unload the tool.

7037   'The spindle must be empty for the LOAD and UNLOAD modes'

DETECTION   During execution.

CAUSE   To change the magazine mode to load or unload mode, the spindle (and the arms in the magazines where the case might be).

SOLUTION   Empty the spindle and the arms.

7039   'A tool cannot be in the spindle and in the arm 2 at the same time'

DETECTION   During execution.

CAUSE   This special case can only occur when using asynchronous or synchronous magazines with a tool changer and requesting the same tool that is in arm 2

SOLUTION   This situation is atypical and it can probably occur only after an error. To correct it, program a T0 M6 to return the tool to the magazine and be able to resume normally.

7040   'Operation error: First empty arm 1'

DETECTION   During execution.

CAUSE   This special case can only occur when using asynchronous or synchronous magazines with a tool changer and requesting the same tool that is in arm 1

SOLUTION   This situation is atypical and it can probably occur only after an error. To correct it, program a T0 M6 to return the tool to the magazine and be able to resume normally.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 7000-7999

Page 108 of 128

7041 **'No tool is active'**

DETECTION   During execution.

CAUSE   An edge has been programmed, but there is no active tool in the spindle.

SOLUTION   Place a tool in the spindle.

7042 **'POS has been programmed and the magazine is not in LOAD mode'**

DETECTION   During execution.

CAUSE   A tool loading position has been programmed, but the tool manager is not in this mode.

SOLUTION   The loading position is only allowed in load mode.

7043 **'The requested tool is the active tool of another channel'**

DETECTION   During execution.

CAUSE   A tool has been programmed that is already the active tool of another channel.

SOLUTION   Unload the requested tool from the spindle of the other channel, by programming T0 M6 in that channel.

Errors 7000-7999

ERRORS 8000-8999

8200   'Lexer fault when recognizing integers'

DETECTION   During editing and execution in MDI.

CAUSE   An error has occurred when converting a string of characters into a numerical value. It usually occurs because some value programmed in the instruction or variable is wrong.

SOLUTION   Check in the programming manual the values admissible for the parameters of the instruction or indexes of the variable.

8201   'Lexer fault when recognizing floating values'

DETECTION   During editing and execution in MDI.

CAUSE   An error has occurred when converting a string of characters into a numerical value. It usually occurs because some value programmed in the instruction or variable is wrong.

SOLUTION   Check in the programming manual the values admissible for the parameters of the instruction or indexes of the variable.

8203   'Comment opening missing'

DETECTION   During editing.

CAUSE   The comment closing character has been detected without detecting the opening character first.

SOLUTION   Check that the comments have both the opening and closing characters "(" and ")".

8204   '$ or # missing'

DETECTION   During editing and execution in MDI.

CAUSE   The possible causes are:
- An instruction has been programmed without "#".
- A flow controlling instruction has been programmed without "$".

SOLUTION   All instructions must begin with the # character and all flow controlling instructions must begin with the "$" character.

8205   '$ missing'

DETECTION   During editing.

CAUSE   A flow controlling instruction has been programmed without the beginning character "$".

SOLUTION   Program "$" before the name of the control instruction.

8206   '# missing'

DETECTION   During editing.

CAUSE   An instruction has been programmed without the beginning character "#".

SOLUTION   Program "#" before the name of the instruction.

8207   'Name of the program or subroutine too long'

DETECTION   During editing.

CAUSE   The maximum number of characters allowed for the name of a program or subroutine has been exceeded.

SOLUTION   The maximum number of characters allowed is 14.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 8000-8999

Page 111 of 128

8209   'Wrong active axis'

DETECTION   During editing.
CAUSE   The axis name has been programmed with the wrong wild character.
SOLUTION   The names of the axis with wild character are @1 through @5.

8210   'Integer limits exceeded'

DETECTION   During editing.
CAUSE   The programmed integer value is too high.
SOLUTION   The maximum value for an integer is 4294967295.

8211   'The limits of the whole portion have been exceeded'

DETECTION   During editing.
CAUSE   The whole portion of the number has the wrong value.
SOLUTION   The range of valid values for the whole portion of a number is ±99999.

8212   'The limits of the fractional portion have been exceeded'

DETECTION   During editing.
CAUSE   The decimal portion of the number has the wrong value.
SOLUTION   The range of valid values for the decimal portion of a number is ±0.99999.

8213   'The format of the fractional portion have been exceeded'

DETECTION   During editing.
CAUSE   The maximum number of decimals allowed in a number has been exceeded.
SOLUTION   The maximum number of decimals allowed in a number is 5.

8214   'Unknown flow controlling instruction'

DETECTION   During editing.
CAUSE   The instruction programmed after the "$" is wrong.
SOLUTION   Check the syntax of the instruction.

8218   'Wrong character'

DETECTION   During editing.
CAUSE   An invalid character has been detected in the block.
SOLUTION   Check the syntax of the block.

8221   'Syntax Error'

DETECTION   During editing and execution in MDI.
CAUSE   Wrong syntax of the programmed instruction or variable.
SOLUTION   Check the syntax of the instruction or variable in the programming manual.

8222   'Nonexistent M function'

DETECTION   During editing.
CAUSE   The programmed M function does not exist.
SOLUTION   Check the existing M functions in the programming manual.

8223   'The M functions are mutually exclusive or identical'

DETECTION   During editing and execution in MDI.
CAUSE   The possible causes are:
    • The same M function has been programmed more than once in the same block.
    • Incompatible M functions have been programmed in the same block.
SOLUTION   The solutions are:
    • An M function can only be programmed once in a block.
    • Refer to the programming manual to check the incompatibility of the M functions.

8224   'M function out of range'

DETECTION   During editing.
CAUSE   The programmed M function does not exist.
SOLUTION   Check the existing M functions in the programming manual.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 8000-8999

Page 112 of 128

8225   'G function out of range'

DETECTION	During editing and execution in MDI.
CAUSE	The programmed G function does not exist.
SOLUTION	Check the existing M functions in the programming manual.

8226   'H function out of range'

DETECTION	During editing.
CAUSE	The programmed H function does not exist.
SOLUTION	Check the existing H functions in the programming manual.

8227   'Negative S allowed only with G63'

DETECTION	During editing.
CAUSE	A negative spindle speed has been programmed without having programmed function G63 in the block.
SOLUTION	The spindle speed must have a positive value. A negative value is only allowed when programming G63 in the same block.

8228   'Tool number out of range'

DETECTION	During editing.
CAUSE	A negative tool number has been programmed.
SOLUTION	The tool number must always be zero or positive.

8230   'F feedrate programmed twice'

DETECTION	During editing.
CAUSE	The F function has been programmed more than once in the same block.
SOLUTION	Program the F function only once in the block.

8232   '"A" programmed twice'

DETECTION	During editing.
CAUSE	The A axis has been programmed more than once in the block.
SOLUTION	Program the A axis only once in the block.

8233   '"B" programmed twice'

DETECTION	During editing.
CAUSE	The B axis has been programmed more than once in the block.
SOLUTION	Program the B axis only once in the block.

8234   '"C" programmed twice'

DETECTION	During editing.
CAUSE	The C axis has been programmed more than once in the block.
SOLUTION	Program the C axis only once in the block.

8235   '"U" programmed twice'

DETECTION	During editing.
CAUSE	The U axis has been programmed more than once in the block.
SOLUTION	Program the U axis only once in the block.

8236   '"V" programmed twice'

DETECTION	During editing.
CAUSE	The V axis has been programmed more than once in the block.
SOLUTION	Program the V axis only once in the block.

8237   '"W" programmed twice'

DETECTION	During editing.
CAUSE	The W axis has been programmed more than once in the block.
SOLUTION	Program the W axis only once in the block.

8238   '"X" programmed twice'

DETECTION	During editing.
CAUSE	The X axis has been programmed more than once in the block.
SOLUTION	Program the X axis only once in the block.

8239　'"Y" programmed twice'

DETECTION	During editing.
CAUSE	The Y axis has been programmed more than once in the block.
SOLUTION	Program the Y axis only once in the block.

8240　'"Z" programmed twice'

DETECTION	During editing.
CAUSE	The Z axis has been programmed more than once in the block.
SOLUTION	Program the Z axis only once in the block.

8241　'Parameter out of range'

DETECTION	During execution.
CAUSE	The programmed value for the parameter of the instruction is too high.
SOLUTION	Program a smaller value for the parameter of the instruction.

8242　'Bracket missing'

DETECTION	During editing and execution in MDI.
CAUSE	The possible causes are: • The parameters of the programmed instruction must be between brackets. • The variable requires programming an index between brackets.
SOLUTION	Check the syntax of the instruction or variable in the programming manual.

8243　'"I" programmed twice'

DETECTION	During editing.
CAUSE	The I axis has been programmed more than once in the block.
SOLUTION	Program the "I" value only once in the block.

8244　'"J" programmed twice'

DETECTION	During editing.
CAUSE	The J axis has been programmed more than once in the block.
SOLUTION	Program the "J" value only once in the block.

8245　'"K" programmed twice'

DETECTION	During editing.
CAUSE	The K axis has been programmed more than once in the block.
SOLUTION	Program the "K" value only once in the block.

8247　'Only R1 can be part of an expression'

DETECTION	During editing.
CAUSE	The radius value has not been programmed correctly.
SOLUTION	The radius can only be programmed with "R" or "R1".

8250　'Axis variable without writing permission'

DETECTION	During editing.
CAUSE	An attempt has been made to write an axis variable that does not have a writing permission.
SOLUTION	The variable can only be read.

8251　'Global variable without writing permission'

DETECTION	During editing.
CAUSE	An attempt has been made to write a global variable that does not have a writing permission.
SOLUTION	The variable can only be read.

8253　'Axes missing in G20'

DETECTION	During editing.
CAUSE	Not all the parameters required by function G20 have been programmed.
SOLUTION	Check the G20 programming syntax.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 8000-8999

Page 114 of 128

8254   'Too many axes in G20'

DETECTION   During editing.
CAUSE   Some of the parameters programmed for G20 are not allowed.
SOLUTION   Check the G20 programming syntax.

8256   'Nonexistent interpolator cycle instruction'

DETECTION   During editing and execution in MDI.
CAUSE   The programmed instruction does not exist.
SOLUTION   Refer to the programming manual.

8257   'There can be nothing after the name of the program or subroutine'

DETECTION   During editing.
CAUSE   The possible causes are:
               1. In the block defining the name of the main program or subroutine, other characters
                  have also been written.
               2. The name of the main program or subroutine has a blank space.
SOLUTION   The definition of the main program or subroutine can only be accompanied by a
               comment. The forbidden characters are:
               1. Main program: \ / : * ? " < > | and blank space.
               2. Local subroutine: / ? " < > |) and blank space.

8258   'Expression expected after #TIME'

DETECTION   During editing and execution in MDI.
CAUSE   The #TIME instruction has been programmed incorrectly.
SOLUTION   The dwell may be programmed in two ways with the instruction #TIME:
               • #TIME [<expression>]
               • #TIME [<expression>]

               The parameter <expression> may be a whole number (integer), an arithmetic
               parameter or a control variable.

8265   'Unknown or incomplete word'

DETECTION   During editing.
CAUSE   The possible causes are:
               1. The programmed function, instruction or expression is not valid.
               2. M function has been programmed wrong for a particular spindle.
               3. #TOOL AX instruction programmed wrong.
SOLUTION   Refer to the programming manual to check the syntax of the instruction, function or
               expression to be programmed.

8267   'Cycle variable without writing permission'

DETECTION   During editing.
CAUSE   An attempt has been made to write a cycle variable that does not have a writing
               permission.
SOLUTION   The variable can only be read.

8275   'Too many M functions in the same block'

DETECTION   During editing.
CAUSE   Too many M functions in the same block.
SOLUTION   The maximum number of M functions allowed in the same block is 7.

8276   'Too many H functions in the same block'

DETECTION   During editing.
CAUSE   Too many H functions in the same block.
SOLUTION   The maximum number of H functions allowed in the same block is 7.

8279 'Mirror image repeated'

DETECTION	During editing.
CAUSE	The possible causes are:

1. Function G11, G12 or G13 has been programmed more than one in the same block.
2. Functions G10 and G11, G12 or G13 have been programmed in the same block.
3. Functions G14 and G11, G12 or G13 have been programmed in the same block.

SOLUTION	The solutions are:

1. Functions G11, G12 and G13 may be combined in the same block, but each of them can only be programmed once.
2. Function G10 cannot be programmed in the same block as G11, G12 or G13.
3. Function G14 cannot be programmed in the same block as G11, G12 or G13.

8280 'Negative spindle positioning (orienting) speed'

DETECTION	During editing.
CAUSE	A negative spindle speed has been programmed in M19 (Sx.POS).
SOLUTION	The positioning (orienting) speed must be positive.

8281 'Spindle positioning (orienting) speed programmed twice'

DETECTION	During editing and execution in MDI.
CAUSE	The spindle positioning speed in M19 has been programmed more than once in the same block.
SOLUTION	Program the spindle positioning speed "S.POS" only once.

8282 'Cycle parameter repeated'

DETECTION	During editing.
CAUSE	The canned cycle parameter has been programmed more than once in the block.
SOLUTION	Each canned cycle parameter can only be programmed once in the block.

8283 'Wrong parameter in canned cycle'

DETECTION	During editing.
CAUSE	The parameter programmed for this canned cycle is wrong.
SOLUTION	Check the programming manual for the parameters required and allowed by each canned cycle.

8284 'PLC variable without writing permission'

DETECTION	During editing.
CAUSE	An attempt has been made to write a PLC variable that does not have a writing permission.
SOLUTION	The variable can only be read.

8285 'G20: negative sign only allowed on the longitudinal axis'

DETECTION	During editing.
CAUSE	An axis that is not longitudinal has been programmed with a negative sign in function G20.
SOLUTION	In G20, only the longitudinal axis can have a negative sign. The longitudinal axis is the one that contains the tool and is indicated in parameter 3 or 5.

8290 'Variable of the tool manager without writing permission'

DETECTION	During editing.
CAUSE	An attempt has been made to write a variable of the tool manager that does not have a writing permission.
SOLUTION	The variable can only be read.

8291 'Variable of the machine parameters without writing permission'

DETECTION	During editing.
CAUSE	An attempt has been made to write a variable of machine parameters that does not have a writing permission.
SOLUTION	The variable can only be read.

FAGOR

CNC8070
REF. 0402

ERROR SOLUTIONS

Errors 8000-8999

Page 116 of 128

8297 **'Parameter repeated'**

DETECTION During editing.

CAUSE The possible causes are:

1. The parameter CONTERROR has been programmed more than once in the #HSC instruction.
2. In the subroutine calling instruction #PCALL, #MCALL or G function with associated subroutine, some parameter has been written more than once.
3. Some parameter has been programmed more than once in the #PROBE or #POLY instruction.

SOLUTION Program each parameter only once in the block.

8299 **'Brackets missing around the list of parameters'**

DETECTION During editing and execution in MDI.

CAUSE The programmed instruction requires that its list of parameters be between parenthesis.

SOLUTION Check the syntax of the instruction in the programming manual.

8300 **'G170, G171 and G157 require at least one axis'**

DETECTION During editing.

CAUSE No axis has been programmed with function G170, G171 or G157.

SOLUTION Program the axis or axes to be affected by function G170, G171 or G157.

8302 **'The minimum X coordinate must be smaller than the maximum'**

DETECTION During editing.

CAUSE In the #DGWZ instruction, the programmed minimum X coordinate is equal to or greater than the maximum.

SOLUTION The minimum coordinate must be smaller than the maximum.

8303 **'The minimum Y coordinate must be smaller than the maximum'**

DETECTION During editing.

CAUSE In the #DGWZ instruction, the programmed minimum Y coordinate is equal to or greater than the maximum.

SOLUTION The minimum coordinate must be smaller than the maximum.

8304 **'The minimum Z coordinate must be smaller than the maximum'**

DETECTION During editing.

CAUSE In the #DGWZ instruction, the programmed minimum Z coordinate is equal to or greater than the maximum.

SOLUTION The minimum coordinate must be smaller than the maximum.

8306 **'G201 requires programming the special function #AXIS'**

DETECTION During editing.

CAUSE The #AXIS instruction has not been programmed in the same block as function G201.

SOLUTION Function G201 requires programming the #AXIS instruction in the same block. This instruction must indicate the axes affected by the G function.

8307 **'The third primary axis has been programmed wrong'**

DETECTION During editing.

CAUSE In the G20 instruction, parameter 5 has been programmed the same as parameter 1 or parameter 2.

SOLUTION Parameter 5 must be different from parameter 1 and parameter 2.

8308 **'Expression or K expected after G04'**

DETECTION During editing.

CAUSE Function G4 has been programmed wrong.

SOLUTION Function G4 can be programmed as:

1. G4 <Dwell time>
2. G4 <Dwell time>

8309 'Too many axes have been programmed'

DETECTION	During editing.
CAUSE	Too many axes have been programmed in the #POLY instruction.
SOLUTION	The maximum number of axes that can be programmed is 3.

8310 'Required parameter missing'

DETECTION	During editing.
CAUSE	The possible causes are:
	1. Some required parameter has not been programmed in the #POLY instruction.
	2. Some required parameter has not been programmed in the instruction #CS DEF, #CS ON, #CS NEW, #ACS DEF, #ACS ON or #ACS NEW.
	3. Some required parameter has not been programmed in the #PROBE instruction.
SOLUTION	Check the syntax of the instructions in the programming manual.

8311 'Wrong parameter value'

DETECTION	During editing.
CAUSE	The possible causes are:
	1. A wrong value has been programmed for some argument of the instruction #CS DEF, #CS ON, #CS NEW, #ACS DEF, #ACS ON or #ACS NEW.
	2. A wrong value has been programmed for CONTERROR in the #HSC instruction.
	3. A wrong value has been programmed for EP or for R in the #POLY instruction.
	4. A wrong value has been programmed for the index of an arithmetic parameter.
SOLUTION	Check the syntax of the instructions in the programming manual. The index of an arithmetic parameter must always be positive or zero.

8312 '#CS/#ACS: wrong mode'

DETECTION	During editing.
CAUSE	A wrong value has been programmed for MODE in the instruction #CS DEF, #CS ON, #CS NEW, #ACS DEF, #ACS ON or #ACS NEW.
SOLUTION	Check the syntax of the instructions in the programming manual.

8313 '#CS/#ACS: wrong coordinate system number'

DETECTION	During editing.
CAUSE	A wrong value has been programmed for the coordinate system number in the instruction #CS DEF, #CS ON, #CS NEW, #ACS DEF, #ACS ON or #ACS NEW.
SOLUTION	The system number must have a value between 1 and 5, both included.

8314 'G30/G73 programmed wrong'

DETECTION	During editing.
CAUSE	In function G30/G73 only one of parameters I or J has been programmed.
SOLUTION	Program either both parameters I and J or none of them.

8315 'Wrong probe cycle number'

DETECTION	During editing and execution in MDI.
CAUSE	The programmed probe cycle is wrong.
SOLUTION	The right probe cycles are PROBE1 through PROBE8.

8316 'Parameter not allowed'

DETECTION	During editing and execution in MDI.
CAUSE	The parameter programmed for the instruction is wrong.
SOLUTION	Refer to the programming manual to check the right parameters for each cycle and instruction.

8317 'Negative D function'

DETECTION	During editing.
CAUSE	A negative tool offset has been programmed.
SOLUTION	The tool offset must have a positive value.

FAGOR

CNC8070
REF. 0402

ERROR SOLUTIONS

Errors 8000-8999

Page 118 of 128

8318 **'Only one active axis 1 allowed per block'**

DETECTION	During editing.
CAUSE	The @1 axis has been programmed more than once in the block.
SOLUTION	Program @1 only once in the block.

8319 **'Only one active axis 2 allowed per block'**

DETECTION	During editing.
CAUSE	The @2 axis has been programmed more than once in the block.
SOLUTION	Program @2 only once in the block.

8320 **'Only one active axis 3 allowed per block'**

DETECTION	During editing.
CAUSE	The @3 axis has been programmed more than once in the block.
SOLUTION	Program @3 only once in the block.

8321 **'Only one active axis 4 allowed per block'**

DETECTION	During editing.
CAUSE	The @4 axis has been programmed more than once in the block.
SOLUTION	Program @4 only once in the block.

8322 **'Only one active axis 5 allowed per block'**

DETECTION	During editing.
CAUSE	The @5 axis has been programmed more than once in the block.
SOLUTION	Program @5 only once in the block.

8323 **'Comparison expected'**

DETECTION	During editing.
CAUSE	"=" has been programmed instead of "==".
SOLUTION	Program "==" instead of "=".

8327 **'Some parenthesis missing'**

DETECTION	During editing and execution in MDI.
CAUSE	The number of opening parenthesis does not match the number of closing parenthesis.
SOLUTION	Check that each opening parenthesis has its corresponding closing parenthesis.

8328 **'#SET AX: only zero position allowed'**

DETECTION	During editing.
CAUSE	An integer value other than zero has been programmed in the #SET AX instruction.
SOLUTION	The #SET AX instruction only allows the zero value to indicate that the position is not occupied by any axis.

8329 **'V expected'**

DETECTION	During editing.
CAUSE	The name of a variable has been programmed that does not begin with a "V.".
SOLUTION	The name of a variable both in the part-program and via MDI must begin with the prefix "V.".

8330 **'Axis name or number expected'**

DETECTION	During editing and execution in MDI.
CAUSE	The programmed instruction or variable requires an axis name or number.
SOLUTION	Check the syntax of the instruction or variable in the programming manual.

8331 **'Axis name or number not expected'**

DETECTION	During editing and execution in MDI.
CAUSE	The programmed instruction or variable does not admit an axis name or number.
SOLUTION	Check the syntax of the instruction or variable in the programming manual

8402 **'Wrong tool magazine number'**

DETECTION   During editing.

CAUSE   The magazine number for which the variable of the tool manager is requested is wrong.

SOLUTION   The magazine number must be between 1 and 4. If the magazine number is not indicated, it will assume the first one.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Errors 8000-8999

Page 120 of 128

TOOL AND TOOL MAGAZINE TABLE

'The tool cannot be loaded into position %1'

DETECTION	During the manual loading of a tool from this dialog box, from the magazine list or loading the magazine table.
CAUSE	The tool does not fit in that magazine position, it is already in the magazine or it is not defined in the tool table.
SOLUTION	Choose a big enough magazine position. Check that the tool is defined in the table.

'The %1 tool cannot be unloaded'

DETECTION	During the manual unloading of a tool from this dialog box, on the magazine list or loading the tool table or the tool magazine table.
CAUSE	The tool is not in the magazine.
SOLUTION	Select a tool that is in the magazine.

'The maneuver cannot be executed. Check that the spindle is empty'

DETECTION	When executing a ground tool loading or unloading block.
CAUSE	The spindle is not empty or there is a channel in error status.
SOLUTION	Unload the tool from the spindle. Check that the status of all the channels is "READY".

'Wrong table type selected'

DETECTION	When loading the tool table or magazine table.
CAUSE	The file being loaded either does not correspond to the table to be loaded or has been edited externally.
SOLUTION	Choose the file corresponding to the table to be loaded.

'The tool does not exist or is already loaded into the magazine'

DETECTION	When loading a tool into the magazine with or without maneuver.
CAUSE	The tool does not exist or is already in the magazine, in the spindle or in the tool changer arms (if any).
SOLUTION	Define a tool. If it is in the spindle or in the tool changer arms, load it into the magazine.

'Tool %1 cannot be eliminated'

DETECTION	When deleting a tool or when loading the tool table.
CAUSE	The tool cannot be found.
SOLUTION	The tool is missing, therefore, cannot be eliminated. If it occurs when loading the table, initialize the table with the softkey and load the table again.

'Tool %1 does not exist or is not loaded into the magazine'

DETECTION	When unloading a tool from the magazine, when forcing the tool position or when loading the magazine table.
CAUSE	The tool does not exist or is not loaded into the magazine.
SOLUTION	Define the tool and load it in the magazine.

'Error when renaming the tool'

DETECTION	When trying to change the tool name.
CAUSE	It cannot create the tool in the database (bd8070.mdb). The data base may be being used by another application or it may not have write permission.
SOLUTION	The data base must have write permission. If the tool data base is being used by another application, wait for it to be released (freed).

'A tool that is in the spindle, in the magazine or in the changer arms cannot be renamed'

DETECTION	When trying to change the tool name.
CAUSE	An attempt has been made to change the name of a tool but there is already another tool with that name in the spindle, magazine or changer arms (if any).
SOLUTION	Choose another number or unload a tool to ground.

'A tool cannot be placed in the spindle if there is one in changer arm 2'

DETECTION	When forcing a tool as spindle tool.
CAUSE	There is a tool in changer arm 2.
SOLUTION	Remove the tool from changer arm 2.

'The tool of the changer arm cannot be placed in the spindle'

DETECTION	When forcing a tool as spindle tool.
CAUSE	The tool is in changer arm 1.
SOLUTION	Remove the tool from changer arm 1.

'The position does not exist'

DETECTION	When loading or unloading a tool from the magazine with or without maneuver.
CAUSE	The magazine position does not exist.
SOLUTION	Choose an existing magazine position or create it (machine parameter).

'Path %1 not found'

DETECTION	When loading, saving or printing the tool table or magazine table.
CAUSE	The selected path does not exist.
SOLUTION	Define the path properly or create it.

'File %1 not found'

DETECTION	When loading the tool table or magazine table.
CAUSE	The data file does not exist.
SOLUTION	Choose existing files and indicate its path correctly.

'Error when saving the data in %1'

DETECTION	When saving the tool table or magazine table.
CAUSE	The data file could not be created, already exists and has no write permission, is being used by another application or there is no room on the disk.
SOLUTION	Choose another directory to save the tables, give them write permission, close the application that is using it or free disk memory space.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Tool and tool
magazine table

Page 122 of 128

'Error opening file %1'

DETECTION   When loading, saving or printing the tool table or magazine table.

CAUSE   The possible causes are:

1. When loading a table. The data file does not exist, it does not have read permission or is being used by another application.

2. When saving a table. The data file could not be created, already exists and has no write permission, is being used by another application or there is no room on the disk.

3. When printing a table. When printing to a file, It could not be created, already exists and has no write permission, is being used by another application or there is no room on the disk. When printing to a printer, the printer either does not exist or is configured wrong.

SOLUTION   The possible solutions are:

1. When loading a table. Choose existing files, indicate their path correctly, give them read permission or close the application that is using them.

2. When saving a table. Choose another directory to save the tables, give them write permission, close the application that is using it or free disk memory space.

3. When printing a table. When printing to a file, choose another directory to save the tables, give them write permission, close the application that is using it or free disk memory space. When printing to a printer, choose an existing printer that is configured correctly.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Tool and tool magazine table

Page 123 of 128

Tool and tool
magazine table

MESSAGES OF THE PROFILE EDITOR

'Unresolved profile'

CAUSE   The profile to be saved is not resolved completely either when "FINISHING" or when "SAVING AND CONTINUING"

SOLUTION Resolve the profile to be saved. Only resolved profiled may be saved.

'Insufficient memory'

CAUSE   There isn't enough system memory to go on with the profile editor.

'Geometry error'

CAUSE   Wrong geometry when accessing the profile editor with a selected profile. The selected profile has some erroneous data in the path definition.

SOLUTION Correct the profile. All the paths that defined the profile must be properly defined.

'Error in element data'

CAUSE   The data entered in an element are not correct.

SOLUTION Correct the data of the element.

'Error in profile data'

CAUSE   Wrong data when editing a "circular" or "rectangular" profile.

SOLUTION Correct the data of the element.

'The arc does not go through its starting coordinate'

CAUSE   In an arc element, the data for the center, radius and starting point are not coherent.

SOLUTION Correct the data of the element.

'The arc does not go through its final coordinate'

CAUSE   In an arc element, the data for the center, radius and final point are not coherent.

SOLUTION Correct the data of the element.

'There is no arc that complies with all the data'

CAUSE   No arc can be found that is coherent with the known data.

SOLUTION Correct the data of the element.

'Element not tangent to the previous one'

CAUSE   The data entered in an element tangent to the previous element is not coherent.

SOLUTION Correct the data of the element.

'Wrong value'

CAUSE   The value entered to modify a corner (rounding, chamfer, tangential entry or tangential exit) is wrong.

SOLUTION Correct the data of the element. The value of the corner must be lower than the paths where it has been defined.

'Error in the axis of the plane'

CAUSE   Some axis of the plane is wrong. The same axis has been placed in the "CONFIGURATION" of the plane or some axis of the selected profile is not defined at the CNC.

SOLUTION   The plane must be formed by two different axes. Both axes must be present at the CNC.

FAGOR

CNC8070

REF. 0402

ERROR SOLUTIONS

Messages of the
profile editor

Page 126 of 128