CNCelite

8058/8060
8065/8070

Error solving manual.

Ref. 2209

FAGOR ⋑

AUTOMATION

TRANSLATION OF THE ORIGINAL MANUAL

This manual is a translation of the original manual. This manual, as well as the documents derived from it, have been drafted in Spanish. In the event of any contradictions between the document in Spanish and its translations, the wording in the Spanish version shall prevail. The original manual will be labeled with the text "ORIGINAL MANUAL".

MACHINE SAFETY

It is up to the machine manufacturer to make sure that the safety of the machine is enabled in order to prevent personal injury and damage to the CNC or to the products connected to it. On start-up and while validating CNC parameters, it checks the status of the following safety elements. If any of them is disabled, the CNC shows the following warning message.

- Feedback alarm for analog axes.
- Software limits for analog and sercos linear axes.
- Following error monitoring for analog and sercos axes (except the spindle) both at the CNC and at the drives.
- Tendency test on analog axes.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any of the safety elements being disabled.

HARDWARE EXPANSIONS

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any hardware manipulation by personnel unauthorized by Fagor Automation.

If the CNC hardware is modified by personnel unauthorized by Fagor Automation, it will no longer be under warranty.

COMPUTER VIRUSES

FAGOR AUTOMATION guarantees that the software installed contains no computer viruses. It is up to the user to keep the unit virus free in order to guarantee its proper operation. Computer viruses at the CNC may cause it to malfunction.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC due a computer virus in the system.

If a computer virus is found in the system, the unit will no longer be under warranty.

DUAL-USE PRODUCTS

Products manufactured by FAGOR AUTOMATION since April 1st 2014 will include "-MDU" in their identification if they are included on the list of dual-use products according to regulation UE 428/2009 and require an export license depending on destination.

It is possible that CNC can execute more functions than those described in its associated documentation; however, Fagor Automation does not guarantee the validity of those applications. Therefore, except under the express permission from Fagor Automation, any CNC application that is not described in the documentation must be considered as "impossible". In any case, Fagor Automation shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC if it is used in any way other than as explained in the related documentation.

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, hence no absolute match is guaranteed. However, the contents of this document are regularly checked and updated implementing the necessary corrections in a later edition. We appreciate your suggestions for improvement.

The examples described in this manual are for learning purposes. Before using them in industrial applications, they must be properly adapted making sure that the safety regulations are fully met.

INDEX

BLANK PAGE

0000-0999

0001   **'SYSTEM ERROR'**

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

0002   **'SYSTEM WARNING'**

DETECTION	During execution.
CAUSE	Warning of internal situations that could become system errors.
SOLUTION	Usually the CNC is restored by closing the warning. If the error persists, contact your supplier.

0003   **'Error when requesting memory. Restart Windows & CNC'**

DETECTION	During CNC startup.
CAUSE	The CNC does not have enough memory or the memory is too fragmented.
SOLUTION	Restart the unit and re-initiate the CNC. If the memory is too fragmented, when starting the unit up and re-initiate the CNC, the error will disappear. If the error persists after powering up several times, contact your supplier.

0004   **'Checksum error in PLC data'**

DETECTION	During CNC startup.
CAUSE	The PLC data related to marks, counters, timers and registers saved into a disk is not valid. The plcdata.bin file that contains these data does not exist, is not accessible or is corrupt.
CONSEQUENCE	The PLC data related to marks, counters, timers and registers is 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	During CNC startup.
CAUSE	The CNC data related to coordinates, zero offsets, etc. that are saved into the disk is not valid. The orgdata.tab file that contains these data does not exist, is not accessible or is corrupt.
CONSEQUENCE	The CNC data related to coordinates, zero offsets, parts counter, kinematics, etc. are lost.
SOLUTION	If the error persists after powering the CNC up several times, contact your supplier.

0006   **'Preparation takes longer than half the cycle time.**

DETECTION	On CNC startup or during execution.
CAUSE	Data preparation at the CNC per PLC cycle takes too long.
CONSEQUENCE	The PREPFREQ parameter does not have the desired effect.
SOLUTION	Decrease the value of the PREPFREQ parameter of the channel.

0007   **'To complete the reset, restart the CNC.**

DETECTION	After a CNC reset.
CAUSE	The user has reset the CNC twice in a row and none of them has ended correctly.
SOLUTION	Restart the CNC. If the user presses the [RESET] key for the third time, the CNC application shuts down.

0008   'The key has been rejected.

DETECTION   On CNC startup or during execution.

CAUSE   The CNC has detected that two incompatible keys have been pressed at the same time. The [START] key, the spindle start keys and the key for spindle orientation must always be pressed alone; if they are pressed at the same time, they are both canceled.

SOLUTION   If it has been by mistake, ignore the error. If the error persists or it comes up during start-up, verify that no keys are pressed (stuck) on the keyboard. If the error persists after powering the CNC up several times, contact your supplier.

0009   System warning: AEP fifo management failure.

DETECTION   During execution.

CAUSE   The program has a large number of G-functions or arithmetic parameters.

SOLUTION   Reduce the number of arithmetic parameters used or program #FLUSH (channel dump) at program points where small waits are unimportant.

0010   'Error when testing the RAM with battery'

DETECTION   During CNC startup.

CAUSE   The start-up test has detected a checksum error in the data of the NVRAM and, therefore, a failure in it.

CONSEQUENCE The saved data may not be correct (related to coordinates, zero offsets, parts counter, kinematics, etc.).

SOLUTION   If the error persists after powering the CNC up several times, contact your supplier.

0011   'Checksum error in block search data'

DETECTION   During CNC startup.

CAUSE   Checksum error in the data required to execute an automatic block search.

CONSEQUENCE An automatic block search cannot be executed.

SOLUTION   If the error persists after powering the CNC up several times, contact your supplier.

0012   There is no CPU time to execute all the PREPFREQ.

DETECTION   During execution.

CAUSE   The CNC needs too much time to analyse the program.

SOLUTION   Reduce the number of blocks to be prepared per cycle (parameter PREPFREQ).

0013   SYSTEM ASSERT.

DETECTION   During CNC startup.

CAUSE   The CNC has generated an undetermined error.

SOLUTION   If the error persists, contact Fagor Automation.

0014   Watchdog detected in last session.

DETECTION   During CNC startup.

CAUSE   The CNC has been closed due to a reset by watchdog.

SOLUTION   If the error persists, contact Fagor Automation.

0015   Detected equipment shutdown/restart.

DETECTION   During CNC startup.

CAUSE   An undetermined cause has led to a shutdown/restart of the equipment.

SOLUTION   If the error persists, contact Fagor Automation.

0016   OFF/ON detected while starting up the operative system.

DETECTION   During CNC startup.

CAUSE   The CNC has detected an unexpected shutdown due to a power failure, during the boot of the operating system.

SOLUTION   Check the CNC power supply. The CNC data relating to coordinates, zero offsets, parts counter, kinematics, etc. may be lost. If the error persists after powering the CNC up several times then contact Fagor Automation.

0017   OFF/ON detected due to power outage.

DETECTION   During CNC startup.

CAUSE   The CNC has detected an unexpected shutdown due to a power failure.

SOLUTION   Check the CNC power supply. The CNC data relating to coordinates, zero offsets, parts counter, kinematics, etc. may be lost. If the error persists after powering the CNC up several times then contact Fagor Automation.

0018   OFF/ON detected by blue screen.

DETECTION	During CNC startup.
CAUSE	The previous CNC shutdown was due to a low level exception (blue screen).
SOLUTION	The CNC data relating to coordinates, zero offsets, parts counter, kinematics, etc. may be lost. If the error persists after powering the CNC up several times then contact Fagor Automation.

0020   'Wrong access to a variable'

DETECTION	On CNC startup or when changing pages.
CAUSE	The CNC is accessing an interface variable that does not exist.
SOLUTION	Contact the machine manufacturer or the person who designed the interface screens to remove or correct the access to the variable.

0021   There is no CAN keyboard.

DETECTION	During CNC startup.
CAUSE	The CNC did not detect the CAN keyboard.
SOLUTION	Check the CAN bus status in diagnostic mode and verify whether the CNC recognizes the keyboard. If the error persists, contact the Service Department.

0022   'Variable that may be modified during setup'

DETECTION	When defining variables during setup.
CAUSE	A variable has been defined in the setup environment that is not admitted.
SOLUTION	Refer to the operating manual for the list of variables that are admitted.

0023   'The program must be stopped before initiating the trace for the first time.

DETECTION	When starting a trace at the oscilloscope.
CAUSE	It is the first time that the oscilloscope starts a trace and there is a program in execution. The trace uses drive variables that either are not in the machine parameters table or are in that table but are asynchronous.
SOLUTION	Stop the program in execution.

0024   'Error when initiating the trace'

DETECTION	When starting a trace at the oscilloscope.
CAUSE	There are no variables defined in the channels of the oscilloscope, there are more than two Sercos variables of the same drive or the syntax of one of the variables is wrong.
SOLUTION	Check the variables defined in the channels of the oscilloscope. The oscilloscope can only access two Sercos variables of each drive.

0025   'Error when registering PLC defines'

DETECTION	When registering the variables associated with the external PDEF symbols defined in the PLC program.
CAUSE	The plc_prg.sym that contains the necessary information to create the variables associated with the external PDEF symbols is corrupt.
SOLUTION	Delete the plc_prg.sym file and compile the PLC program to create this file again. If the error persists, contact your supplier.

0026   'Variable not allowed in the oscilloscope environment'

DETECTION	When defining variables defined in a channel of the oscilloscope.
CAUSE	The variable defined in the oscilloscope is dummy, binary, string or password protected.
SOLUTION	Check the documentation for the list of allowed variables.

0027   The axis is not mechatrolink.

DETECTION	When defining variables defined in a channel of the oscilloscope.
CAUSE	The variable or parameter is unique to a mechatrolink axis, however, the axis requesting the variable or parameter is not a mechatrolink axis
SOLUTION	Select a Mechatrolink axis. There must be an axis variable or parameter for the requested axis.

0028 A mechatrolink parameter cannot be associated with a channel of the oscilloscope.

DETECTION   When defining variables defined in a channel of the oscilloscope.

CAUSE   A mechatrolink parameter or a DRV variable associated with a mechatrolink parameter is defined in the oscilloscope.

SOLUTION   Access to variables and/or parameters of the Mechatrolink devices is only available for the Mlink-II mode with 32 bytes.

0029 'Only MLINKII admits DRV or MLINK variables in a channel of the oscilloscope'

DETECTION   When defining variables defined in a channel of the oscilloscope.

CAUSE   A Mechatrolink variable or a DRV variable is defined in the oscilloscope, but in the CNC the Mlink-I mode is active (parameter MLINK).

SOLUTION   Access to variables and/or parameters of the Mechatrolink devices is only available for the Mlink-II mode with 32 bytes.

0030 'The axis of the variable must be programmed using its name'

DETECTION   During execution.

CAUSE   In some variable of the drive (DRV), the axis is programmed using its logic number or index in the channel.

SOLUTION   In these variables, the axis must be programmed using its name.

0031 'Only MLINKII admits mechatrolink parameters in the oscilloscope'

DETECTION   When defining parameters in a channel of the oscilloscope.

CAUSE   A Mechatrolink parameter is defined in the oscilloscope, but in the CNC the Mlink-I mode is active (MLINK parameter).

SOLUTION   Access to variables and/or parameters of the Mechatrolink devices is only available for the Mlink-II mode with 32 bytes.

0032 Mechatrolink parameter or variable out of range.

DETECTION   During the validation of the machine parameters.

CAUSE   The parameter or variable has the wrong value.

SOLUTION   The valid Mechatrolink parameter range is MLINK0 to MLINKF. The valid Mechatrolink variable range is P0 to PFFFF.

0033 Array not allowed in Mechatrolink parameters and variables.

DETECTION   During execution.

CAUSE   The Mechatrolink parameters and variables (OEM machine parameters) do not admit arrays.

SOLUTION   Check the program. Each Mechatrolink parameter and variable can also be associated with an axis.

0034 The axis has not been programmed in the Mechatrolink parameter or variable.

DETECTION   During execution.

CAUSE   The axis has not been programmed in the Mechatrolink parameter or variable (OEM machine parameters).

SOLUTION   Check the program. The Mechatrolink parameters and variables have to be associated with an axis.

0035 Too many variables in oscilloscope.

DETECTION   At the oscilloscope.

CAUSE   There are more than two oscillograph channels with variables of the same controller.

SOLUTION   The maximum number of variables of the same controller that can be plotted simultaneously in the oscillogram is two.

0036 Inconsistency in the recovery of the active tool.

DETECTION   During CNC startup.

CAUSE   The information on the active tool stored on the disk does not correspond to that saved by the tool manager, either due to an improper shutdown or a system error. The file that contains these data does not exist, is not accessible or is corrupt.

SOLUTION   Check what the correct tool should be before continuing. Define in the tool table which tool is active, and if necessary, make a tool change. If the error persists after powering the CNC up several times, contact your supplier.

FAGOR ⬤
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·8·

0037   Inconsistency in the recovery of the next tool.

DETECTION   During CNC startup.

CAUSE   The information on the next tool stored on the disk does not correspond to that saved by the tool manager, either due to an improper shutdown or a system error. The file that contains these data does not exist, is not accessible or is corrupt.

SOLUTION   Check what the correct tool should be before continuing. For the CNC, define which is the next tool (for example via MDI) and make a tool change if required. If the error persists after powering the CNC up several times, contact your supplier.

0038   Inconsistency in the recovery of tool correctors.

DETECTION   During CNC startup.

CAUSE   The information on the correctors stored on the disk does not correspond to that saved by the tool manager, either due to an improper shutdown or a system error. The file that contains these data does not exist, is not accessible or is corrupt.

SOLUTION   Check what the correct correctors should be before continuing. In the tool table, define which tool is active and make a tool change if required. If the error persists after powering the CNC up several times, contact your supplier.

0039   Inconsistency in the recovery of the position in the magazine for the tool in the spindle.

DETECTION   During CNC startup.

CAUSE   The magazine position of the tool in the spindle does not correspond to the one stored by the tool manager due to an incorrect shutdown or a system failure. The file that contains these data does not exist, is not accessible or is corrupt.

SOLUTION   Check the correct position before proceeding. If the error persists after powering the CNC up several times then contact Fagor Automation.

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

DETECTION   During the validation of the machine parameters.

CAUSE   In the M functions table, there is a function with associated subroutine and Before-Before or Before-After synchronization type.

SOLUTION   The CNC always executes the subroutine associated with an M function at the end of the block where the function has been programmed. Define the M function without synchronization or with After-After synchronization.

0041   'Duplicate M in the table'

DETECTION   During the validation of the machine parameters.

CAUSE   There is a duplicate function in the M functions table.

SOLUTION   Correct the definition of the functions. The table cannot have two M functions with the same number.

0042   'Wrong machine parameter value'

DETECTION   During the validation of the machine parameters.

CAUSE   The machine parameter has the wrong value.

SOLUTION   Set the machine parameter with a value within the admitted limits. The error window shows which is the wrong parameter and the maximum and minimum value admitted.

0043   'Restart the CNC to assume the new value'

DETECTION   During the validation of the machine parameters.

CAUSE   The user has changed a machine parameter and the CNC application must be restarted in order to assume its new value.

SOLUTION   Restart the CNC.

0044   'Wrong axis name or undefined axis name'

DETECTION   During the validation of the machine parameters.

CAUSE   The machine parameter is not defined or it has the wrong axis name.

SOLUTION   Correct the machine parameters. The valid axis names are the ones defined in parameter AXISNAME.

In parameter AXISNAME, the axis name must be defined by 1 or 2 characters. The first character must be one of the letters X - Y - Z - U - V - W - A - B - C. The second character is optional and will be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range X, X1…X9,...C, C1…C9.

0045 **'Wrong spindle name'**

DETECTION   During the validation of the machine parameters.

CAUSE   The user has defined the machine parameter with the wrong spindle name.

SOLUTION   Correct the machine parameters. The valid spindle names are the ones defined in parameter SPDLNAME.

In parameter SPDLNAME, the spindle name must be defined by 1 or 2 characters. The first character must be the letter S. The second character is optional and will be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range S, S1 ... S9.

0046 **'Nonexistent axis'**

DETECTION   During the validation of the machine parameters.

CAUSE   The possible causes are:
- The user has assigned the name of an axis that does not exist to a machine parameter.
- Either the VMOVAXIS or VCOMPAXIS axis is undefined in the basic volumetric compensation.

SOLUTION   Correct the machine parameters. The valid axis names are the ones defined in parameter AXISNAME.

0047 **'A main axis cannot be defined as slave'**

DETECTION   During the validation of the machine parameters.

CAUSE   The slave axis of a gantry axis is the master of another gantry axis.

SOLUTION   Correct the gantry axes table. The master axis of a gantry axis cannot be the slave of another gantry axis.

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

DETECTION   During the validation of the machine parameters.

CAUSE   The slave axis of a gantry axis is already defined as the slave axis in another gantry axis.

SOLUTION   Correct the gantry axes table. An axis cannot be the slave of several masters.

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

DETECTION   During the validation of the machine parameters.

CAUSE   The master axis of a gantry axis is the slave of another gantry axis or vice versa.

SOLUTION   Correct the gantry axes table. The master axis of a gantry axis cannot be the slave of another gantry axis or vice versa.

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

DETECTION   During the validation of the machine parameters.

CAUSE   Both axes of a gantry pair are not of the same type; linear or rotary.

SOLUTION   The axes of a gantry pair must be of the same type, linear or rotary (parameter AXISTYPE). Check the gantry axes table and/or the AXISTYPE parameter of both axes.

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

DETECTION   During the validation of the machine parameters.

CAUSE   The two axes of a gantry axis do not have the same properties.

SOLUTION   Check the gantry axes table and/or the following machine parameters of the axes.
- The linear axes must have parameters AXISMODE, FACEAXIS and LONGAXIS set the same way.
- The rotary axes must have parameters AXISMODE, SHORTESTWAY and CAXIS set the same way.

0052 **'Module difference too small'**

DETECTION   During the validation of the machine parameters.

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

SOLUTION   Check the resolution of the axis; if correct, increase MODUPLIM or decrease MODLOWLIM.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·10·

0053 **'Parameter MGPAXIS repeated in several handwheels'**

DETECTION   During the validation of the machine parameters.

CAUSE   There are two or more handwheels assigned to the same axis in the Jog table parameters.

SOLUTION   One axis can only have one handwheel assigned to it.

0054 **'The MOVAXIS and COMPAXIS axis must be different'**

DETECTION   During the validation of the machine parameters.

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

SOLUTION   The two axes of each cross compensation table must be different (parameters MOVAXIS and COMPAXIS).

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

DETECTION   During the validation of the machine parameters.

CAUSE   In the cross compensation tables, checking the different associations of compensated (affected) axes (parameter COMPAXIS) and those (parameter MOVAXIS) whose movements affect the other ones, there is an axis whose movement is affected by itself.

SOLUTION   Check the relationship between the defined cross compensations. Check parameters MOVAXIS and COMPAXIS of the defined cross compensations.

0056 **'Compensation table positions not in ascending order'**

DETECTION   During the validation of the machine parameters.

CAUSE   In the compensation tables, the points to be compensated are not ordered correctly or the value to compensate in all the points have a zero value.

SOLUTION   The POSITION parameter within the compensation tables must have ascending values. The value to compensate cannot be zero in all the points.

0057 **'Compensation table with error slope greater than 1'**

DETECTION   While validating the machine parameters (leadscrew error compensation table)

CAUSE   In the leadscrew compensation tables, the difference between two consecutive errors is greater than the gap between those points.

SOLUTION   The error slope in leadscrew compensation tables cannot be greater than 1. Increase the gap between points; 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   During the validation of the machine parameters.

CAUSE   The CNC application 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   During the validation of the machine parameters.

CAUSE   The CNC application 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   During the validation of the machine parameters.

CAUSE   Parameter MAXMANFEED is greater than G00FEED.

SOLUTION   Decrease the value of parameter MAXMANFEED; it must be lower than G00FEED.

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

DETECTION   During the validation of the machine parameters.

CAUSE   Parameter JOGRAPFEED is greater than G00FEED.

SOLUTION   Decrease the value of parameter JOGRAPFEED; it must be lower than G00FEED.

0062 **'The continuous Jog feedrate exceeds the maximum feedrate set for the axis'**

DETECTION   During the validation of the machine parameters.

CAUSE   Parameter JOGFEED is greater than G00FEED.

SOLUTION   Decrease the value of parameter JOGFEED; it must be lower than G00FEED.

0063   'The incremental jog feedrate exceeds the maximum feedrate set for the axis'

DETECTION	During the validation of the machine parameters.
CAUSE	Parameter INCJOGFEED is greater than G00FEED.
SOLUTION	Decrease the value of parameter INCJOGFEED; it must be lower than G00FEED.

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

DETECTION	During the validation of the machine parameters.
CAUSE	The Io types of both gantry axes are not the same.
SOLUTION	Both axes must have the same type of reference mark (parameter I0TYPE).

0065   'A Hirth axis cannot be Gantry"

DETECTION	During the validation of the machine parameters.
CAUSE	'A hirth axis cannot be part of the gantry axis'
SOLUTION	'The axis cannot be a hirth axis (parameter HIRTH). Use another type of axis to make the gantry axis.

0066   'A Gantry axis cannot have REFSHIFT'

DETECTION	During the validation of the machine parameters.
CAUSE	One of the axes making up the gantry axis has parameter REFSHIFT set to a value other than zero in some set of parameters.
SOLUTION	Set parameter REFSHIFT all the sets to 0.

0067   'A Gantry axis cannot be unidirectional'

DETECTION	During the validation of the machine parameters.
CAUSE	A unidirectional rotary axis cannot be part of the gantry axis.
SOLUTION	The axis cannot be unidirectional (parameter UNIDIR). Use another type of axis to make the gantry axis.

0068   'Gantry/Tandem axes: the slave cannot go before the master in AXISNAME'

DETECTION	During the validation of the machine parameters.
CAUSE	The slave axis is defined before the master axis in the axis name assigning tables (parameter AXISNAME).
SOLUTION	In the table, define the master axis before the slave axis or swap the master and slave axes in the gantry or tandem pair.

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

DETECTION	During the validation of the machine parameters.
CAUSE	The slave axis of a gantry axis has a home switch but the master axis doesn't.
SOLUTION	In a gantry axis, there may be a home switch on the master axis, on both axes or on none (parameter DECINPUT).

0070   'Gantry/Tandem axes: LIMIT+ and LIMIT- must be the same for the master and the slave '

DETECTION	During the validation of the machine parameters.
CAUSE	In a gantry or tandem pair, the software travel limits are different on the two axes.
SOLUTION	Set both axes with the same software travel limits (parameters LIMIT+ and LIMIT-).

0071   'Following error monitoring is not active at the CNC'

DETECTION	During the validation of the machine parameters.
CAUSE	In some set of parameters, following error monitoring is not active. This situation can only be allowed during setup; once setup is completed, this watch must be enabled.
SOLUTION	Activate the following error monitoring in all sets of parameters (parameter FLWEMONITOR).

0072   'Feedback alarm not activated'

DETECTION	During the validation of the machine parameters.
CAUSE	In some set of parameters of an analog axis or spindle, the feedback alarm is not activated. This situation can only be allowed during setup; once setup is completed, this watch must be enabled.
SOLUTION	Activate the feedback alarm in all the sets of the analog axes and spindles (parameter FBACKAL).

0073 **'Software travel limits not activated'**

DETECTION	During the validation of the machine parameters.
CAUSE	The software travel limits of some axis are not activated. Both LIMIT+ and LIMIT- parameters of the axis are set to 0.
SOLUTION	Set the software travel limits of all the axes (parameters LIMIT+ and LIMIT).

0074 **'Tendency test not activated'**

DETECTION	During the validation of the machine parameters.
CAUSE	The tendency test of some axis or spindle is not activated. This situation should only be allowed during setup; once the setup is completed, the tendency test must be activated.
SOLUTION	Activate the tendency test for the axes and spindles (parameter TENDENCY).

0075 **'Wrong I/O configuration table'**

DETECTION	During the validation of the machine parameters.
CAUSE	Parameters NDIMOD and NDOMOD must be the same as the number of inputs and outputs detected by hardware.
SOLUTION	Correct parameters NDIMOD and NDOMOD.

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

DETECTION	During the validation of the machine parameters.
CAUSE	The value of parameter CHNAXIS is higher than the value of parameter NAXIS or the value of parameter CHNSPDL is higher than the value of NSPDL.
SOLUTION	Correct the machine parameters.

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

DETECTION	During the validation of the machine parameters.
CAUSE	There is an axis or spindle assigned to several channels.
SOLUTION	Correct machine parameters CHAXISNAME and CHSPDLNAME n all the channels. An axis or spindle can only belong to a channel or to none of them.

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

DETECTION	During the validation of the machine parameters.
CAUSE	Some gantry axis is formed by axes of different channels.
SOLUTION	Both axes of a gantry axis must belong to the same channel.

0079 **'A slave gantry axis cannot be parked'**

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The CNC has detected that the slave axis of a gantry pair is parked; the PARKED signal of the slave axis is active.
SOLUTION	Unpark the axis or cancel the gantry axis.

0080 **'To validate the axis, you must validate the GENERAL PARAMETERS table'**

DETECTION	During the validation of the machine parameters.
CAUSE	The user has added an axis to the system (parameter NAXIS) and, without validating the general parameters table, has tried to validate the parameter table of one of the new axes.
SOLUTION	Validate the general parameter table before validating the parameters of the axis.

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

DETECTION	During the validation of the machine parameters.
CAUSE	'The in-position zone is smaller than the resolution of the axis.
SOLUTION	Increase the in-position zone of the axis (parameter INPOSW).

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

DETECTION	During the validation of the machine parameters.
CAUSE	In a compensation, the moving axis (parameter MOVAXIS) is rotary and it doesn't have the same module limits in all the parameter sets.
SOLUTION	Assign the same module limits (parameters MODUPLIM and MODLOWLIM) in all the parameter sets.

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

DETECTION	During the validation of the machine parameters.
CAUSE	The two axes of a gantry axis have different types of drives; analog or Sercos or Mechatrolink.
SOLUTION	The axes of a gantry pair must have the same type of drive (parameter DRIVETYPE).

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

DETECTION	During the validation of the machine parameters.
CAUSE	There is an axis or spindle without swapping permission that is not assigned to any channel.
SOLUTION	The axes or spindles that cannot be swapped (parameter AXISEXCH) must necessarily be assigned to one channel.

0085   'There is no digital axis (sercos/Mechatrolink)'

DETECTION	During the validation of the machine parameters.
CAUSE	The OEM parameters table contains drive variables (DRV) but there are no digital axes in the system (Sercos or Mechatrolink).
SOLUTION	Eliminate the defined drive variables (DRV) or define the right digital axes.

0086   'It is not a digital axis (Sercos/Mechatrolink)'

DETECTION	During the validation of the machine parameters.
CAUSE	There is a drive variable (DRV) for an axis that is not digital (Sercos/Mechatrolink).
SOLUTION	Eliminate the variable of that axis.

0087   'Too many DRV variables'

DETECTION	During the validation of the machine parameters.
CAUSE	There are too many drive variables (DRV) defined in the OEM parameters table.
SOLUTION	The OEM parameters table can have up to 100 drive variables.

0088   'Trace of internal variables activated'

DETECTION	During the validation of the machine parameters.
CAUSE	The CNC is executing the trace of an internal variable.
SOLUTION	Contact Fagor.

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

DETECTION	During the validation of the machine parameters.
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 associated with it, but it is not on the list of the system axes.
SOLUTION	Being impossible to start up with the user configuration, the CNC starts up with the default configuration. Correct the machine parameter settings to eliminate the rest of errors and warnings. 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	During the validation of the machine parameters.
CAUSE	Errors or warnings have come up while validating machine parameters related to the axes or spindles of a channel. For example, the number of axes in the system is higher than the number of axes defined in parameter AXISNAME.
SOLUTION	Being impossible to start up with the user configuration, the CNC starts up with the default configuration. Correct the machine parameter settings to eliminate the rest of errors and warnings.

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

DETECTION	During the validation of the machine parameters.
CAUSE	There are drive variables (DRV) in the OEM machine parameters with the same Sercos identifier (ID) and different mnemonic.
SOLUTION	DRV variables with the same identifier must have the same mnemonic.

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

DETECTION   During the validation of the machine parameters.

CAUSE   There are drive variables (DRV) in the OEM machine parameters with the same identifier (ID) and different access type (synchronous or asynchronous) or different access mode (read or write).

SOLUTION   DRV variables with the same identifier must have the same type of access (parameter TYPE) and the same access mode (parameter MODE).

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

DETECTION   During the validation of the machine parameters.

CAUSE   There are drive variables in the OEM machine parameters with the same mnemonic and different Sercos identifier (ID), different access type (synchronous or asynchronous) or different access mode (read or write).

SOLUTION   The DRV variables with the same mnemonic must have the same Sercos identifier (parameter ID), the same type of access (parameter TYPE) and the same access mode (parameter MODE).

0094 **'Some channel must have parameter HIDDENCH = No'**

DETECTION   During the validation of the machine parameters.

CAUSE   All the channels of the system are defined as hidden.

SOLUTION   The CNC does not allow all the channels of the system to be hidden; some of them must be visible (parameter HIDDENCH).

0095 **'An axis cannot be MASTERAXIS of several Gantry pairs'**

DETECTION   During the validation of the machine parameters.

CAUSE   There are two gantry axes with the same master axis.

SOLUTION   Correct the configuration of the gantry axes.

0096 **A tandem axis must be Sercos type and with same OPMODE than the master.**

DETECTION   During the validation of the machine parameters.

CAUSE   The axes of a tandem axis are not velocity-Sercos.

SOLUTION   Both axes of a tandem axis must be velocity-Sercos (parameters DRIVETYPE and OPMODEP).

0097 **'Tandem/gantry pair eliminated because they were preceded by an empty pair'**

DETECTION   During the validation of the machine parameters.

CAUSE   The gantry or tandem pairs of axes do not occupy consecutive positions in their tables; there is an empty space or undefined position in one of them.

SOLUTION   Gantry or tandem pairs of axes must occupy consecutive positions in the tables. If there is an empty space in the table, i.e. an unassigned position, the CNC cancels the pairs defined behind it.

0098 **'The AXISEXCH parameter of the master and slave axes have been set differently'**

DETECTION   During the validation of the machine parameters.

CAUSE   There is a gantry o tandem pair whose master and slave axes have parameter AXISEXCH set differently.

SOLUTION   Assign the same value to parameter AXISEXCH of both axes.

0099 **'The value of PROBEFEED is too high to brake applying DECEL and JERK'**

DETECTION   During the validation of the machine parameters.

CAUSE   The maximum probing feedrate (parameter PROBEFEED) is higher than the feedrate needed to brake within the distance set by PROBERANGE with the acceleration and jerk values of the axis.

SOLUTION   The value of this parameter must be smaller than the feedrate needed to brake within the distance set by PROBERANGE with the acceleration and jerk values of the axis. The warning window shows the maximum feedrate that may be reached.

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

DETECTION   During execution.

CAUSE   The number of variables modified at the CNC that must be reported to the interface exceeds the maximum admitted.

SOLUTION   If the error persists, contact your supplier.

0103 **Validation of machine parameters not completed (timeout).**

DETECTION   During the validation of the machine parameters.

CAUSE   The parameterization has caused a crash in the machine parameter validation.

SOLUTION   Verify the machine parameters that have caused the crash (if known) or restart the CNC.

0104 **'Communication time out'**

DETECTION   During execution.

CAUSE   The CNC does not end successfully the reading / writing of an external variable.

SOLUTION   If the error persists, contact your supplier.

0105 **'Parameters cannot be validated while executing a program'**

DETECTION   During the validation of the machine parameters.

CAUSE   The user has attempted to validate a machine parameter table while a part-program is in execution or interrupted, or there is an independent movement being executed.

SOLUTION   Wait for the execution of the program or independent command to finalize or cancel it.

0106 **'Parameters cannot be validated: Spindle or axis in motion or in synchronization'**

DETECTION   During the validation of the machine parameters.

CAUSE   The user has tried to validate a machine parameter table while a spindle or an axis is moving. An axis may be moving as a result of a command of independent axis.

SOLUTION   Stop the axis or spindle movement.

0107 **'Error when registering DRV variables'**

DETECTION   During the validation of the machine parameters.

CAUSE   The CNC has generated an error when trying to register the variables defined in the OEM parameters table.

SOLUTION   Contact Fagor.

0108 **'Oscillo: The sampling time cannot be readjusted with the new LOOPTIME'**

DETECTION   During execution.

CAUSE   The user has used the oscilloscope without validating a trace, has modified parameter LOOPTIME and has validated the machine parameters.

SOLUTION   The warning will no longer be displayed when the user executes a trace at the oscilloscope. Setting the sampling period of a trace of the oscilloscope depends on parameter LOOPTIME. In order for the CNC to be able to set this, it must have a validated trace of the oscilloscope; i.e. the trace must have been executed at least once.

0109 **'Parameters cannot be validated: spindle in synchronization'**

DETECTION   On CNC power-up or when validating the machine parameters or when compiling the PLC program.

CAUSE   The CNC does not admit this type of actions with active synchronized spindles because they require a system reset.

SOLUTION   The parameters must be validated or the PLC program must be compiled before synchronizing the spindles or the spindles must be momentarily de-synchronized in order to carry out the desired action.

0110 **'The user kinematics could not be loaded'**

DETECTION   During CNC startup.

CAUSE   The CNC does not show the file c:\FagorCNC\drivers\kinematic.sys (or c:\Cnc8070\drivers\kinematic.sys).

SOLUTION   Checks that no errors come up when doing the make of the user kinematics and that the kinematic.sys driver is generated correctly.

0111 **'Error when initializing user kinematics data'**

DETECTION   During CNC startup.

CAUSE   Error in the function for initializing and loading the user kinematics data (UserTransforDataInit) implemented in the file Kin_iniData.c.

SOLUTION   Check and correct the possible reasons why this function generates an error.

0112 **'Error when initializing user kinematics'**

DETECTION   When activating a user kinematics.

CAUSE   Error in the function for initializing the kinematics (UserTransforInit) implemented in the file Kin_impl.c.

SOLUTION   Check and correct the possible reasons why this function generates an error.

0113 **'Error when initializing user kinematics parameters'**

DETECTION   When activating a user kinematics.

CAUSE   Error in the function for initializing the kinematics (UserTransforParamInit) implemented in the file Kin_impl.c.

SOLUTION   Check and correct the possible reasons why this function generates an error.

0114 **Parameters cannot be validated while simulating a program in the channel.**

DETECTION   During the validation of the machine parameters.

CAUSE   The user has attempted to validate the parameters during a program simulation.

SOLUTION   Halt the program simulation to validate the parameters.

0115 **Validation code -CPU type-, NOT coherent with the hardware.**

DETECTION   During CNC startup.

CAUSE   The type of CPU defined in the validation code is not valid for this hardware.

SOLUTION   Contact Fagor to obtain a valid validation code.

0116 **'Error reading technological table'**

DETECTION   During CNC start-up, access to the technology table or execution of the #TECHTABLE command.

CAUSE   The ..\mtb\data\techTable\TTDsc.xml file defining the table or ..\mtb\data\techTable\Material Files*xml with the table data does not exist or is not correct. There may be changes in the table (new variables, new enumerated types) and the CNC has not been restarted.

SOLUTION   Check the name and content of the table. If there is a table, restart the CNC.

0117 **The defined number of cores in the validation code is NOT consistent with the hardware.**

DETECTION   During CNC startup.

CAUSE   The defined number of cores in the validation code is not valid for the hardware.

SOLUTION   Contact Fagor to obtain a valid validation code.

0118 **No SERCOS III hardware found or driver load error.**

DETECTION   On CNC power-up or when validating the machine parameters.

CAUSE   The CNC cannot detect the Sercos III associated hardware or it cannot load the driver.

SOLUTION   Restart the CNC. If the error persists, contact Fagor Automation.

0119 **Correct SW Update for PwC devices connected to SERCOS3 - OK.**

DETECTION   During CNC startup.

CAUSE   The CNC has updated the PwC software.

SOLUTION   The CNC informs of the update; no action is required.

0120 **SW Update Error for devices connected to SERCOS3 - FAILED.**

DETECTION   During CNC startup.

CAUSE   The CNC has not updated the software for the devices connected to the Sercos III bus.

SOLUTION   The CNC informs that the update has failed. The device works with the previous software version. If the error persists after powering the CNC up several times then contact Fagor Automation.

0121 **SW Update Error for devices connected to SERCOS3 - S3 connect FAILED.**

DETECTION   During CNC startup.

CAUSE   The CNC has not updated the software for the devices connected to the Sercos III bus.

SOLUTION   The CNC informs that the update has failed. The device works with the previous software version. If the error persists after powering the CNC up several times then contact Fagor Automation.

0122 Not possible to validate machine parameter change.

DETECTION	During CNC startup.
CAUSE	The CNC cannot validate the machine parameters because it has not finished starting up.
SOLUTION	Wait for the CNC to start up to validate the machine parameters.

0123 SW Update Error for devices connected to SERCOS3 - Checksum error.

DETECTION	During CNC startup.
CAUSE	The CNC has not updated the software for the devices connected to the Sercos III bus.
SOLUTION	The CNC informs that the update has failed. The device works with the previous software version. If the error persists after powering the CNC up several times then contact Fagor Automation.

0124 The defined number of cores in the validation code is NOT consistent with the hardware.

DETECTION	During CNC startup.
CAUSE	The defined number of cores in the validation code is not valid for the hardware.
SOLUTION	Contact Fagor to obtain a valid validation code.

0125 Invalid file read value.

DETECTION	During execution.
CAUSE	The CNC has found incorrect data in the file it is loading; for example, when loading non-linear gap calibration data (#LOAD).
SOLUTION	Define the data correctly. See the error window for the maximum and minimum values.

0126 Variable read/write not completed (timeout).

DETECTION	During execution.
CAUSE	The minimum timeout allotted to the variable read/write process has been exceeded.
SOLUTION	Reset the error. If the error persists after powering the CNC up several times then contact Fagor Automation.

0127 Error in recording PLD device S3, invalid device for PLD version.

DETECTION	During CNC startup.
CAUSE	Some of the Sercos III devices are not valid for this version of PLD.
SOLUTION	Contact Fagor Automation.

0128 Error in recording PLD device S3, invalid PLD version file.

DETECTION	During CNC startup.
CAUSE	The PLD version of the CNC is not valid for the Sercos III device.
SOLUTION	Contact Fagor Automation.

0129 Invalid device.

DETECTION	During execution.
CAUSE	The device associated with the regulator variable to be read or written is invalid.
SOLUTION	If the error was generated by reading a variable from the DRV table, review the parameters associated with the variable in the DRV table. If the error persists, contact Fagor Automation.

0130 EtherCAT not available.

DETECTION	During execution.
CAUSE	The EtherCAT controller variable cannot be read or written because the bus is not operational.
SOLUTION	Contact Fagor Automation.

0131 Error in the reading of controller variable.

DETECTION	During execution.
CAUSE	The function for reading the controller variables has generated an error.
SOLUTION	If the error was generated by reading a variable from the DRV table, review the parameters associated with the variable in the DRV table. If the error persists, contact Fagor Automation.

0132   **Error in writing regulator variable.**

DETECTION	During execution.
CAUSE	The write function of the controller variables has generated an error.
SOLUTION	If the error was generated in the writing of a variable in the DRV table, review the parameters associated with the variable in the DRV table. If the error persists, contact Fagor Automation.

0150   **#OPEN: file handle in use.**

DETECTION	During execution.
CAUSE	The file handle indicated in the instruction #OPEN is being used by an #OPEN of another file.
SOLUTION	Use a different file handle.

0150   **'Too many open files'**

DETECTION	While executing a part-program with global 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   **#WRITE: text or cncWrite.txt file not found.**

DETECTION	During execution.
CAUSE	The text corresponding to the programmed identifier not found. It is possible that the cncWrite.txt file does not exist or that the identifier is not defined.
SOLUTION	Check that the cncWrite.txt file exists and that it contains the identifier.

0151   **'Writing access denied'**

DETECTION	When accessing a file.
CAUSE	The CNC has tried to write in a file that does not have a writing permission.
SOLUTION	Give the file writing permission.

0152   **'The file cannot be opened'**

DETECTION	When accessing a file.
CAUSE	It is not possible to open the file specified in the #OPEN instruction.
SOLUTION	Check that the file exists and that it has the appropriate permissions. If the MUTED option has been programmed in the #OPEN instruction, the variable G.FILEERRNO indicates the cause of the error.

0153   **'Reading access denied'**

DETECTION	When accessing a file.
CAUSE	The CNC has tried to read a file that does not have a reading permission.
SOLUTION	Give the file reading permission.

0154   **Protected program or routine'**

DETECTION	When accessing a file.
CAUSE	The CNC has tried to read an encrypted file without having permission to read.
SOLUTION	Contact the manufacturer of the machine to obtain the file encrypting codes.

0155   **'The quick charge of the file is not possible'**

DETECTION	When accessing a file.
CAUSE	The CNC has tried to read a file whose size is larger than the memory area assigned for file quick charge.
SOLUTION	Execute from RAM memory (subroutines with extension fst) only the files that are called upon more often.

0156   **Demasiados ficheros abiertos para escritura.**

DETECTION	During execution.
CAUSE	The #OPEN instruction has attempted to open too many files.
SOLUTION	Close a file.

0157   #OPEN/#CLOSE: Wrong file handle

DETECTION	During editing and execution.
CAUSE	The file identifier is wrong.
SOLUTION	Check the syntax of the instruction in the programming manual. Valid identifiers are F1 to F4 and their programming is optional. The file identifier allows for a channel to write to a file opened in an other channel.

0158   #WRITE requires a previous #OPEN.

DETECTION	During execution.
CAUSE	The CNC has attempted to write (#WRITE) to a file that is not open.
SOLUTION	The CNC must open a file (#OPEN) before writing to it (#WRITE).

0159   #OPEN: File already open in another channel.

DETECTION	During execution.
CAUSE	A channel tried to open a file that is already opened in another channel.
SOLUTION	The possible solutions are: • Close the file (#CLOSE). • It is not necessary to open the same channel in all channels. A channel can write to a file opened via another channel if it knows the file identifier (F1 to F4).

0160   'Axis/Set not available in the system'

DETECTION	During execution.
CAUSE	The possible causes are: • The instruction #SET AX or #CALL AX is trying to add to a channel an axis that does not exist or is in another channel. • A parameter set that does not exist has been programmed in function G112.
SOLUTION	Check the program. If the axis is in another channel, release it with the instruction #FREE AX.

0161   #OPEN: APPEND mode not allowed for a non-Unicode file.

DETECTION	During execution.
CAUSE	A channel attempted to open a non-Unicode file in "APPEND" access mode.
SOLUTION	Convert the file to Unicode format with a third-party editor. The "APPEND" access mode only allows Unicode files to be opened. The "DELETTE" access mode can open non-Unicode files, but it deletes their content.

0162   #OPEN: The parameter TYPE does not match the file format.

DETECTION	During execution.
CAUSE	The file format does not match the format selected in TYPE.
SOLUTION	The possible solutions are: • On opening the file with the "A" command (APPEND), the TYPE command must match the file format. If the file does not exist, create a new one in the format specified by the Command TYPE. • Open a file with the "D" command (DELETE). The CNC deletes the file and creates a new one in the format specified in the command TYPE.

0163   #OPEN: The indicated file or path cannot be found.

DETECTION	During execution.
CAUSE	The instruction #OPEN has verified that the file does not exist ("R" command).
SOLUTION	The instruction #OPEN has been programmed to display this error. To resume the execution without prompting an error, use the "MUTED" command.

0165   'RT IT Overflow'

DETECTION	During CNC operation.
CAUSE	The real time interruptions exceed the allowed amount of time. Some possible causes may be the installation of some device, driver or application that is incompatible with the CNC.
SOLUTION	If the error comes up often, it may be necessary to adjust the parameter LOOPTIME. Analyze the cases where the error occurs and contact your supplier.

0166   'Jerk limit overshoot'

DETECTION	During the execution of a part-program.
CAUSE	The axis is exceeding its jerk limit.
SOLUTION	Analyze the cases where it occurs and contact your machine manufacturer.

0167   **'No RT IT'**

DETECTION   During CNC operation.

CAUSE   The real time interruption does not kick in.

SOLUTION   Restart the CNC. If the error persists, contact your supplier.

0168   **Position Loop Overflow.**

DETECTION   During CNC operation.

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

SOLUTION   Adjust parameter LOOPTIME.

0169   **Safety temperature exceeded. Unit integrity at risk. Contact SAT.**

DETECTION   During CNC operation.

CAUSE   Unit integrity is at risk The CNC checks every minute the unit temperature; if in three samples in a row the temperature exceeds 60 ºC (140 ºF), the CNC issues this warning and activates the OVERTEMP mark. The temperature increase may be due to a failure in the hardware cooling system or to excessive room temperature. This warning shows the current temperature.

SOLUTION   Respect the dimensions recommended for the enclosure minimum distance recommended between the enclosure walls and the central unit. If necessary, install fans for cooling the enclosure. If the error persists, turn the CNC off and contact the Service Department.

0170   **'Low battery voltage'**

DETECTION   During CNC power-up or after a reset.

CAUSE   The CNC checks the battery voltage on start-up and at each reset. The battery is discharged; its useful life cycle has ended.

SOLUTION   Contact the manufacturer to replace the battery. When the CNC is turned off, the battery keeps the necessary data for the CNC (for example, the position values).

0171   **'LOOPTIME overflow'**

DETECTION   In CNCREADY conditions.

CAUSE   The real time interruptions exceed the allowed amount of time.

SOLUTION   If the error comes up often, it may be necessary to adjust the parameter LOOPTIME. Analyze the cases where it occurs and contact your machine manufacturer.

0172   **CPU fan stopped. Overtemperature risk. Contact SAT.**

DETECTION   During CNC operation.

CAUSE   Unit over-temperature risk The CNC has detected that the CPU fan is stopped. When the CPU has a fan, during regular operation of the CNC, it monitors and verifies that the fan is running. This test is run every minute, same as the temperature watch.

SOLUTION   If the error persists, contact the Service Department.

0173   **Start not allowed due to safety overtemperature. Turn off the CNC and contact SAT.**

DETECTION   During CNC operation.

CAUSE   Every time [START] is pressed, the CNC checks that the room temperature does not exceed 65 ºC (149 ºF) and, if it does, inhibits the [START] while the error occurs. The temperature increase may be due to a failure in the hardware cooling system or to excessive room temperature. This error shows the current temperature.

SOLUTION   Respect the dimensions recommended for the enclosure minimum distance recommended between the enclosure walls and the central unit. If necessary, install fans for cooling the enclosure. If the error persists, turn the CNC off and contact the Service Department.

0174   **Rotary axes, it is not required to complete a full turn.**

DETECTION   During the validation of the machine parameters.

CAUSE   The CNC cannot activate the volumetric compensation because there is more than one rotary axis module between the moving axes (MOVAXIS parameter), and the travel of the second axis is not complete (0 to 360º).

SOLUTION   Define the modulus of the second rotary axis between 0 and 360° and validate the volumetric compensation.

0200   **'Failure when requesting a VxD'**

DETECTION   When reading the battery status.

CAUSE   The CNC cannot be connected with VcompciD.

SOLUTION   Contact your supplier.

0201 **'Mains failure. PC powered by a battery'**

DETECTION   When reading the battery status.

CAUSE   There has been a CNC power failure and the emergency battery is powering the CNC.

SOLUTION   Whether the mains failure is fortuitous or caused by the operator, let the CNC finish its shut-down sequence. If the mains failure has been fortuitous, check the possible causes.

0250 **DYNOVR out of range allowed by MINDYNOVR and MAXDYNOVR.**

DETECTION   During execution.

CAUSE   The variable V.G.DYNOVR contains a value beyond the limits set by the parameters MINDYNOVR and MAXDYNOVR.

SOLUTION   Define the variable with an appropriate value or modify the dynamics using the bar-override ([CTRL] [H]).

0251 **Nonexistent volumetric compensation.**

DETECTION   During execution.

CAUSE   The PLC has attempted to activate a volumetric compensation that does not exist.

SOLUTION   Check that the software option associated with volumetric compensation is active. Check the list of defined volumetric compensations (parameter VOLCOMP n).

0252 **Nonexistent variable without an active material.**

DETECTION   During execution.

CAUSE   An attempt was made to read the name of the active material but it does not exist.

SOLUTION   Activate a material from the technological tables.

0253 **COMPID not valid.**

DETECTION   During the validation of the machine parameters.

CAUSE   The COMPID parameter of the kinematics must be less than 7 and there cannot be more than one kinematics with the same COMPID.

SOLUTION   Check the COMPID parameter.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·22·

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   The speed of the spindle used with function G63 is zero.

SOLUTION   Program a spindle speed.

1005 'Motion block with zero feed'

DETECTION   During execution.

CAUSE   No feedrate is active in the channel.

SOLUTION   Program the feedrate F.

1006 'G20: spindle not allowed'

DETECTION   During execution.

CAUSE   Function G20 does not allow programming the spindle.

SOLUTION   Check the program.

1007 'The programmed function requires an nonexistent main axis'

DETECTION   During execution.

CAUSE   The programmed function needs one or two main axes and these do not exist or are parked.

SOLUTION   Check the program.
- Following functions G11, G12, G13 and G14 require one of the two axes of the main plane.
- Functions G2, G3, G8, G9, G30, G36, G37, G38, G39, G73 require both axes of the main plane.
- Both axes of the main plane are also needed to activate collision detection (#CD) and for function G20 when collision detection is active.
- Restore the missing axis with the instruction #CALL AX/#SET AX.
- Unpark the axis (#UNPARK).

1008 'Coordinates out of range'

DETECTION   During execution.

CAUSE   The possible causes are:
- The coordinate programmed for the axis is too large.
- Function G101 tries to include an offset too large in the axis.

SOLUTION   Check the program.

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

DETECTION   During execution.

CAUSE   The dwell has been programmed twice in the G4 function and in the same block, directly with a number and with parameter K.

SOLUTION   Program the dwell function G4 only once.

1010 'Program G4 K'

DETECTION   During execution.

CAUSE   The dwell time has not been programmed in function G4.

SOLUTION   Program G4 as G4 <time> or G4K<time>, where <time> is the dwell time in seconds. In both cases, the dwell must be programmed after G4.

1011   'G4: dwell out of range'

DETECTION   During execution.

CAUSE   The dwell time programmed in function G4 is too long.

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   The dwell time programmed in function G4 is negative.

SOLUTION   Program a value equal to or greater than 0.

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

DETECTION   During execution.

CAUSE   The face axis (parameter FACEAXIS) cannot have both the mirror image and programming in diameters active at the same time.

SOLUTION   Check the program.

1015   'Center coordinates out of range'

DETECTION   During execution.

CAUSE   One of the I, J, K values is too high for the center of the circular interpolation or for the center of rotation of the coordinate system.

SOLUTION   Program a smaller value.

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

DETECTION   During execution.

CAUSE   Negative coordinates cannot be programmed in absolute coordinates (G90) while programming in diameters is active (parameter DIAMPROG).

SOLUTION   Programming in absolute coordinates and diameters does not admit negative coordinates.

1017   'G198: negative software limit out of range'

DETECTION   During execution.

CAUSE   The value of the negative software limit is too high.

SOLUTION   Check the program.

1018   'G199: positive software limit out of range'

DETECTION   During execution.

CAUSE   The value of the positive software limit is too high.

SOLUTION   Check the program.

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

DETECTION   During execution.

CAUSE   Function G101 tries to include a measuring offset in an axis that was not involved in the measurement or the offset has been canceled (G102).

SOLUTION   To include a measuring offset (G101), the axis must have carried out a measurement.

1020   'Negative ramp time'

DETECTION   During execution.

CAUSE   The ramp time of function G132 is negative.

SOLUTION   Program a value equal to or greater than 0.

1021   'Ramp time out of range'

DETECTION   During execution.

CAUSE   The ramp time of function G132 is too long.

SOLUTION   Check the program.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·24·

1022   'Percentage of Feed-Forward out of range'

DETECTION	During execution.
CAUSE	The percentage of feed forward (G134) or AC forward (G135) is too high.
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 set number of 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 set number of the axis is too high.
.SOLUTION	The set programmed for the axis must be greater than zero and smaller than or equal to machine parameter NPARSETS of the axis.

1025   'Programmed distance equal to zero'

DETECTION	During execution.
CAUSE	Null movement in the G63 block.
SOLUTION	Check the program.

1026   'Wrong circular path with the programmed radius'

DETECTION	During execution.
CAUSE	The 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	Zero radius for the circular interpolation; 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	Check the program.

1029   'Zero radius on circular path'

DETECTION	During execution.
CAUSE	The possible causes are: • Zero radius in a circular interpolation. • Being function G265 active, the CNC calculates a zero radius based on the center coordinates programmed in the circular interpolation. • Function G264 being active, both center coordinates are zero.
SOLUTION	The radius of a circular interpolation cannot be zero. Both coordinates of the center of a circular interpolation cannot be zero.

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

DETECTION	During execution.
CAUSE	G200, G201 or G202 are missing in the block of the #AXIS instruction.
SOLUTION	Check the program.

1031   '#AXIS expected'

DETECTION	During execution.
CAUSE	The #AXIS instruction is missing in the block of function G201.
SOLUTION	Check the program.

1032   'Spindle position missing for M19'

DETECTION	During execution.
CAUSE	The spindle position is missing in the block of function M19.
SOLUTION	Check the program.

1035   '#SLOPE: parameter out of range'

DETECTION	During execution.
CAUSE	The value of some parameter of the #SLOPE instruction is too high.
SOLUTION	Program smaller values.

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

DETECTION	During execution.
CAUSE	The CNC has found parameters I, J, K with function G0, G1, G100 or G63 active. The CNC ignores these parameters.
SOLUTION	These functions do not need these parameters.

1038   'Radius compensation cannot be active while measuring'

DETECTION	During execution.
CAUSE	The CNC tries to execute G100 while tool 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	The CNC tries to carry out a measurement (G100) on an axis that has a previous measuring offset.
SOLUTION	Use function G102 to cancel the measuring offset included in the axis.

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

DETECTION	During execution.
CAUSE	The CNC cannot be home an axis that is in additive manual mode (G201).
SOLUTION	Use function G202 to cancel the additive manual mode of the axis in order to home it. After the home search, activate the additive manual mode (G201) again.

1041   'Corrected circular path center out of range'

DETECTION	During execution.
CAUSE	In a circular interpolation programmed either with a radius and the coordinates of the end point or with the coordinates of the middle point, end point and having function G265 active.
	The coordinates of the center of the interpolation calculated by the CNC are too large. The coordinates programmed for the center, middle point or the radius are too large.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	Parameter 5 of 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 coincide with the first one or with the second one.

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

DETECTION	During execution.
CAUSE	In function G20 (plane change), the first (parameter 1) and the second (parameter 2) axes of the plane are the same axis.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	In function G20 (plane change) the first axis of the plane (parameter 1) is wrong.
SOLUTION	The first axis of the plane must be one of the first three axes of the channel.

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) is wrong.
SOLUTION	The second axis of the plane must be one of the first three axes of the channel.

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

DETECTION   During execution.

CAUSE   In function G20 (plane change) the third axis of the plane is missing or is 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. The parameter must neither coincide with the first or with the second one and must be one of the first three axes of the channel.

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 (parameter FACEAXIS).

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.

1051 **'The axis does not exist or is not available in the channel'**

DETECTION   During execution.

CAUSE   The possible causes are:
  • The CNC has tried to execute an independent movement in a spindle.
  • The axis programmed in a variable is not available.

SOLUTION   Check the program. In order to be able to interpolate the spindle as independent axis, it must be active as C axis.

1052 **'Values resulting from the measurement out of range'**

DETECTION   During execution.

CAUSE   The CNC has carried out a measurement with function G100 and the coordinate or offset obtained is too large.

SOLUTION   The value obtained in that probing move must be between -2147483647 and 2147483646.

1153 **'Too many local subroutines defined in the program'**

DETECTION   During execution.

CAUSE   The program has more local subroutines than admitted by the CNC.

SOLUTION   Use fewer local subroutines; group several subroutines in one or use global subroutines. The CNC admits 130 local subroutines per program.

1054 **'Nonexistent fixture'**

DETECTION   During execution.

CAUSE   The programmed fixture (clamp) number does not exist.

SOLUTION   The fixture (clamp) number must be a value between 0 and 10.

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

DETECTION   During execution.

CAUSE   The CNC does not allow modifying the tool radius (variable: (V.)G.TOR) or programming a tool change and/or tool offset in the same block.

SOLUTION   Program both instructions in different blocks.

1056 **'Too many external variables'**

DETECTION   During execution.

CAUSE   The CNC has reached the maximum number of external variables.

SOLUTION   Check the program. The CNC admits up to 500 external variables.

1057 **'Variable without reading permission'**

DETECTION	During execution.
CAUSE	The CNC has tried to read, via part-program or MDI, a variable that does not have reading permission via program.
SOLUTION	The variable cannot be read from a part-program or MDI. Refer to the documentation on permissions of the variable.

1058 **'User variable not initialized'**

DETECTION	During execution.
CAUSE	The user variable V.P.name or V.S.name has not been defined.
SOLUTION	Check the program. Define the variable properly.

1059 **'Variable without writing permission'**

DETECTION	During execution.
CAUSE	The CNC has tried to write, via part-program or MDI, a variable that does not have writing permission via program.
SOLUTION	The variable cannot be written from a part-program or MDI. Refer to the documentation on permissions of the variable.

1060 **'N label value out of range'**

DETECTION	During execution.
CAUSE	Invalid block number "N".
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 block.
SOLUTION	Program the functions in different blocks.

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

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

1064 **'Incompatible G functions (G196/G197)'**

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

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

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

1066 **'Incompatible G functions (G136/G137)'**

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

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

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

1068 **'Incompatible G functions (G151/G152)'**

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

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

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

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

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

1071 **'Incompatible G functions (G70/G71)'**

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

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

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

1073 **'Incompatible G functions (G90/G91)'**

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

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

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

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

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

1076 **'Incompatible G functions (G100/G101/G102)'**

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

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

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

1078 **'Incompatible G functions (G134/G135)'**

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

1079 **'Incompatible G functions (G138/G139)'**

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

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

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

1081 **'Incompatible G functions (G264/G265)'**

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

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

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

1083 **'Incompatible G functions (G36/G37/G38/G39)'**

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

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

DETECTION	During execution.
CAUSE	The CNC has tried to change the work plane or the axes that make up the plane, while tool radius 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	Define the work plane. If the channel has yielded its axes to other channels, recover the missing axis using the instruction #CALL AX or #SET AX.

1087 **'"=" expected'**

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

1088 **'Wrong offset number'**

DETECTION	During execution.
CAUSE	In function G159, the programmed zero offset does not exist.
SOLUTION	Check the program.

1089 **Incompatible M functions (M3/M4/M5/M19, M8-M9, M41-M44).**

DETECTION	During execution.
CAUSE	Two or more M functions incompatible with each other have been programmed for the same spindle in the block.
SOLUTION	Program the M functions of the same spindle in different blocks.

1090   'Nonexistent H function'

DETECTION   During execution.
CAUSE   The H function does not exist.
SOLUTION   The function number must be between 1 and 65534.

1091   'T function programmed twice'

DETECTION   During execution.
CAUSE   More than one T function have been programmed in the same block.
SOLUTION   Each block can only contain one T function. Program both functions 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   Each block can only contain one D function. Program both functions 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   Each block can only contain one F function. Program both functions 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   The feedrate has been programmed with the E function.
SOLUTION   Program the feedrate with the F function.

1097   'Unknown spindle name'

DETECTION   During execution.
CAUSE   The spindle name is not valid, the spindle does not exist in the system or the spindle does not belong to the channel.
SOLUTION   The valid names for the spindle are S, S1, ..., S9. The spindle programmed to in the block must exist in the system configuration and depending on which instruction it is, also in the channel configuration. A channel can only control its spindles.

1098   'S speed programmed twice'

DETECTION   During execution.
CAUSE   There are two or more S functions programmed for the same spindle in the same block.
SOLUTION   There can only be one speed for each spindle in the same block.

1100   'Parameter index out of range'

DETECTION   During execution.
CAUSE   The arithmetic parameter does not exist; it is not within the range admitted by the machine parameters.
SOLUTION   Check the program. Check the valid arithmetic parameter range in the machine parameters.

Machine parameters.	Valid range.
MINLOCP - MAXLOCP	Local arithmetic parameters.
MINGLBP - MAXGLBP	Global arithmetic parameters.
MINCOMP - MAXCOMP	Common arithmetic parameters.

1101 **'#SET IPOPOS instruction programmed wrong'**

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added.

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 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 program.

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

DETECTION	During execution.
CAUSE	The CNC tries to move an axis that does not exist or is not available in the system or in the channel. The axis programmed in an instruction or variable does not exist in the system or in the channel.
SOLUTION	Verify that the programmed axis exists in the channel and that it is available (is not parked).

1108 **'Axis programmed twice'**

DETECTION	During execution.
CAUSE	In one of the following functions, an axis has been programmed more than once.
	• Axis movement in G0, G1, G2, G3, G8 or G9.
	• Threading G33 or G63.
	• Instructions #FACE or #CYL.
	• Plane selection, G20.
	With functions that imply axis movements, programming an axis twice may be because the axis has been programmed in both Cartesian and Polar coordinates.
SOLUTION	Check the program.

1109 **'Wrong axis index'**

DETECTION	During execution.
CAUSE	In functions G20 and G74, the index programmed with the axis name is wrong.
SOLUTION	The axis index must be between 1 and the maximum number of axes of the system or channel.

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

DETECTION	During execution.
CAUSE	One of parameters I, J, K has been programmed more than once in the same block.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	An instruction has not been programmed alone in the block.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added. The only exception is to program $IF and $GOTO in the same block.

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

DETECTION	During execution.
CAUSE	The instruction is not programmed alone in the block and the additional information is not a $GOTO.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added. The only exception is to program $IF and $GOTO in the same block.

1113 **'$ELSE not expected'**

DETECTION	During execution.
CAUSE	The CNC has detected an $ELSE instruction without a previous $IF instruction.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The instruction has not been programmed alone in the block.
SOLUTION	Check the program.

1115 **'$ELSEIF not expected'**

DETECTION	During execution.
CAUSE	The CNC has detected an $ELSEIF instruction without a previous $IF instruction.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The instruction has not been programmed alone in the block.
SOLUTION	Check the program.

1117 **'$ENDIF not expected'**

DETECTION	During execution.
CAUSE	The CNC has detected an $ENDIF instruction without a previous $IF instruction.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The instruction has not been programmed alone in the block.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The instruction has not been programmed alone in the block.
SOLUTION	Check the program.

1120 **'$CASE not expected'**

DETECTION	During execution.
CAUSE	The CNC has detected an $CASE instruction without a previous $SWITCH instruction.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The instruction has not been programmed alone in the block.
SOLUTION	Check the program.

1122   '$DEFAULT not expected'

DETECTION   During execution.
CAUSE   The CNC has detected a $DEFAULT instruction without a previous $SWITCH instruction.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

1124   '$ENDSWITCH not expected'

DETECTION   During execution.
CAUSE   The CNC has detected an $ENDSWITCH instruction without a previous $SWITCH instruction.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

1126   '$FOR: invalid counter variable'

DETECTION   During execution.
CAUSE   The counter of the $FOR instruction is not valid.
SOLUTION   The counter of the $FOR instruction may be a variable or an arithmetic parameter.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The block that contains the $FOR instruction has more than 5100 characters.
SOLUTION   Write the block that contains the $FOR instruction shorter.

1129   '$ENDFOR not expected'

DETECTION   During execution.
CAUSE   The CNC has detected an $ENDFOR instruction without a previous $FOR instruction.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been 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   The instruction has not been 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 $WHILE instruction 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 CNC has detected an $ENDWHILE instruction without a previous $WHILE instruction.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

1136   **'$ENDDO not expected'**

DETECTION   During execution.
CAUSE   The CNC has detected an $ENDDO instruction without a previous $DO instruction.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

1139   **'$BREAK not expected'**

DETECTION   During execution.
CAUSE   The CNC has detected a $BREAK instruction, but there is no open control loop; $IF, $ELSE, $FOR, $WHILE, $DO or $CASE.
SOLUTION   Check the program. The CNC uses the $BREAK instruction is used to end a $CASE or to exit from an $IF, $ELSE, $WHILE, $FOR or $DO loop before it ends.

1140   **'$CONTINUE not expected'**

DETECTION   During execution.
CAUSE   The CNC has detected a $CONTINUE instruction, but there is no open control loop; $FOR, $WHILE or $DO.
SOLUTION   Check the program. The CNC uses the $CONTINUE instruction to return to the starting point of a $FOR, $WHILE or $DO loop.

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

DETECTION   During execution.
CAUSE   The instruction has not been 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 instruction has not been programmed alone in the block.
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 CNC cannot create the blending tool path with the programmed radius.
SOLUTION   Check the programmed radius. 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 CNC does not have a second motion block for generating the joining tool path (blend).
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 CNC does not have a first motion block for generating the joining tool path (blend).
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 has exceeded the maximum number of nesting levels.
SOLUTION	Correct the program decreasing the number of calls to subroutines (local and global) that imply a new nesting level. The CNC admits up to 20 nesting levels.

1153   'Too many local subroutines defined in the program'

DETECTION	During execution.
CAUSE	The program has more local subroutines than admitted by the CNC.
SOLUTION	Use fewer local subroutines; group several subroutines in one or use global subroutines. The CNC admits 100 local subroutines per program.

1154   'File name too long'

DETECTION	During execution.
CAUSE	The file name has too many characters. The name of a program or subroutine may have a maximum of 63 characters and the path 120. When programming the name of a program or subroutine with the 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. Change the location of the program or subroutine to reduce the number of characters of the path.

1155   File access impossible.

DETECTION	During execution.
CAUSE	It is not possible to open the file programmed in the instruction. The programming of the file may be direct or associated with an element of the block (G function with associated subroutine).
SOLUTION	Check that the file exists, is valid and not corrupted. When calling subroutines, check that the name and the path are correct. If the path is not defined in the call to the subroutine, the CNC will apply the default search criterion (see the programming manual).

1156   'Main program not found'

DETECTION	During execution.
CAUSE	The CNC cannot find the main program.
SOLUTION	In a program with local subroutines, the main program must have a name (%name).

1157   'Global subroutine not found'

DETECTION	During execution.
CAUSE	The CNC cannot find the global subroutine.
SOLUTION	Check that the name and the path of the subroutine are correct. If the path is not defined in the call to the subroutine, the CNC will apply the default search criterion (see the programming manual).

1158   There is no subroutine associated with G.

DETECTION	During execution.
CAUSE	A user subroutine (G500-G599) or OEM subroutine has been programmed (G180-G189/G380-G399) and the subroutine does not exist.
SOLUTION	Check that the G function is correct or define the associated subroutine.

1159   'Name of the local subroutine too long'

DETECTION   During execution.
CAUSE   The subroutine name has too many characters. The name of a subroutine may have a maximum of 63 characters.
SOLUTION   Check the program.

1160   'Local subroutine not found'

DETECTION   During execution.
CAUSE   The CNC cannot find the local subroutine. The possible causes are:
- The local subroutine is not defined at the beginning of the program.
- The program has activated a local subroutine as modal and then it executed a user subroutine (for example, G500 to G599) with motion blocks. The user subroutine cannot find the local subroutine because it belongs to another program.

SOLUTION   The possible solutions are:
- 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 must be defined at the beginning of the program.
- Define the local subroutine as global.

1161   '$ control blocks open'

DETECTION   During execution.
CAUSE   The CNC has found a "$" control block that does not have its corresponding closing instruction.
SOLUTION   Check the program.

1162   'M17/M29/#RET not expected'

DETECTION   During execution.
CAUSE   The CNC has detected an M17, M29 function or #RET 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 end with M17, M29 or #RET.

1163   'M30/M02 not expected'

DETECTION   During execution.
CAUSE   The CNC has detected an M02 or M30 function as end of program.
SOLUTION   Check that all the local and global subroutines end with M17, M29 or #RET.

1164   'Unknown term in mathematical expression'

DETECTION   During execution.
CAUSE   The mathematical expression is wrong.
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   The mathematical expression contains a square root (SQRT) of a negative number.
SOLUTION   Check the program.

1167   'Logarithm of a negative number or zero'

DETECTION   During execution.
CAUSE   The mathematical expression contains a logarithm (LOG/LN) of a negative number or zero.
SOLUTION   Check the program.

1168   'Variable index out of range'

DETECTION   During execution.

CAUSE   One of the indexes defined in 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 to G.LUP7, G.LUPACT and MTB.P.

1169   The statement #GET IPOOFFSET must be programmed alone in the block.

DETECTION   During execution.

CAUSE   The instruction has not been programmed alone in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added.

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

DETECTION   During execution.

CAUSE   The instruction has not been programmed alone in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added.

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

DETECTION   During execution.

CAUSE   The instruction has not been programmed alone in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added. The only exception is the #AXIS instruction that must be programmed in the same block as function G201.

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

DETECTION   During execution.

CAUSE   The CNC has tried to execute an instruction that is incompatible with tool radius compensation.

SOLUTION   Cancel tool radius compensation to execute the instruction.

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

DETECTION   During execution.

CAUSE   The instruction has not been programmed alone in the block.

SOLUTION   Check the program.

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

DETECTION   During execution.

CAUSE   The CNC has tried to activate a second axis coupling (slaving) without deactivating the first one.

SOLUTION   Check the program. Cancel the first coupling (slaving) before activating the second one. To have both couplings, cancel the first one and activate both with a single #LINK instruction.

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

DETECTION   During execution.

CAUSE   No axis coupling (slaving) has been defined in the #LINK instruction.

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:
- The master axis of the coupling does not exist or is not available in the channel.
- The CNC has tried 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:
  • The slave axis of the coupling does not exist or is not available in the channel.
  • The CNC has tried 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   The slave axis of the coupling is one of the main three axes.

SOLUTION   Check the program.

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

DETECTION   During execution.

CAUSE   The master and slave axes of the coupling are not of the same type, linear or rotary.

SOLUTION   Both axes of the coupling must be of the same type (parameter AXISTYPE).

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

DETECTION   During execution.

CAUSE   The master and slave axes of the coupling are rotary, but they are not of the same type, linearlike or module.

SOLUTION   Both axes of the coupling must be of the same type (parameter AXISMODE).

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

DETECTION   During execution.

CAUSE   The slave axis of the coupling is active in additive manual mode (G201).

SOLUTION   Cancel the additive manual mode in order to be able to couple the axis.

1182   **'#LINK: too many couplings programmed'**

DETECTION   During execution.

CAUSE   Too many couplings have been defined.

SOLUTION   The maximum number of couplings that may be active in a channel is equal to the number of axes of the channel minus three.

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

DETECTION   During execution.

CAUSE   The instruction has not been programmed alone in the block.

SOLUTION   Check the program.

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

DETECTION   During execution.

CAUSE   The master and slave axes of the coupling are the same axis.

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 axis is the slave of several masters.

SOLUTION   An axis can only be the slave of one master.

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

DETECTION   During execution.

CAUSE   An axis cannot be the slave in a coupling and the master in another.

SOLUTION   An axis cannot be the master in a coupling and the slave in another.

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 program.

1189   '#MPG: too many parameters'

DETECTION	During execution.
CAUSE	The instruction has too many parameters.
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	The #MPG instruction tries to define a negative or zero handwheel resolution.
SOLUTION	The movement distance per handwheel pulse must be positive and other than zero.

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

DETECTION	During execution.
CAUSE	The #INCJOG instruction tries to define a negative or zero 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	The #INCJOG instruction tries to define a negative or zero feedrate.
SOLUTION	The feedrate of the axis 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 programmed feedrate is too high.
SOLUTION	Check the program.

1194   '#INCJOG: too many parameters'

DETECTION	During execution.
CAUSE	The instruction has too many parameters.
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 too many parameters.
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	The #CONTJOG instruction tries to define a negative or zero 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	The lower axis travel limit for jogging has a positive value.
SOLUTION	The lower axis travel limit for jogging must be negative or zero.

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

DETECTION	During execution.
CAUSE	The value of the lower travel limit is too low.
SOLUTION	Check the program.

1199   '#SET OFFSET: negative upper offset'

DETECTION	During execution.
CAUSE	The upper axis travel limit for jogging has a negative value.
SOLUTION	The upper axis travel limit for jogging must be positive or zero.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

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

DETECTION   During execution.
CAUSE   The value of the upper travel limit is too high.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The values of the lower axis travel limits for jogging are zero.
SOLUTION   Check the program.

1202   **#SET IPOPOS: Too many parameters.**

DETECTION   During execution.
CAUSE   The instruction has too many parameters.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

1204   **'Nonexistent instruction or programmed wrong'**

DETECTION   During execution.
CAUSE   The instruction does not exist or is programmed wrong.
SOLUTION   Check the program.

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   "," missing in the instruction or function.
SOLUTION   Check the program.

1209   **'Axis index out of range'**

DETECTION   During execution.
CAUSE   The possible causes are:
   • In instructions #CALL AX/#SET AX the position for some axis is wrong; the position is taken, the position exceeds the maximum admitted or there is no room for the axis.
   • The axis name has been programmed with the wrong wild character.
SOLUTION   The possible solutions are:
   • The instruction can place the axes in any unoccupied position between 1 and a number equal to the total number of axes plus spindles of the system.
   • 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 more than once in the instruction.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   Two axes try to occupy the same position in the channel.
SOLUTION   Check the program.

1212   **'#CALL AX/#SET AX: repeated axis number.**

DETECTION   During execution.
CAUSE   Two axes try to occupy the same position in the channel.
SOLUTION   Check the program.

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   Too many axes programmed; the number of axes exceeds the number of axes of the system.
SOLUTION   Check the configuration of the axes of a channel The number of axes of a channel cannot exceed the number of axes of the system.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The name of one of the axes is being used by a C axis.
SOLUTION   Check the program.

1217   '#CALL AX: index being used'

DETECTION   During execution.
CAUSE   One of the positions is occupied by another axis.
SOLUTION   Check the configuration of the axes defined for the channel, two axes cannot be in the same position. An axis can use any free 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   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

1219   '"," or "]" expected'

DETECTION   During execution.
CAUSE   Syntax error in the instruction.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   An axis in additive manual mode G201 cannot be eliminated from the channel.
SOLUTION   Use function G202 to cancel the additive manual mode of the axis in order remove the axis from the channel.

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

DETECTION   During execution.
CAUSE   The instruction has not been programmed alone in the block.
SOLUTION   Check the program.

1222   '#COMMENT END not expected'

DETECTION   During execution.
CAUSE   There is an end-of-comment block (#COMMENT END), but the open-comment block (#COMMENT BEGIN) is missing.
SOLUTION   Check the program.

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

DETECTION   During execution.
CAUSE   The CNC cannot find the end of the program. It may be because there is an open-comment block (#COMMENT BEGIN), but the end-of-comment block (#COMMENT END) is missing.
SOLUTION   Check the program.

1224 **'Operator unknown or missing'**

DETECTION	During execution.
CAUSE	No assignment operator has been programmed after the variable or parameter.
SOLUTION	The valid assignment operators are "=", "+=", "-=", "*=", "/=".

1225 **'Division by zero'**

DETECTION	During execution.
CAUSE	One of the programmed operations is dividing by zero.
SOLUTION	Check the program. 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 missing in the plane to compensate the location code (shape) of lathe tool.**

DETECTION	During execution.
CAUSE	None of the axes of the plane has been defined as face axis.
SOLUTION	Define one of the axes of the main plane as face axis (parameter FACEAXIS).

1227 **Side axis missing in the plane to compensate the location code (shape) of lathe tool.**

DETECTION	During execution.
CAUSE	None of the axes of the plane has been defined as longitudinal axis.
SOLUTION	Define one of the axes of the main plane as longitudinal axis (parameter LONGAXIS).

1229 **Too many conditional instruction nesting levels.**

DETECTION	During execution.
CAUSE	Too many $IF, $SWITCH, $FOR, $WHILE and $DO instruction nestings.
SOLUTION	Check the program. The maximum number of nesting levels is 20.

1233 **'The programmed zero offset exceeds the data range'**

DETECTION	During execution.
CAUSE	The defined zero offset exceeds the maximum admitted.
SOLUTION	Check the program.

1236 **'Macro name too long'**

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

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

DETECTION	During execution.
CAUSE	In the text for replacing a macro, a macro has been included that does not begin with the "\" character.
SOLUTION	The replacement text of the macro must be between quote marks and may include other macros which must be delimited by the \" characters; for example "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 number of characters allowed is 140.

1239 **'Too many macros'**

DETECTION	During execution.
CAUSE	Too many macros at the CNC.
SOLUTION	The maximum is 50 macros. The table of macros may be erased with the instruction #INIT MACROTAB.

1240 **'Nonexistent axis'**

DETECTION	During execution.
CAUSE	The macro is not defined in the program.
SOLUTION	Define the macro before using it. The macro may be defined in a program.
	The CNC stores in a table the macros defined in a program or in MDI/MDA mode so they are available for all the programs executed afterwards. The CNC initializes the table of macros on start-up or with the instruction #INIT MACROTAB.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

1241   **'The replacement text of the macro is missing'**

DETECTION	During execution.
CAUSE	The replacement text associated with the macro is 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 quote marks.

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

DETECTION	During execution.
CAUSE	When working at constant surface speed (G96), the CNC has limited the spindle speed because the face axis is very close to the center of rotation.
SOLUTION	Increase the speed limit (G192) or accept the limitation.

1245   **'G96: no face axis (FACEAXIS) has been defined in the active plane'**

DETECTION	During execution.
CAUSE	None of the axes of the plane has been defined as face axis.
SOLUTION	Define one of the axes of the main plane as face axis (parameter FACEAXIS).

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 (feed per spindle revolution).
SOLUTION	Activate the feedrate as a function of time (G94)

1247   **'Threading is not possible while G96 is active'**

DETECTION	During execution.
CAUSE	The CNC has tried to execute a rigid tapping (G63) while function G96 is active (constant surface speed).
SOLUTION	Activate the constant turning speed mode (G97).

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

DETECTION	During execution.
CAUSE	The CNC does not admit programming functions G63 (rigid tapping) and G192 (turning speed limitation) in the same block.
SOLUTION	Program both instructions in different blocks.

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

DETECTION	During execution.
CAUSE	The CNC has tried to change the spindle gear (G112) while function G96 was 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	The CNC has tried to access the jog mode while function G96 was active.
SOLUTION	Cancel function G96 to access the jog mode.

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

DETECTION	During execution.
CAUSE	The CNC has tried to eliminate the face axis from the configuration of the channel (#FREE AX) while function G96 was active.
SOLUTION	Cancel function G96 to eliminate the face axis from the configuration of the channel.

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

DETECTION	During execution.
CAUSE	The CNC does not admit programming functions M19 (spindle orientation) and G192 (turning speed limitation) in the same block.
SOLUTION	Program both instructions in different blocks.

1255   **'Negative acceleration percentage'**

DETECTION	During execution.
CAUSE	In function G130, the programmed percentage of acceleration is negative.
SOLUTION	The percentage of acceleration must be equal to or greater than zero.

1256   'Acceleration percentage out of range'

DETECTION	During execution.
CAUSE	In function G130, the programmed percentage of acceleration is too high.
SOLUTION	Check the program.

1257   'The leadscrew pitch has been programmed twice'

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

1258   'Leadscrew pitch equal to zero'

DETECTION	During execution.
CAUSE	In function G33, the thread pitch is zero.
SOLUTION	Check the program. Program the thread pitch with parameters I J K.

1259   'Leadscrew pitch out of range'

DETECTION	During execution.
CAUSE	In function G33, the programmed thread pitch is too large.
SOLUTION	Check the program.

1261   'Unknown kinematics type'

DETECTION	During execution.
CAUSE	The CNC has tried to activate an undefined kinematics in machine parameters. The CNC has tried to activate function #RTCP or #TLC, or a coordinate transformation (#CS/ACS) in mode 6 without having a kinematics active.
SOLUTION	Check syntax. First activate the kinematics and then the desired function.

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

DETECTION	During execution.
CAUSE	The channel does not have enough axes to activate function RTCP, TLC or the coordinate transformation. The number of axes required will depend on the kinematics to be activated.
SOLUTION	Correct the configuration of the axes of the channel (instruction: #SET AX) in order to be able to activate the kinematics.

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

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

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

DETECTION	During execution.
CAUSE	The CNC does not allow executing the programmed function if coordinate transformation is active. Some incompatible functions are G74, G198, G199, #LINK, probing cycles, etc.
SOLUTION	Cancel coordinate transformation to execute the rest of the functions.

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

DETECTION	During execution.
CAUSE	The CNC does not allow executing the programmed function if function RTCP or TLC is active. Some incompatible functions are G74, G198, G199, #KIN ID. LINK.
SOLUTION	Cancel the RTCP or TLC function to execute the rest of the functions.

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

DETECTION	During execution.
CAUSE	The CNC has tried to modify the TLC function while it was active.
SOLUTION	The CNC does not allow modifying the TLC function while it is active. To modify the TLC function, first deactivate it and then activated again.

1267   'The RTCP functionality is canceled with the instruction #RTCP OFF'

DETECTION	During execution.
CAUSE	While RTCP is active, a #RTCP instruction other than #RTCP OFF has been programmed.
SOLUTION	To cancel the RTCP function, program #RTCP OFF. To change the values of the RTCP, it must be canceled first.

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

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The programmed angle is wrong.
SOLUTION	Program an angle within ±360º.

1270   'The coordinate transformation cannot be calculated'

DETECTION	During execution.
CAUSE	The CNC has not been able to solve a transformation from part-coordinates to machine coordinates or vice versa.
SOLUTION	Cancel the transformation, change the position of the axes and activate the transformation again.

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

DETECTION	During execution.
CAUSE	The CNC has tried to modify the configuration of the axes of a kinematics while it was active.
SOLUTION	Cancel the kinematics before modifying the configuration of the axes of the channel.

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

DETECTION	During execution.
CAUSE	The CNC has tried to modify the configuration of the axes of an inclined plane while it was active.
SOLUTION	Cancel the inclined plane transformation to modify the configuration of the axes of the channel.

1275   'Position calculated by the inverted kinematics transformation out of range'

DETECTION	During execution.
CAUSE	The CNC has tried to do an unsolved transformation from machine coordinates to part coordinates.
SOLUTION	Cancel the transformation, change the position to be accessed and activate the transformation again. Check the transformation if it is a user transformation.

1277   'The resulting zero offset exceeds the data range'

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

1278   'G131/G133: wrong value'

DETECTION	During execution.
CAUSE	In function G131 or G133, the programmed percentage of acceleration or jerk is not valid.
SOLUTION	Program a percentage of acceleration or jerk that is 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 program.

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

DETECTION	During execution.
CAUSE	The number of data identifiers (%D or %d) appearing in the #MSG, #ERROR or #WARNING instruction does not match the number of parameters to be displayed.
SOLUTION	Check the program.

1282 **'Message too long'**

DETECTION	During execution.
CAUSE	The text message of the instruction #MSG, #ERROR or #WARNING is too long.
SOLUTION	It may have up to 69 characters, including the characters that replace the data identifiers.

1283 **'Too many format indicators'**

DETECTION	During execution.
CAUSE	There are more than 5 data identifiers (%D ó %d) in the instruction #MSG, #ERROR, #WARNING.
SOLUTION	Reduce the number of data identifiers.

1284 **'Arithmetic expression expected'**

DETECTION	During execution.
CAUSE	There are data identifiers (%D ó %d) in the instruction #MSG, #ERROR, #WARNING, but the parameters to be displayed are missing.
SOLUTION	Check the program.

1285 **'Tool radius written twice'**

DETECTION	During execution.
CAUSE	The tool radius has been programmed more than once in the same block.
SOLUTION	Program the tool radius only once in the block.

1286 **'Tool length written twice'**

DETECTION	During execution.
CAUSE	The tool length has been programmed more than once in the same block.
SOLUTION	Program the tool length only once in the block.

1287 **'"[" expected'**

DETECTION	During execution.
CAUSE	The opening bracket "]" is missing in the programmed expression or instruction.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the program.

1289 **The command #MSG message is too long.**

DETECTION	During execution.
CAUSE	The message programmed in the #MSG/#WARNING/#ERROR instruction is too long.
SOLUTION	The maximum length of text in these instructions is 70 characters.

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

DETECTION	During execution.
CAUSE	The possible 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.
	• The values programmed for the center of the circular interpolation, being G264 active, are wrong.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	There are too many S functions in the same block.
SOLUTION	The maximum number of S functions allowed in the same block is 4.

1292 **'M function programmed twice'**

DETECTION   During execution.
CAUSE   The same M function has been programmed more than once in the block.
SOLUTION   Program both functions in different blocks.

1293 **'H function programmed twice'**

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

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

DETECTION   During execution.
CAUSE   The tool length transformation exceeds the valid numeric format.
SOLUTION   Modify the values of the length transformation or those of the tool.

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

DETECTION   During execution.
CAUSE   Wrong character in the name of a label, subroutine or variable.
SOLUTION   Check the program.

1303 **'Variable name too long'**

DETECTION   During execution.
CAUSE   Too many characters for the name of the variable.
SOLUTION   The maximum number of characters allowed is 13.

1304 **'Wrong spindle speed'**

DETECTION   During execution.
CAUSE   The programmed speed is too low.
SOLUTION   Check the program.

1305 **'Programming not allowed while #MCS is active'**

DETECTION   During execution.
CAUSE   The CNC does not allow executing the programmed function if the MCS function is active. Some incompatible functions are:
- Zero offsets (G54-G59, G159, G92, G158, G53) on/off
- Measuring offset (G101, G102) 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   The CNC has tried to change the kinematics while tool radius compensation was active.
SOLUTION   Cancel the compensation to define the new work plane.

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

DETECTION   During execution.
CAUSE   The slave axis of the coupling is part of the active kinematics.
SOLUTION   Cancel the kinematics to be able to couple the axis. The axis involved in the active kinematics can be master of a coupling.

1309 **'File name expected'**

DETECTION	During execution.
CAUSE	No program has been selected for execution.
SOLUTION	Select the program to be executed.

1310 **'Program line too long'**

DETECTION	During execution.
CAUSE	Too many characters in the #EXBLK instruction.
SOLUTION	The maximum number of characters allowed is 128.

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

DETECTION	During execution.
CAUSE	The G102 function tries to exclude the measurement offset of an axis that has no measurement offset included.
SOLUTION	The G102 function makes no sense for an axis without measuring offset.

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

DETECTION	During execution.
CAUSE	In these instructions, the number of the coordinate system is not valid.
SOLUTION	Program a value between 1 and 5.

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

DETECTION	During execution.
CAUSE	The instructions do not have parameters and no coordinate system has been stored. When programming one of these instructions without parameters, the CNC tries to activate the transformation stored last.
SOLUTION	Check the program. Define and save some coordinate system.

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

DETECTION	During execution.
CAUSE	One or more required parameters have not been programmed.
SOLUTION	Check the program. These instructions require programming the coordinate system number, the definition mode, the components of the translation vector and the rotation angles.

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

DETECTION	During execution.
CAUSE	The CNC has tried to change the parameters of an active coordinate system.
SOLUTION	Check the program. The CNC does not allow to change the parameters of an active coordinate system; it does allow changing the parameters of a coordinate system that has been already defined, but it is not active.

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

DETECTION	During execution.
CAUSE	The The CNC has exceeded the limit of coordinate system combinations.
SOLUTION	The allows combining different coordinate systems with each other to build new ones. The CNC allows combining 10 coordinate systems.

1320 **'Too many labels'**

DETECTION	During execution.
CAUSE	The program has too many block labels. The labels to identify a block may be of the following type
SOLUTION	The maximum number of labels for each type is 128. The labels may be represented by the letter N followed by the block number or by [name] type labels.

1321 **'Label name too long'**

DETECTION	During execution.
CAUSE	The name of the label has too many characters.
SOLUTION	The maximum number of characters allowed is 15.

1322 **'Label defined several times'**

DETECTION	During execution.
CAUSE	The block label is repeated (duplicate) in 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   Check the program.

1324   Undefined label in the same subroutine as the $GOTO, #RPT or #ABORT.

DETECTION   During execution.

CAUSE   The tag programmed in $GOTO, #RPT or #ABORT does not exist in the subroutine or program.

SOLUTION   Define the jump tag in the same program or subroutine as the instructions.

1325   'Block number defined several times'

DETECTION   During execution.

CAUSE   The block number "N" is repeated (duplicate) in the program.

SOLUTION   Do not repeat the block number.

1326   'Wrong value to be assigned to a variable'

DETECTION   During execution.

CAUSE   The value of the variable is too high.

SOLUTION   Check the program.

1327   Spindle position programmed twice.

DETECTION   During execution.

CAUSE   The spindle orientation speed (M19) has been programmed more than once in the block.

SOLUTION   Check the program. Program the positioning speed only once in the block.

1328   '$FOR instruction without $ENDFOR'

DETECTION   During execution.

CAUSE   A $FOR instruction has been programmed, but its $ENDFOR is missing.

SOLUTION   Check the program.

1330   'Mirror image programmed wrong'

DETECTION   During execution.

CAUSE   Function G14 (mirror image) has been programmed wrong.

SOLUTION   Check the program.

1331   '#TANGFEED RMIN: negative radius not allowed'

DETECTION   During execution.

CAUSE   The programmed radius is less than or equal to zero.

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 program.

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

DETECTION   During execution.

CAUSE   Being tool radius compensation active, the CNC has tried to change the axis configuration of the channel and the change affects the first two axes of the work plane.

SOLUTION   Cancel tool radius compensation in order to make the changes in the axis configuration of the channel that affect the work plane.

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

DETECTION   During execution.

CAUSE   An axis movement has been programmed in the same block as function G200.

SOLUTION   Check the program.

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 programmed axes may be a C axis (parameter CAXIS).

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 programmed axes must be a C axis (parameter CAXIS).

1339   'The selection has no effect'

DETECTION	During execution.
CAUSE	The programmed instruction has no effect because it is already active, the same instruction with the same parameters is already programmed in a previous block.
SOLUTION	Check the program.

1340   'The deselection has no effect'

DETECTION	During execution.
CAUSE	The CNC has tried to execute the #CAX OFF instruction, but there is no spindle working as C axis.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The CNC does not allow canceling the C axis while the RTCP or TLC function is active.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The CNC has tried to execute the #FACE OFF instruction, but there is no machining on the face of the part.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The CNC has tried to change the work plane (G17-G20) while lateral machining is active.
SOLUTION	Check the program.

1345   'G20: Axes programmed wrong'

DETECTION	During execution.
CAUSE	In function G20 (plane change), the first two axes of the plane (parameters 1 and 2) are wrong.
SOLUTION	The two axes must be different and they must be among the first three axes of the channel.

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

DETECTION	During execution.
CAUSE	The CNC has tried to execute the #CYL OFF instruction, but there is no machining on the side of the part.
SOLUTION	Check the program.

1348   '#CYL: wrong radius'

DETECTION	During execution.
CAUSE	The radius programmed in the #CYL instruction is negative or zero. If the radius is variable, it tries to go through the center of the cylinder generating a zero radius.
SOLUTION	Check the program. The radius must be positive and if it is a variable radius, it cannot go through the center of the 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. The CNC does not admit this option (parameter ALINGC).

SOLUTION   Position the axis in the positive portion with respect to the rotation center before activating the machining operation along the face axis.

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   The CNC has tried to define or delete a variable that is not a user variable.

SOLUTION   Check the program.

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

DETECTION   During execution.

CAUSE   The user variable already exists.

SOLUTION   Check the program.

1353   'Too many values to initialize the array'

DETECTION   During execution.

CAUSE   When initializing an user array variable, the CNC initializes more positions than it has.

SOLUTION   Check the program.

1354   'Error when reading the variable'

DETECTION   During execution.

CAUSE   The CNC cannot read the variable.

SOLUTION   Check the program.

1355   'The variable cannot be deleted'

DETECTION   During execution.

CAUSE   The CNC has tried to delete a variable from the system.

SOLUTION   The CNC can only delete user variables (prefixes P and S).

1356   'Variable or parameter expected'

DETECTION   During execution.

CAUSE $IF EXIST instruction programmed wrong.

SOLUTION   The $IF EXIST instruction only allows arithmetic parameters or variables.

1357   '#DELETE: wrong character'

DETECTION   During execution.

CAUSE   An invalid character has been detected in the block.

SOLUTION   Check the syntax of the block. The instruction must be programmed alone in the block or next to the block label. This instruction only admits user variables.

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

DETECTION   During execution.

CAUSE   The user variable does not exist.

SOLUTION   Check the program.

1359   The commands #VAR/#ENDVAR/#DELETE do not allow this type of variable.

DETECTION   During execution.

CAUSE   An attempt was made to delete a variable from the CNC or an attempt was made to create a new variable with one of the prefixes predefined by the CNC.

SOLUTION   Predefined CNC variables cannot be deleted. These commands are only applicable to variables with prefix P and S.

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

DETECTION	During execution.
CAUSE	The CNC has tried to execute a G33/G63/G95/G96/G97 function while the C axis was active.
SOLUTION	Cancel the C axis in order to be able to execute the function.

1362 **'Wrong array dimension.**

DETECTION	During execution.
CAUSE	It is an array variable, but the number of programmed arrays is wrong.
SOLUTION	Check the syntax of the variable.

1363 **'Wrong declaration of array variables'**

DETECTION	During execution.
CAUSE	User variables that are array variables must be declared between #VAR and #ENDVAR instructions.
SOLUTION	Check the program.

1364 **'Too many array variable indexes'**

DETECTION	During execution.
CAUSE	The user variable is a multi-dimensional array with more than 4 dimensions.
SOLUTION	Check the program.

1365 **'Negative spindle speed not allowed'**

DETECTION	During execution.
CAUSE	Negative spindle speed.
SOLUTION	The spindle speed must be positive; a negative value is only allowed when programming G63 in the same block.

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	Check the program.

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

DETECTION	During execution.
CAUSE	The circular interpolation is defined with the radius and the center.
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 program.

1370 **'#HSC: double programming'**

DETECTION	During execution.
CAUSE	The HSC activation and cancellation have been programmed in the same block.
SOLUTION	Check the syntax of the instructions in the programming manual.

1371 **'#HSC: invalid mode.'**

DETECTION	During execution.
CAUSE	The CNC has tried to activate the HSC mode with an unknown parameter or has tried to change the work mode without activating it first.
SOLUTION	Check the program.

1373 **'#HSC: wrong parameter'**

DETECTION	During execution.
CAUSE	The value of parameter FAST or parameter CORNER of the #HSC instruction is wrong.
SOLUTION	Check the program.

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   In the #CAX instruction, the name of the C axis is missing and the default name (parameter CAXIS) is not indicated in the machine parameters.
SOLUTION   In the #CAX instruction, indicate the name that will identify the "C" axis.

1377   'Parameter written with wrong index'

DETECTION   During execution.
CAUSE   The arithmetic parameter does not exist or is write-protected.
SOLUTION   Check the program. Check the valid arithmetic parameter range in the machine parameters.

Machine parameters.	Valid range.
MINLOCP - MAXLOCP	Local arithmetic parameters.
MINGLBP - MAXGLBP	Global arithmetic parameters.
MINCOMP - MAXCOMP	Common arithmetic parameters.

The write-protected global parameters are those defined by machine parameters ROPARMIN - ROPARMAX.

1378   'Parameter read with wrong index'

DETECTION   During execution.
CAUSE   The arithmetic parameter does not exist; it is not within the range admitted by the machine parameters.
SOLUTION   Check the program. Check the valid arithmetic parameter range in the machine parameters.

Machine parameters.	Valid range.
MINLOCP - MAXLOCP	Local arithmetic parameters.
MINGLBP - MAXGLBP	Global arithmetic parameters.
MINCOMP - MAXCOMP	Common arithmetic parameters.

1380   'Canned cycle programmed wrong'

DETECTION   During execution.
CAUSE   In the block defining a canned cycle, nothing must be programmed after the cycle parameters.
SOLUTION   Check the program.

1381   'Nonexistent canned cycle'

DETECTION   During execution.
CAUSE   The programmed canned cycle does not exist.
SOLUTION   Check the program.

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 programming manual for the parameters required and allowed by each canned cycle.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·54·

1383 **'Mandatory parameter not programmed in canned cycle'**

DETECTION	During execution.
CAUSE	A parameter required in the canned cycle has not been programmed.
SOLUTION	Check the programming manual for the parameters required and allowed by each canned cycle.

1384 **'M function not allowed with motion'**

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	Program the M function in a block without movement or define the M function in the machine parameters with execution after movement.

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

DETECTION	During execution.
CAUSE	The CNC has tried to write the "V.G.TOL" variable in the same block where a tool change or tool offset change is programmed.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The CNC has tried to write the "V.G.TOFL.xn" 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	There are too many M functions in the same block.
SOLUTION	From version V2.00, the CNC admits up to 14 M functions per block; in earlier versions, the limit was 7 functions per block.

1388 **'No more H functions allowed'**

DETECTION	During execution.
CAUSE	There are too many H functions in the same block.
SOLUTION	The maximum number of H functions allowed in the same block is 7.

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

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

1390 **'Incompatible G functions (G98/G99)'**

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

1392 **'Parameter programmed twice'**

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

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

DETECTION	During block search.
CAUSE	After a tool inspection, the active tool offset D does not match the tool offset D programmed before interrupting the execution. 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, the CNC 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	A G180 or G189 function has been programmed, but the associated subroutine does not exist. A G74 function has been programmed alone in the block, but the associated subroutine does not exist.
SOLUTION	Together with function G74, program the axes to be homed or define the associated subroutine (parameter REFPSUB). For functions G180 through G189, define the associated subroutine (parameter OEMSUB).

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

DETECTION	During execution.
CAUSE	If function G74 has an associated subroutine, it must be programmed alone in the block or together with the block label.
SOLUTION	Check the program.

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	The hirth axis only admits multiples of its pitch.

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

DETECTION	During execution.
CAUSE	The CNC has tried to activate (G171) or deactivate (G170) as a hirth axis one that is not (parameter HIRTH).
SOLUTION	Check the program. 'The axis cannot be activated as a hirth axis.

1399   'The axis cannot be activated as Hirth'

DETECTION	During execution.
CAUSE	The possible causes are: • The hirth axis is one of the axes of the plane and tool radius compensation and/or collision detection is active. • The hirth axis is part of the active coordinate transformation. • The hirth axis is part of the active kinematics and also an #RTCP, #TLC or #TOOL ORI instruction is active.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The CNC has tried to make a tool change while function RTCP was active.
SOLUTION	Cancel the RTCP function to make the tool change.

1401   '#TLC ON not admitted without unselecting first'

DETECTION	During execution.
CAUSE	The CNC has tried to activate the TLC function and it is already active.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The CNC has tried to couple two hirth axes, but one of them is not hirth or they both have different pitches.
SOLUTION	In order to include a hirth axis in a coupling, both axes must be hirth (parameter HIRTH) and have the same pitch.

1403   '#LINK: a coupling cannot be defined if a Hirth axis is deactivated'

DETECTION	During execution.
CAUSE	The CNC has tried to couple two hirth axes, but one of them is deactivated.
SOLUTION	In order to couple two hirth axes, both of them must be active (G171).

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

DETECTION	During execution.
CAUSE	The CNC has tried to make a spindle gear change using an M41 through M44 function, but the gear does not exist.
SOLUTION	The number of gears available at the spindle is determined in the machine parameters (parameter NPARSETS). The CNC will accept the M41 through M44 functions of the existing spindle gears.

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

DETECTION	During execution.
CAUSE	The spindle has an automatic gear changer (parameter AUTOGEAR) and a speed has been programmed that is higher than any of the existing gears for that spindle.
SOLUTION	Program a lower spindle speed that may be reached with one of the gears existing for that spindle. Check that the maximum speed defined in each gear is correct.

1406 **'#CALL: does not admit parameter programming'**

DETECTION	During execution.
CAUSE	The #CALL instructiont does not allow to program parameters.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	When executing a 2D or 3D pocket, the CNC could not decode some of the data. This may happen when the pocket data has been edited manually and a numeric value has been replaced with a variable, the bracket at the end of the instructions #DATAP2D, #DATAP3D has been eliminated, etc.
SOLUTION	Edit the pocket again with the cycle editor.

1408 **G63/G74/M19/CAX: Programming not allowed without encoder on the spindle.**

DETECTION	During execution.
CAUSE	The CNC cannot execute the programmed function or instruction 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	The CNC has tried to execute a tool change and the subroutine associated with the T function (parameter TOOLSUB) contains another tool change.
SOLUTION	A T function cannot be programmed inside the subroutine associated with the tool change

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 depends on the CNC model; 200 blocks (CNC 6065) or 40 (CNC 8060).

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 command "S.POS".

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

DETECTION	During execution.
CAUSE	The #PARK instruction only allows parking one axis.
SOLUTION	Program a #PARK block for each axis to be parked.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

Ref. 2209

·57·

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.
SOLUTION	The path of a program or subroutine may have a maximum of 120 characters. Move the program or subroutine to another directory to reduce the number of characters of the path.

1418   'It is not possible to park axes of the active kinematics or transformation'

DETECTION	During execution.
CAUSE	The CNC has tried to park an axis that is part the active kinematics or transforms #CS, #ACS, #ANGAX, #TANGCTRL.
SOLUTION	Check the program. To park that axis, cancel the active kinematics or transformation.

1419   'Slaved axes (Gantry or Tandem) cannot be parked.'

DETECTION	During execution.
CAUSE	The CNC has tried to park an axis that is part of a gantry, tandem axis or of an active coupling (#LINK).
SOLUTION	Check the program. To park that axis, cancel the coupling. The axes belonging to a gantry or tandem pair cannot be parked.

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

DETECTION	During execution.
CAUSE	Some $IF, $FOR, etc. does not have its corresponding closing instruction.
SOLUTION	Check the program.

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 coordinate 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 coordinate 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 as method for aligning the plane is wrong.
SOLUTION	Check the program.

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.

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	During execution.
CAUSE	The CNC generated the pocket with a tool radius other than the current one.
SOLUTION	Generate the pocket again.

1427   'Axis programmed wrong'

DETECTION	During execution.
CAUSE	Syntax error when programming the instruction or G function; some axis is repeated in a movement, the home searching order for the axes is missing in the G74 function or the order of the axes in the system is missing in the G20 function.
SOLUTION	Check the program.

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 starting point of the cycle are defined after the G function that defines the cycle. The cycle considers the coordinates of the starting point as its own 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.
SOLUTION	The maximum number of subroutines that may be executed in the same block is 5. Program the subroutines in different blocks or use subroutine nesting as necessary.

1430   **'Numeric format exceeded'**

DETECTION	During execution.
CAUSE	The value assigned to a data, variable or parameter 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	In instruction contains the slave axis of an active coupling or of Gantry pair.
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   **The configuration of a parked axis cannot be modified.**

DETECTION	During execution.
CAUSE	The CNC has tried to include (#CALL / #SET) a parked axis between the first three axes of the channel.
SOLUTION	A parked axis cannot be part of the first of three axes of the channel; unpark the axis (#UNPARK).

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

DETECTION	During execution.
CAUSE	The CNC has tried to include the master axis of an active coupling or that of a Gantry pair in the channel configuration (#CALL or #SET). When including the master axis, the CNC also includes the slave axis which can never occupy one of the main three positions of the channel. The CNC issues an error because there is no free position other than the main three 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 CNC has tried to include the master axis of an active coupling or that of a Gantry pair in the channel configuration (#CALL or #SET). When including the master axis, the CNC also includes the slave axis which can never occupy one of the main three positions of the channel. The CNC issues an error 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 stop block of the block search is missing.
SOLUTION	Once the block search option has been selected, the softkey menu shows the option to select the stop block. Select the block where the block search will end.

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

DETECTION   During execution.

CAUSE   The CNC has tried to activate a coordinate transformation (#TLC, #RTCP, #TOOL ORI, #CS or #ACS), but one of the axes involved in the transformation is a Hirth axis.

SOLUTION   A hirth axis must not be part of the coordinate transformation. An axis involved in the transformation must not be a Hirth axis (G170).

1440   'Wrong tool or edge'

DETECTION   During execution.

CAUSE   The tool and/or offset indicated in the variable does not exist.

SOLUTION   Check the program. Check the existing tools and tool offsets.

1441   'The kinematics has not been activated'

DETECTION   On CNC power-up or when validating the machine parameters.

CAUSE   The kinematics that the CNC applies by default (parameter KINID) is unknown or the axes required for the kinematics are not the right ones.

SOLUTION   Check that the type of kinematics is correct and that the axes involved in it are properly defined and belong to the channel. 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   After a reset of the CNC or when starting the execution of a part-program.

CAUSE   The CNC has canceled a kinematics, either because it is an unknown kinematics or because the axes required for that kinematics are not the right ones.

SOLUTION   Check that the type of kinematics is correct and that the axes involved in it are properly defined and belong to the channel. 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   After a reset of the CNC, when starting the execution of a part-program or during home search.

CAUSE   The axes used in the inclined plane are not the right ones. The CNC has tried to carry out a home search while function #CS/ACS was active.

SOLUTION   To make an inclined plane, the first three axes of the channel must be defined, linear, unparked and not slaves of a coupling or a gantry pair. Deactivate the #CS/ACS functions before searching home.

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.

SOLUTION   The first three axes of the transformation or kinematics must be linear (parameter AXISTYPE).

1445   'Wrong parameter value'

DETECTION   During execution.

CAUSE   A parameter has an invalid value in an instruction or a fixed cycle.

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.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·60·

1447   'Software option not allowed'

DETECTION   During execution.

CAUSE   The possible causes are:
- The CNC does not have the software option required to execute the programmed command.
- A drive included on the list of dual-use products (identified with the text -MDU) has tried to exceed the 550 Hz to achieve G00FEED.

SOLUTION   The possible solutions are:
- In diagnosis mode, it is possible to check the software options offered by the CNC.
- The drives included on the list of dual-use products (identified with the text -MDU) do not exceed 550 Hz.

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 perpendicular to the plane, as it may be the case of angular spindles.
- Placing the tool perpendicular to the active plane implies exceeding the limits.

SOLUTION   Define another plane or position the tool in another point. Use another spindle, if possible.

1449   '#PATH instruction programmed wrong'

DETECTION   During execution.

CAUSE   The syntax of the #PATH instruction is wrong.

SOLUTION   Check the program.

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

DETECTION   During execution.

CAUSE   The value of the V.G.TOOLORIF1 or V.G.TOOLORIF2 variable is wrong because the tool cannot be positioned perpendicular to the active plane. The possible causes are:
- The spindle type does not allow positioning the tool perpendicular to the plane, as it may be the case of angular spindles.
- Placing the tool perpendicular to the active plane implies exceeding the limits.

SOLUTION   Define another plane or position the tool in another point. Use another spindle, if possible.

1451   'Nonexistent variable for the type of axis'

DETECTION   During execution.

CAUSE   The variable does not exit for the programmed 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 programmed 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 axis name is defined by 1 or 2 characters. The first character must be one of the letters X - Y - Z - U - V - W - A - B - C. The second character is optional and will be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range X, X1…X9,...C, C1…C9.

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 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 required parameter is missing
SOLUTION	Check the program.

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, the starting parameter of the interpolation (SP) must be smaller than the final parameter (EP) and the curvature radius must be greater than zero.

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

DETECTION	During execution.
CAUSE	There are more than three axes in the polynomial.
SOLUTION	Only three axes can be involved in the polynomial interpolation.

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 making the starting point of the polynomial 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 program.

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 with a G9, 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.

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	The CNC has tried to activate one of the functions while the other one was active.
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 V.A.POSLIMIT.xn and V.A.NEGLIMIT.xn 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	One of the two coordinates of the Polar origin is missing in the G30 function.
SOLUTION	Check the program. The coordinates of the Polar origin must be programmed in both main axes.

1469 **'Negative or zero radius not allowed'**

DETECTION	During execution.
CAUSE	In a movement in Polar coordinates, the Polar radius is negative or zero.
SOLUTION	The Polar radius must always be greater than zero. When programming with incremental coordinates, 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 unidirectional rotary axis (parameter UNIDIR) 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	One of the two coordinates of the center of rotation is missing in the G73 function.
SOLUTION	Check the program. 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 rotation angle has not been programmed in function G73.
SOLUTION	Check the program. 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 CNC has tried to execute the #POLY instruction while a 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 CNC generated the pocket with a tool radius other than the current one.
SOLUTION	Generate the pocket again.

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

DETECTION	During execution.
CAUSE	The CNC generated the pocket with a tool's cutting length other than the current one.
SOLUTION	Generate the pocket again.

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

DETECTION	During execution.
CAUSE	The CNC generated the pocket with a tool entry angle other than the current one.
SOLUTION	Generate the pocket again.

1479 **'G74: A subroutine has not been associated'**

DETECTION	During execution.
CAUSE	A G74 function has been programmed alone in the block, but the associated subroutine does not exist.
SOLUTION	Together with function G74, program the axes to be homed or define the associated subroutine (parameter REFPSUB).

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

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1481 **'Wrong channel number'**

DETECTION	During execution.
CAUSE	Wrong channel number 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 syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1484 **'Signal number out of range'**

DETECTION	During execution.
CAUSE	The signal number programmed in the instruction #WAIT, #MEET or #SIGNAL is wrong.
SOLUTION	Check the program.

1485 **'#WAIT/#MEET not effective'**

DETECTION	During execution.
CAUSE	The #WAIT or #MEET instruction is not going to generate any wait because the synchronization mark is programmed for the same channel as the instructions.
SOLUTION	Check the program.

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

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added.

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

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1489 **'Axis name repeated in the resulting group'**

DETECTION	During execution.
CAUSE	The #RENAME instruction has tried to rename more than one axis with the same name.
SOLUTION	Rename the axes so there aren't two axes with same name in the channel.

1490 **'G63 requires programming M19 before'**

DETECTION	During execution.
CAUSE	The CNC has tried 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 the #SELECT PROBE instruction 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 CNC has tried to select a probe (#SELECT PROBE) or carry out a probing movement (G100), but there is no digital probe input associated with the probe.

SOLUTION   Associate a digital input with the probe (parameter PRBDI1 or PRBDI2).

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

DETECTION   During execution.

CAUSE   The CNC has tried to keep functions #SPLINE ON, G41, G42 and G136 active at the same time.

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   Restore the initial configuration of the first three axes of the channel.

1496 '#PROBE1: programming not allowed while #TOOL AX[-] is active'

DETECTION   During execution.

CAUSE   The CNC has tried 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   The CNC has tried to change a non-numerical variable using the operators "+=", "-=", "*=" amd "/=".

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 instruction. The CNC issues an error when exceeding the number of nesting levels of RPT instructions and subroutines, if any.

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   Using the #EXEC instruction, the CNC has tried to execute a program in a channel that is in error, is executing another program or is in jog mode and cannot switch into 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.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

1502 **'The variable requires programming an array index'**

DETECTION	During execution.
CAUSE	It is an array variable and the index has not been indicated.
SOLUTION	Check the list of variables in the manuals.

1503 **'The variable requires programming an axis'**

DETECTION	During execution.
CAUSE	It is an axis variable and the axis name has not been indicated.
SOLUTION	Check the list of variables in the manuals.

1504 **'The variable does not allow programming an array index'**

DETECTION	During execution.
CAUSE	The syntax of the variable is wrong.
SOLUTION	Check the list of variables in the manuals.

1505 **'The variable does not allow programming an axis'**

DETECTION	During execution.
CAUSE	The syntax of the variable is wrong.
SOLUTION	Check the list of variables in the manuals.

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	Check the program.

1507 **'The second label of the #RPT must be programmed alone in the block'**

DETECTION	During execution.
CAUSE	In a block repetitions (#RPT), the second label must be programmed alone in the block without any other type of data.
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:
	• One of the axes involved in function G201 is a C axis and is one of the first three axes of the channel.
	• One of the axes programmed in the #FACE instruction is in additive manual mode, G201.
SOLUTION	Cancel the C axis or the 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:
	• The CNC has tried to move an axis that does not exist or is not available in the channel.
	• The CNC has tried to execute an instruction that involves an axis that does not exist or is not available in the channel.
	• The CNC has tried to read or write a variable of an axis that does not exist or is not available in the channel.
SOLUTION	Check the program.

1511   'The tool is not in the magazine'

DETECTION	During execution.
CAUSE	The CNC has tried to use the V.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	The CNC has tried to use the V.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	The possible causes are: • 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 exit for the axis type (linear or rotary). • It is an axis variable, but it does not exist for the type of drive of the axis. • The combined operator (+= -= *= /=) is not admitted for that variable. • The value of the variable is not valid.
SOLUTION	Check the program.

1514   'A 2D-3D pocket cannot be executed while G72 is active'

DETECTION	During execution.
CAUSE	The CNC has tried to execute the pocket with an active scaling factor.
SOLUTION	Cancel the scaling factor.

1515   'One or several axes of the original configuration are not available'

DETECTION	During execution.
CAUSE	A channel has released an axis with a temporary change permission (parameter AXISEXCH) and cannot restore it with a reset or at the 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	Check the program.

1517   A zero value is not allowed.

DETECTION	During execution.
CAUSE	A null value has been programmed into a G instruction or function.
SOLUTION	Check the program.

1518   'NR requires programming a movement in the block'

DETECTION	During execution.
CAUSE	A repetition blocks (NR) has been programmed in a block 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	A repetition blocks (NR) has been programmed in a block that contains an M, T, D or H function.
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	A repetition blocks (NR) has been programmed in a block that contains the $GOTO instruction.
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 repetition of blocks (NR) has been programmed in a block that contains a call to a subroutine (L, LL, #CALL, #PCALL, #MCALL or G180-G189).
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	Check the program.

1523   T must be programmed in the block.

DETECTION	During execution.
CAUSE	POS and/or MZ has been programmed in a block without programming the tool.
SOLUTION	The POS and MZ commands must be programmed in the same block as the tool.

1525   'The axis cannot be exchanged'

DETECTION	During execution.
CAUSE	The CNC has tried to change a channel axis (#CALL, #SET oR #FREE) and the axis does not have permission to change (parameter AXISEXCH).
SOLUTION	In order for the CNC to be able to change the channel axes and spindles, they must have permission. Parameter AXISEXCH sets whether the axis or the spindle has permission to change channels and if it does, whether the change is temporary or permanent; in other words, whether the change is maintained after an M02, M30, a reset or after restarting the CNC.

1526   '#EXEC: the indicated channel is not a CNC channel'

DETECTION	During execution.
CAUSE	The CNC has tried to use the #EXEC instruction to execute a program in a channel that is not a CNC channel, but a PLC channel.
SOLUTION	Check the program.

1527   'Wrong identifier after %'

DETECTION	During execution.
CAUSE	In the instruction #MSG, #ERROR, #WARNING or #WARNINGSTOP, an invalid identifier has been programmed.
SOLUTION	Check the program.

1529   'Identifier list or "]" expected after end quotes'

DETECTION	During execution.
CAUSE	In the instruction #MSG, #ERROR, #WARNING or #WARNINGSTOP, an invalid character has been programmed after the message to be displayed.
SOLUTION	Check the program. After the end-quotes of the message to be displayed, it only admits the list of variables or parameters to be included in the text.

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 syntax of the instruction is wrong or there is more data in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1532 **'Program: #MASTER <spindle name>'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong or there is more data in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1533 **'Program: #FREESP [sp1, sp2, ..]'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong or there is more data in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1534 **'Program: #CALLSP [sp1, sp2, ..]'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong or there is more data in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1535 **'Program: #SETSP [sp1, sp2, ..]'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong or there is more data in the block.

SOLUTION   Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1538 **'A spindle cannot be restored from the channel'**

DETECTION   During execution.

CAUSE   A channel has released a spindle with a temporary change permission (parameter AXISEXCH) and cannot restore it with a reset or at the beginning of a program because the channel that took it has not released (freed) it.

SOLUTION   The other channel will release the spindle with reset or with the beginning of another program. The spindle release can also be programmed with the instruction #FREE SP.

1539 **'Spindle name repeated in the resulting group'**

DETECTION   During execution.

CAUSE   The #RENAME instruction has tried to rename more than one spindle with the same name.

SOLUTION   Rename the spindles so there aren't two with same name in the channel.

1540 **'Programming not allowed without a master spindle in the channel'**

DETECTION   During execution.

CAUSE   The possible causes are:
- The CNC has tried to read or write a variable of 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

1541 **'The spindle cannot be eliminated while the C axis is active'**

DETECTION   During execution.

CAUSE   The CNC has tried to eliminate from the channel (#FREE or #SET) a spindle that is working as C axis.

SOLUTION   Deactivate the C axis before eliminating the spindle from the channel.

1542   'The spindle cannot be exchanged'

DETECTION   During execution.

CAUSE   The CNC has tried to change a channel spindle (#CALL, #SET oR #FREE) and the spindle does not have permission to change (parameter AXISEXCH).

SOLUTION   In order for the CNC to be able to change the channel axes and spindles, they must have permission. Parameter AXISEXCH sets whether the axis or the spindle has permission to change channels and if it does, whether the change is temporary or permanent; in other words, whether the change is maintained after an M02, M30, a reset or after restarting the CNC.

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 CNC cannot change the spindle gear if tapping G63 is active or the spindle is working as C axis.

SOLUTION   Deactivate the threading and the C axis to change the spindle gear.

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 through GEAR4 for the spindle. Then, the spindle has no active gear and if it is not used before programming G63, the CNC will not generate a gear automatically, either.

SOLUTION   Before starting a G63 tapping, the spindle must have a gear activated. If the spindle has an automatic gear changer (parameter AUTOGEAR), the CNC generates the gear when programming a speed; otherwise, the gear (M41-M44) must be programmed with the speed.

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 deactivating the C axis 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. The cannel cannot eliminate this spindle while the 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 CNC has tried to activate a C axis while another one was active.

SOLUTION   Deactivate a C axis before activating 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.

1553 **'Too many user variables'**

DETECTION	During execution.
CAUSE	The CNC has exceeded the maximum number of user variables (prefixes P and S) admitted.
SOLUTION	The maximum number of user variables admitted is 20.

1554 **'The PLC has not recognized the START in a #EXEC instruction'**

DETECTION	During execution.
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 the PLC program.

1555 **'Program #MOVE / #MOVE ABS/ADD/INF[...]'**

DETECTION	During execution.
CAUSE	The syntax of the #MOVE instruction is wrong.
SOLUTION	Check the program. Together with the instruction, program one of the commands ABS/ADD/INF and then the call parameters between square brackets.

1556 **'Program type of link: PRESENT, NEXT, NULL, WAITINPOS'**

DETECTION	During execution.
CAUSE	In the #MOVE instruction, the type of transition between movements is wrong.
SOLUTION	Check the program.

1557 **'Program #CAM ON/OFF/SELECT/DESELECT[...]'**

DETECTION	During execution.
CAUSE	The syntax of the #CAM instruction is wrong.
SOLUTION	Check the program. Together with the instruction, program one of the commands ON, OFF, SELECT or DESELECT and then the call parameters between square brackets.

1558 **'Wrong cam number'**

DETECTION	During execution.
CAUSE	In the #CAM instruction, the cam number is wrong.
SOLUTION	The cam number must be between 1 and NLEVAS.

1559 **'Program type of cam: ONCE or CONT'**

DETECTION	During execution.
CAUSE	In the #CAM instruction, the cam type is wrong; the cam type is defined with the commands ONCE (non-periodic cam) and CONT (periodic cam).
SOLUTION	Check the program.

1560 **'Program #FOLLOW ON/OFF[...]'**

DETECTION	During execution.
CAUSE	The syntax of the #FOLLOW instruction is wrong.
SOLUTION	Check the program. Together with the instruction, program one of the commands ON/OFF and then the call parameters between square brackets.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF. 2209

1561   'Invalid master axis'

DETECTION   During execution.
CAUSE   In the #CAM or #FOLLOW instruction, the master axis is wrong.
SOLUTION   The axis must belong to the channel.

1562   'Invalid slave axis'

DETECTION   During execution.
CAUSE   In the #CAM or #FOLLOW instruction, the slave axis is wrong.
SOLUTION   The axis must belong to the channel.

1563   'Function G74 must be programmed with axes or alone in the block'

DETECTION   During execution.
CAUSE   Function G74 has been programmed wrong.
SOLUTION   Function G74 may be programmed either with the axes to be homed or alone in the
                 block. If function G74 is programmed alone in the block, the CNC executes its
                 associated subroutine (parameter REFPSUB).

1564   'A parked axis cannot be programmed'

DETECTION   During execution.
CAUSE   The programmed instruction cannot be executed on a parked axis.
SOLUTION   Check the program.

1565   'The spindle cannot work as C axis, CAXIS = No'

DETECTION   During execution.
CAUSE   The #CAX instruction contains a spindle that does not have permission to work as
                 C axis (parameter CAXIS).
SOLUTION   Check the program.

1566   'Mirror image is not admitted with UNIDIR axes'

DETECTION   During execution.
CAUSE   Mirror image cannot be programmed on a unidirectional rotary axis (parameter
                 UNIDIR), because this type of axes cannot rotate in the opposite direction to the one
                 defined.
SOLUTION   Change the type of axis or do not program the mirror image.

1567   'The Polar radius can only be programmed with R'

DETECTION   During execution.
CAUSE   The radius of a movement in Polar coordinates is programmed with parameter R1
                 or with function G263.
SOLUTION   Program the Polar radius with parameter R.

1568   'Wrong tool magazine number'

DETECTION   During execution.
CAUSE   Wrong tool magazine number defined in the MZ command.
SOLUTION   The tool magazine number must be a value between 1 and 4.

1569   'Execution not admitted, the channel is exclusive to the PLC'

DETECTION   During execution.
CAUSE   When the channel is a PLC channel only, the CNC does not have permission to
                 execute in it programs, MDI blocks, jog movements or any other action from the jog
                 mode.
SOLUTION   To execute those actions in that channel, it must be configured as a CNC channel
                 or as CNC+PLC channel (parameter CHTYPE).

1570   'G74 and simultaneous spindle movement not allowed'

DETECTION   During execution.
CAUSE   An M function for spindle movement and function G74 (home search) have been
                 programmed in the same block.
SOLUTION   Check the program.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·72·

1571   **'The "%" character is not allowed inside a global subroutine'**

DETECTION   During execution.

CAUSE   A global subroutine contains the "%" character in a line other than the one defining the name of the subroutine.

SOLUTION   Check the program.

1572   **'Variable that can only be accessed by its channel'**

DETECTION   During execution.

CAUSE   The CNC has tried to access a tool preparation variable of another channel (prefix "G" or "A").

SOLUTION   The only tool variables that can be accessed from another channel are the ones associated with the tool manager (prefix "TM").

1573   **'Error when capturing samples of the variable'**

DETECTION   During execution.

CAUSE   An error occurred while the oscilloscope tried to read the value of the variable assigned to one of its channels.

SOLUTION   Reset the CNC and capture the trace again.

1574   **'Parameter that cannot be modified from the oscilloscope'**

DETECTION   During execution.

CAUSE   The user has tried to modify from the screen of the oscilloscope a machine parameter that does not have a writing permission.

SOLUTION   This parameter can only be modified from the machine parameters table.

1575   **'Instruction only valid in programs with extension FBS'**

DETECTION   During execution.

CAUSE   Programming of instruction #SPLINE or #BSPLINE.

SOLUTION   These instructions are only valid in programs with extension fbs'

1576   **'"]"not expected'**

DETECTION   During execution.

CAUSE   Syntax error in the instruction.

SOLUTION   Check the program.

1577   **'Program type of synchronization: VEL or POS'**

DETECTION   During execution.

CAUSE   The parameter that defines the type of synchronization in the #FOLLOW instruction is wrong.

SOLUTION   Check the program. The parameter can take the values VEL (velocity) and POS (position).

1578   **'#LINK: A UNIDIR slave axis must be of the same type as the master (UNIDIR)'**

DETECTION   During execution.

CAUSE   The CNC has tried to activate a coupling (#LINK) with two unidirectional rotary axes that are not identical.

SOLUTION   When the axes are unidirectional, both axes must be set with the same turning direction (parameter UNIDIR).

1579   **'Home search not allowed on a master axis with several slave axes'**

DETECTION   During execution.

CAUSE   The CNC has executed a G74 function on the master axis of a coupling with several slaves.

SOLUTION   Undo the axis coupling (slaving) and home the axes separately.

1580   **'#CAX: Requires previously programming M19 in position sercos'**

DETECTION   During execution.

CAUSE   The CNC has tried to activate a Sercos spindle as C axis without previously orienting it with M19.

SOLUTION   Program M19 before activating the spindle as C axis.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·73·

1581 **'An active C axis cannot be eliminated'**

DETECTION   During execution.

CAUSE   Instructions #SET AX and #FREE AX cannot eliminate the C axis from the configuration if it is active.

SOLUTION   Deactivate the C axis before eliminate it from the configuration.

1582 **'NR: The number of repetitions has been programmed twice'**

DETECTION   During execution.

CAUSE   The NR instruction has been programmed more than once in the same block.

SOLUTION   Check the program.

1583 **'#FOLLOW ON: The master and slave axes cannot be the same ones'**

DETECTION   During execution.

CAUSE   The master and slave axes are the same axis.

SOLUTION   Check the program.

1584 **'#FOLLOW ON: the numerator and denominator must be integers'**

DETECTION   During execution.

CAUSE   The numerator and/or denominator are programmed with decimals.

SOLUTION   Program integers in the parameters that define the numerator and denominator of the gear ratio.

1585 **'The axes of the active transformation cannot be temporarily swapping axes'**

DETECTION   During execution.

CAUSE   The axes of the transformed cannot be temporary-exchange axes because the defined transform stays active once activated.

SOLUTION   If possible, define the axes as maintained exchange; otherwise, use other axes for the transform. The temporary-exchange axes can only be used in their channel and when the axes are in their original positions inside the channel.

              Temporary-exchange axes may be used in the #FACE and #CYL transforms because these transforms are canceled with M30 and after a reset.

1586 **'The magazine number (MZ) has not been programmed yet'**

DETECTION   During execution.

CAUSE   In a system with several magazines, the tool and the position have been programmed in the same block, but not the tool magazine where that tool is to be placed.

SOLUTION   In a multi-magazine system, the tool, the position and the magazine must be programmed in the same block.

1587 **'This parameter cannot be modified from setup'**

DETECTION   During execution.

CAUSE   The machine parameter has no permission to be written from a setup environment.

SOLUTION   The machine parameter can be modified directly at the machine parameters table.

1588 **'The slave spindle to be synchronized must belong to the channel'**

DETECTION   During execution.

CAUSE   The master spindle of a synchronization may belong to any channel, but the slave spindle must belong to the channel where the synchronization is programmed.

SOLUTION   Synchronize the slave spindle from its channel, add it to the current channel configuration or use another spindle of the channel as slave.

1589 **'The slave spindle cannot be the same as the master'**

DETECTION   During execution.

CAUSE   In the synchronization, the slave axis is the same as the master axis.

SOLUTION   Check the program.

1590 **'Program #SYNC[Sm,Ss,Nx,Dx,Ox,CLOOP/OLOOP,CANCEL/NOCANCEL]'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong or there is more data in the block.

SOLUTION   Check the program.

1591   'Wrong numerator'

DETECTION   During execution.

CAUSE   In spindle synchronization, the value of the numerator of a gear ratio is wrong; zero, for example.

SOLUTION   The value may be positive or negative, not necessarily an integer, but must be other than zero.

1592   'Wrong denominator'

DETECTION   During execution.

CAUSE   In spindle synchronization, the value of the denominator of a gear ratio is wrong; zero, for example.

SOLUTION   The value may be positive or negative, not necessarily an integer, but must be other than zero.

1593   'The slave spindle to be desynchronized must belong to the channel'

DETECTION   During execution.

CAUSE   A channel cannot cancel the synchronization of a slave spindle that belongs to another channel.

SOLUTION   Cancel the synchronization from the channel that the spindle belongs to.

1594   'The slave spindle is already synchronized with another spindle'

DETECTION   During execution.

CAUSE   The slave spindle is already synchronized with another spindle.

SOLUTION   A spindle can only be the slave of one master. Cancel the previous synchronization or synchronize another spindle that is available.

1595   'A spindle cannot be master and slave of a synchronization at the same time'

DETECTION   During execution.

CAUSE   The slave spindle of a synchronization is the master spindle of another one.

SOLUTION   Check the program.

1596   'The spindle is not synchronized'

DETECTION   During execution.

CAUSE   The CNC has tried to desynchronize a slave spindle that is not currently synchronized.

SOLUTION   Check the program.

1597   'A slave spindle cannot be synchronized with two masters'

DETECTION   During execution.

CAUSE   A slave spindle cannot be synchronized with two masters at the same time.

SOLUTION   Decide which of the two synchronizations you wish to activate.

1598   'A slave spindle cannot be programmed during synchronization'

DETECTION   During execution.

CAUSE   An S speed or an M function has been programmed for the slave axis of a synchronization.

SOLUTION   Delete those data from the program or desynchronize the spindle.

1599   'Gear change is not possible on synchronized spindles'

DETECTION   During execution.

CAUSE   While synchronizing two spindles, the CNC does not allow a gear change in any of them.

SOLUTION   Cancel the synchronization to change the spindle gear.

1600   'A synchronized spindle cannot work as C axis or in G63'

DETECTION   During execution.

CAUSE   The CNC has tried to synchronize a spindle that is either working as C axis or has function G63 active. The CNC has tried to activate the spindle as C axis or function G63, but the spindle is synchronized.

SOLUTION   Cancel the C axis or function G63 to synchronize the spindles. To work with the C axis or with function G63, cancel the spindle synchronization.

1601   'A synchronized spindle cannot be released (freed)'

DETECTION	During execution.
CAUSE	Instructions #SET SP and #FREE SP cannot eliminate a spindle if it is synchronized.
SOLUTION	Cancel the synchronization to eliminate the spindle from the configuration.

1602   'To park a spindle, it must be stopped (M5)'

DETECTION	During execution.
CAUSE	The CNC has tried to park a spindle that is not stopped.
SOLUTION	Stop the spindle before parking it.

1603   'FOLLOW OFF: Program slave axis'

DETECTION	During execution.
CAUSE	The programmed axis is not slave in any synchronization.
SOLUTION	Check the program.

1604   'G31 requires G02/G03 active'

DETECTION	During execution.
CAUSE	Function G31 can only be programmed if a circular interpolation is active.
SOLUTION	Program G02/G03 in the G31 block.

1605   'G31 requires programming the polar angle and its interpolation center'

DETECTION	During execution.
CAUSE	The Polar angle or some coordinate of the center is missing in function G31.
SOLUTION	Check the program.

1606   '#ANGAX: The angular and orthogonal axes must belong to the same channel'

DETECTION	During execution.
CAUSE	The CNC has tried to activate an angular transformation with two axes that do not belong to the same channel.
SOLUTION	The axes of an angular transformation must belong to the same channel.

1607   'Executing a milling cycle requires three axes in the channel'

DETECTION	During execution.
CAUSE	A milling cycle has been programmed in a channel with less than three axes or the longitudinal axis does not exist.
SOLUTION	Executing milling cycles requires three axes in the channel.

1608   'The variable cannot be consulted. Assign the value of the variable to Pn and consult Pn'

DETECTION	During execution.
CAUSE	Particular CNC commands and instructions do not admit reading variables that imply a synchronization between block preparation and execution.
SOLUTION	To avoid this error, assign the value of the variable to a local arithmetic parameter and read this parameter when necessary.

1609   'Executing a turning cycle requires two axes in the channel'

DETECTION	During execution.
CAUSE	A lathe cycle has been programmed in a channel with less than two axes or the first axis is not longitudinal (parameter LONGAXIS) or the second one is not a face axis (parameter FACEAXIS).
SOLUTION	Executing lathe cycles requires two axes in the channel.

1610   'A SERCOS master spindle in POSITION does not admit CLOOP'

DETECTION	During execution.
CAUSE	A spindle configured as position-Sercos cannot be synchronized in closed loop.
SOLUTION	Use another spindle as master.

1611   'On the LATHE XZ model the active plane does not change'

DETECTION	During execution.
CAUSE	The CNC has tried to change the work plane in a lathe having a plane-type axis configuration.
SOLUTION	On a lathe having a plane-type axis configuration, the active plane is always G18. To only change the longitudinal axis, use function G20.

1612   'The axes of the active angular transformation, #ANGAX, cannot be excluded'

DETECTION	During execution.
CAUSE	The CNC has tried to eliminate from the channel the axes that made up the angular transformation.
SOLUTION	Cancel the angular transformation before eliminating the axes of the channel.

1613   'The angular transformation #ANGAX has been canceled'

DETECTION	During execution.
CAUSE	The CNC has eliminated from the channel the axes that made up the angular transformation and the latter has been canceled.
SOLUTION	The angular transformation is canceled; to activate it, put in the channel again the axes that make up the angular transformation.

1614   'It cannot be programmed while the angular transformation #ANGAX is active'

DETECTION	During execution.
CAUSE	The CNC has tried to execute a function incompatible with the angular transformation, for example a home search (G74), modify the software limits (G198 - G199), #OSC instruction.
SOLUTION	Cancel the active angular transformation to execute the incompatible functions.

1615   'The axis cannot be activated as Hirth, because it is part of the #ANGAX transformation'

DETECTION	During execution.
CAUSE	The CNC has tried to activate a hirth axis that belongs to the active angular transformation.
SOLUTION	Cancel the angular transformation to activate the hirth axis.

1616   'G31 does not allow programming the polar radius'

DETECTION	During execution.
CAUSE	A Polar radius has been defined for function G31.
SOLUTION	Function G31 does not allow programming the Polar radius. Function G31 only admits Polar coordinates like G31 Q I J; i.e. programming the angle and one or both center coordinates.

1617   'Q index not allowed'

DETECTION	During execution.
CAUSE	The index of parameter Q is wrong; it only admits Q or Q1.
SOLUTION	Check the program. When programming in Polar coordinates, use parameter Q. For function G33, use parameter Q1.

1618   'Q value programmed twice'

DETECTION	During execution.
CAUSE	The Q parameter has been programmed more than once in the same block.
SOLUTION	Check the program.

1619   'Q1 value programmed twice'

DETECTION	During execution.
CAUSE	The Q1 parameter has been programmed more than once in the same block.
SOLUTION	Check the program.

1620   '#SERVO ON not admitted for POSITION-SERCOS axis or spindle'

DETECTION	During execution.
CAUSE	Axis or spindle is position Sercos.
SOLUTION	This instruction only admits velocity-Sercos axes.

1621   'Q1 not admitted without G33 in the block or G32 active'

DETECTION	During execution.
CAUSE	A parameter Q1 has been programmed without G33 in the block or without function G63 being active'
SOLUTION	Parameter Q must go in the same block as G33. Parameter Q can only go alone in the block if function G63 is active.

1622 **'A temporary-exchange spindle cannot work as maintained C axis'**

DETECTION	During execution.
CAUSE	The CNC can activate a temporary-exchange spindle as maintained C axis (parameter PERCAX).
SOLUTION	If possible, define the spindle as maintained exchange; otherwise, use another spindle as C axis.

1623 **'Invalid set for synchronization SYNCSET'**

DETECTION	During execution.
CAUSE	Parameter SYNCSET has the wrong value.
SOLUTION	Assign a correct value to the parameter.

1624 **'Program #TANGCTRL ON/OFF/SUSP [ ]'**

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program. Together with the instruction, program one of the commands ON/SUSP/OFF and then the call parameters between square brackets. It is also possible to program one of the SUSP/OFF commands without call parameters.

1625 **'An axis in active tangential control must be rotary, non-HIRTH and it must not belong to the active trihedron'**

DETECTION	During execution.
CAUSE	The axes to activate in tangential control must be rotary, not hirth and must not belong to the active trihedron nor be the longitudinal axis.
SOLUTION	Check the program.

1626 **'Program axis for #TANGCTRL ON/OFF/SUSP/RESUME'**

DETECTION	During execution.
CAUSE	The axis or axes are missing in the instruction.
SOLUTION	Check the program.

1627 **'#TANGCTRL: program angle values between 0 and ±359.9999'**

DETECTION	During execution.
CAUSE	The angle programmed in the instruction is wrong.
SOLUTION	Check the program. The angle value must be between 0 and ±359.9999.

1628 **'An axis cannot be eliminated from the configuration while tangential control is active'**

DETECTION	During execution.
CAUSE	The CNC cannot eliminate the tangential axis from the configuration if tangential control is active.
SOLUTION	Cancel the tangential control to eliminate this axis from the configuration.

1629 **'Only one axis can be programmed in tangential control'**

DETECTION	During execution.
CAUSE	Tangential control being active, a movement or another operation has been programmed for the tangential axis.
SOLUTION	Tangential control being active, it is not possible to program movements of the tangential axis; it is up to the CNC to orient this axis. To use this axis, cancel the tangential control.

1630 **'The profile has been programmed wrong with labels'**

DETECTION	During execution.
CAUSE	In the call to the cycle, the parameters that define the beginning and end of the profile (parameters S and E) have been defined wrong.
SOLUTION	Assign the correct values to parameters S and E.

1631 **'Profile programming not allowed' [S,E,<Q>] or [P,<Q>]'**

DETECTION	During execution.
CAUSE	In the call to the cycle, the parameters that define the beginning and end of the profile (parameters S and E), the subroutine that contains the profile (parameter P) or the file that contains the profile (parameter Q) have been defined wrong.
SOLUTION	Check the program.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

1632   'An axis in active tangential control must be rotary with 360º module'

DETECTION	During execution.
CAUSE	The CNC has tried to activate the tangential control on a rotary axis with a module other than 360º.
SOLUTION	Tangential control can only be activated on rotary axes with 360º module. The upper limit of the module is determined by parameter MODUPLIM and the lower limit by parameter MODLOWLIM.

1633   'Program: #DGSPSL <spindle name>'

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program.

1634   'The #MOVE instruction only admits one axis and it must belong to the channel'

DETECTION	During execution.
CAUSE	The programmed axis name is wrong, the axis is in another channel or it is an axis of the system. Spindles cannot be programmed except when they are working as C axis, in which case, they must be programmed with the name of the C axis.
SOLUTION	Check the program.

1635   'There are too many axes in the transformation'

DETECTION	During execution.
CAUSE	There are more axes defined in parameter NKINAX of the coordinate transformation than allowed at the CNC.
SOLUTION	Correct machine parameter NKINAX.

1636   'The FACE and CYL transforms cannot be active at the same time'

DETECTION	During execution.
CAUSE	The CNC has tried to activate the #FACE or #CYL instruction while one of them was active.
SOLUTION	Deactivate one of them before activating the other one.

1637   'Spindle turning direction incompatible with the tool'

DETECTION	During execution.
CAUSE	The pre-established tool rotating direction is incompatible with the current turning direction of the spindle.
SOLUTION	Change the spindle turning direction or correct the turning direction defined for the tool.

1638   'The axes of the angular transform, #ANGAX, must be linear'

DETECTION	During execution.
CAUSE	The CNC has tried to activate an angular transformation with a rotary axis.
SOLUTION	The axes of the angular transformation must be linear.

1639   'The programmed spindle does not belong to the channel'

DETECTION	During execution.
CAUSE	The CNC has tried to use a spindle of another channel.
SOLUTION	A channel can only control its spindles. Use a spindle that belongs to the channel o include in the channel configuration the spindle causing the error.

1640   'Parked spindles cannot be programmed'

DETECTION	During execution.
CAUSE	The CNC has tried to use a parked spindle.
SOLUTION	Use a spindle that is not parked or unpark it.

1641   '#CYL: Expecting the programming of the cylinder development radius'

DETECTION	During execution.
CAUSE	The value of the developing radius is missing in the #CYL instruction.
SOLUTION	Check the programming for a variable radius, define a zero value so the CNC calculates the radius depending on the position of the tool.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·79·

1642   'Slaved TANDEM spindles cannot be programmed'

DETECTION	During execution.
CAUSE	The user has tried to move or home the slave spindle of a tandem pair.
SOLUTION	The slave spindle of a tandem pair is controlled by the CNC, it cannot be moved individually. To move a slave spindle, move the master spindle it is associated with.

1643   'TANDEM spindles cannot be parked'

DETECTION	During execution.
CAUSE	The CNC has tried to park a spindle that belongs to a tandem pair.
SOLUTION	Tandem spindles cannot be parked.

1644   'Spindles cannot be parked in G33/G95'

DETECTION	During execution.
CAUSE	The CNC has tried to park a spindle while function G33 or G95 was active.
SOLUTION	Cancel G33 and G95 before parking a spindle.

1645   '#RET/M17/M29 not allowed between the labels of a #RPT'

DETECTION	During execution.
CAUSE	The CNC has detected an M17, M29 function or #RET between the first and last labels of a repetition of blocks #RPT.
SOLUTION	A subroutine cannot end inside a repetition of blocks.

1646   'Program: #ABORT [label]/#ABORT Nxxxx/#ABORT OFF'

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program. The instructions must be programmed alone in the block, only the block label may be added.

1647   'The gear does not exist in the TANDEM slave'

DETECTION	During execution.
CAUSE	A gear change has been programmed in a tandem axis or spindle. The gear exists in the master axis or spindle, but not in the slave.
SOLUTION	Define the same gears on the master and slave axes and spindles of the tandem.

1649   'User reverse transformation function not found'

DETECTION	During execution.
CAUSE	The PcsToMcs coordinates function is missing when trying to do a transformation from part coordinates to machine coordinates in a user kinematics.
SOLUTION	Check the user kinematics integration process.

1650   'User direct transformation function not found'

DETECTION	During execution.
CAUSE	The McsToPcs coordinates function is missing when trying to do a transformation from machine coordinates to part coordinates in a user kinematics.
SOLUTION	Check the user kinematics integration process.

1651   'Maintained LINK not admitted with temporary axes of another channel'

DETECTION	During execution.
CAUSE	The CNC cannot couple axes with temporary exchange permission (parameter AXISEXCH) if they belong to another channel and the couplings of the channel are defined as maintained (parameter LINKCANCEL).
SOLUTION	Couple the axes in their channel.

1652   #DEF "macro"="command" "macro"="command"... was expected.

DETECTION	During execution.
CAUSE	Poorly timed instruction.
SOLUTION	Check the program.

1653 **'An axis of the inclined plane transformation cannot be a slave'**

DETECTION | During execution.
CAUSE | The CNC has tried to define as slave of a coupling an axis that belongs to the inclined plane.
SOLUTION | Cancel the inclined plane to be able to activate the coupling. An axis that belongs to the inclined plane cannot be the slave axis of a coupling; but the master of a coupling can.

1654 **'Digital bus (Sercos/Mechatrolink)not ready for reading the variable'**

DETECTION | During execution.
CAUSE | With the Sercos bus, the ring is not in phase 4. With the Mechatrolink bus, the bus has not reached phase 3.
| The Sercos ring is not in phase 4.
SOLUTION | Condition the writing of the variable to the SERCOSRDY mark in Sercos or to the MLINKRDY mark in Mechatrolink.

1655 **'Digital bus (Sercos/Mechatrolink)not ready for writing the variable'**

DETECTION | During execution.
CAUSE | With the Sercos bus, the ring is not in phase 4. With the Mechatrolink bus, the bus has not reached phase 3.
SOLUTION | Condition the writing of the variable to the SERCOSRDY mark in Sercos or to the MLINKRDY mark in Mechatrolink.

1656 **'Some axis of the plane is set in diameters'**

DETECTION | When simulating or executing on a lathe a milling canned cycle that contains movements in the axes of the plane.
CAUSE | The coordinate data of the milling canned cycles will be considered in radius. If any of the axes of the plane is in diameters, the CNC interprets its data as radius and warns the user in case he was programming them in diameters.
CONSEQUENCE | The cycle can execute a different geometry than expected.
SOLUTION | Program the data in radius.

1657 **'Wrong variable for spindle that is not a C axis'**

DETECTION | During execution.
CAUSE | The CNC has tried to read an axis variable, but the spindle is activated as C axis.
SOLUTION | The variable only exists for axes and spindles working as C axis.

1658 **'Wrong magazine number'**

DETECTION | During execution.
CAUSE | The magazine number is wrong.
SOLUTION | Check the program. The number of magazine is set in parameter NTOOLMZ.

1659 **'Macro previously defined with #DEF FIX'**

DETECTION | During execution.
CAUSE | The macro already exists; it was defined with the command #DEF FIX. Remember that the macros table is not initialized with the beginning and end of program, only with the instruction #INIT MACROTAB.
SOLUTION | Define the macro with a different name. The macros defined using the command #DEF FIX are not deleted after a reset. Use the command #INIT MACROTAB to reset the whole table of macros; it erases the macros that have been defined with #DEF and with #DEF FIX.

1660 **'Wrong array index'**

DETECTION | During execution.
CAUSE | Commands CNCRD or CNCWR. Some array index programmed in the variable is out of range.
SOLUTION | Check the program. Check the variable and its range of values.

1661 **'The "." is missing after [ChannelNumber]'**

DETECTION | During execution.
CAUSE | Commands CNCRD or CNCWR. The channel number must be between square brackets and followed by a ".".
SOLUTION | Check the program.

FAGOR AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·81·

1662 **'Unknown variable type'**

DETECTION During execution.
CAUSE Commands CNCRD or CNCWR. The programmed variable type does not exist.
SOLUTION Check the program.

1663 **'#DELETE: the variable(s) to be deleted is (are) missing'**

DETECTION During execution.
CAUSE The variable(s) to be deleted is (are) missing in the #DELETE instruction.
SOLUTION Indicate in the #DELETE instruction the variables to be deleted.

1664 **The requested tool is being used in another channel.**

DETECTION During execution.
CAUSE The tool programmed is active in another channel.
SOLUTION Wait for the channel to release the tool. If the tool is not in use, unload the tool from the other channel.

1665 **'Function G174 only admits one axis or spindle'**

DETECTION During execution.
CAUSE More than one axis or spindle have been programmed in function G174.
SOLUTION Program one G174 per axis or spindle.

1666 **'G174: slaved axes (GANTRY or TANDEM) are not admitted'**

DETECTION During execution.
CAUSE The CNC has tried to initialize the machine coordinates (G174) of an axis that is part of a pair of gantry or tandem axes or of an active coupling (#LINK).
SOLUTION Cancel the coupling to initialize the machine coordinates for that axis. The machine coordinates of gantry or tandem axes cannot be initialized.

1667 **'G174: Axes of the active kinematics or transforms are not admitted'**

DETECTION During execution.
CAUSE The CNC has tried to initialize the machine coordinates (G174) of an axis that is part the active kinematics or transforms #CS, #ACS, #ANGAX, #TANGCTRL.
SOLUTION Cancel the active kinematics or transformation to initialize the machine coordinates for that axis.

1668 **'Multi-axis group out of range'**

DETECTION During execution.
CAUSE Variable (V.)MPG.MULAXISNAMExn[group] or (V.)MPG.KEEPPOSxn[group]. The (group) index does not correspond to any multi-axis group defined in the machine parameters.
SOLUTION Check the program. Program the multi-axis group number properly.

1669 **'Program #ROTATEMZ1-4 Pn/+n/-n'**

DETECTION During execution.
CAUSE The syntax of the instruction is wrong or there is more data in the block.
SOLUTION Check the program. Program the position to be selected on the turret or the number of positions to rotate.

1670 **'The indicated magazine must be a TURRET type'**

DETECTION During execution.
CAUSE The instruction #ROTATEMZ is only valid for a turret type magazine.
SOLUTION Check the program. Verify that the magazine is a turret type.

1671 **Too many local subroutines in the file.**

DETECTION During execution.
CAUSE The program has more local subroutines than admitted by the CNC.
SOLUTION Use fewer local subroutines; group several subroutines in one or use global subroutines. The CNC admits 130 local subroutines per program.

1672 **'G74-G174 not admitted for synchronized spindles'**

DETECTION	During execution.
CAUSE	Once the spindles have been synchronized, the CNC does not allow programming a home search or force a coordinate on any of them.
SOLUTION	The CNC homes the spindles before synchronizing them. To do a home search or force the coordinate, cancel the synchronization.

1673 **'Variable that cannot be accessed by program or MDI'**

DETECTION	During execution.
CAUSE	The programmed variable exists, but does not have access permission from the program or MDI, or is a text variable.
SOLUTION	Check the access permissions of the variables.

1674 **'Program #SELECT PROBE [probe number, POS/NEG]'**

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program.

1675 **Variable not accessible for its channel.**

DETECTION	During execution.
CAUSE	The variable is not accessible from the channel.
SOLUTION	Check the access permissions of the variables.

1676 **'The active inclined plane has been recovered'**

DETECTION	During execution.
CAUSE	After power-up, the CNC has recovered the inclined plane that was active when turning the CNC off.
SOLUTION	The inclined plane may be deactivated with the #CS OFF instruction.

1677 **'#RTCP/TLC has been deactivated'**

DETECTION	During execution.
CAUSE	The CNC has tried to carry out a home search while function #RTCP or #TLC was active.
SOLUTION	Deactivate the #RTCP/TLC functions before searching home.

1678 **'Variable only exists for external analog feedback'**

DETECTION	During execution.
CAUSE	The programmed variable only exists for axes with external analog feedback and the indicated axis does not meet this condition.
SOLUTION	Check the program. Check the variables that may be accessed depending on the type of axes in the manuals.

1679 **'#CAM SELECT not allowed when a cam is being executed or being loaded'**

DETECTION	During execution.
CAUSE	The CNC has requested to load a cam that is in execution or already being loaded.
SOLUTION	Wait for the cam to finish execution or be loaded.

1680 **'Wrong part number'**

DETECTION	During execution.
CAUSE	A wrong part number has been programmed in the #DGWZ instruction.
SOLUTION	Program the part number within the P1-P4 range.

1681 **'Too many channels programmed in the instruction'**

DETECTION	During execution.
CAUSE	More than one channel number have been programmed in the #DGWZ instruction.
SOLUTION	Program a single channel number C1-C4.

1682 **'Only one part number may be programmed'**

DETECTION	During execution.
CAUSE	More than one part number have been programmed in the #DGWZ instruction.
SOLUTION	Program a single part number P1-P4.

1683   'The #REPOS instruction is only allowed in interruption subroutines'

DETECTION	During execution.
CAUSE	The #REPOS instruction is not programmed in an interruption subroutine.
SOLUTION	The #REPOS instruction can only be programmed in interruption subroutines. Remove that instruction from any other subroutine or program.

1684   'Local subroutine and M, T, or G with subroutine in the same block'

DETECTION	During execution.
CAUSE	The same program block is executing a call to a local subroutine and an M, T or G function that have an associated subroutine.
SOLUTION	Program the call to the local subroutine in another program block.

1685   '$GOTO: Jump to blocks nested inside a $IF, $FOR, $WHILE, etc. instruction not allowed'

DETECTION	During execution.
CAUSE	The instruction $GOTO is not inside a loop, but it is jumping to a label defined inside an $IF, $FOR, etc. loop.
SOLUTION	Check the program. There could be no jumps into a loop.

1686   The file path is too long.

DETECTION	During execution.
CAUSE	The defined path is too long.
SOLUTION	The maximum length allowed for the path is 120 characters.

1687   'Program: #REPOS <INT/INI> <sequence of axes/spindles>'

DETECTION	During execution.
CAUSE	The #REPOS instruction has been programmed wrong.
SOLUTION	The #REPOS instruction can include an optional parameter INT or INI, followed by a sequence of axes and/or spindles fo the channel without a particular order.

1688   'No #REPOS instructions of any other type are allowed'

DETECTION	During execution.
CAUSE	There are different types (INT/INI) of #REPOS instructions programmed in the same interruption subroutines.
SOLUTION	All #REPOS instructions must be of the same type.

1689   'No blocks are allowed after #REPOS'

DETECTION	During execution.
CAUSE	A block has been programmed between the #REPOS instructions and the end-of-subroutine block in an interruption subroutine.
SOLUTION	The #REPOS instruction must be the last blocks of the interruption subroutine, except the end-of-subroutine block.

1690   'Multiple axis programmed twice'

DETECTION	During execution.
CAUSE	'There cannot be more than one axis of the same multi-axis group programmed in the same block'
SOLUTION	Check the program. Only program the active axis of the multi-axis group.

1691   'Nested G subroutine'

DETECTION	During execution.
CAUSE	In a subroutine associated with a G function, it is not possible to program the G function that is calling this subroutine.
SOLUTION	Check the program. Do not program the G function inside the subroutine. In the case of the G74, the function may be programmed inside the subroutine as long as it is followed by the axes to be homed.

1692   'Program #VOLCOMP ON/OFF [n]'

DETECTION	During execution.
CAUSE	The instruction and/or some of its parameters are programmed wrong.
SOLUTION	Check the program.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·84·

1693   #RTCP cannot be programmed with the current kinematics. Program #FACE or #CYL'

DETECTION	During execution.
CAUSE	The instruction is not allowed with the current active kinematics.
SOLUTION	Program the #FACE or #CYL instruction to activate that transformation.

1694   Coordinate out of range in G174.

DETECTION	During execution.
CAUSE	The programmed coordinate in G174 is out of range.
SOLUTION	Program a dimension between the values ±2147483647.

1695   Program #SWTOUT ON [<TONvalor> <,TOFvalor> <,PONvalor> <,POFvalo> <,G0/G1>] /OFF.

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program.

1696   *.fst subroutines cannot be deleted when one of them is being executed.

DETECTION	During execution.
CAUSE	An attempt was made to delete an fst subroutine while it is running.
SOLUTION	Wait for the program to finish running.

1697   'Program #CSROT [ON]/OFF'

DETECTION	During execution.
CAUSE	The instruction and/or some of its parameters are programmed wrong.
SOLUTION	Check the program.

1698   'Program #DEFROT [action, criteria angle]'

DETECTION	During execution.
CAUSE	The instruction and/or some of its parameters are programmed wrong.
SOLUTION	Check the program.

1699   'Function CSROT cannot be applied to kinematics with manual or Hirth axes'

DETECTION	During execution.
CAUSE	Rotary axes of the kinematics must be continuously servo-controlled.
SOLUTION	Deactivate the instruction #CSROT.

1949   'The cutter cannot machine the whole profile'

DETECTION	While executing a roughing canned cycle, pattern repeat on a lathe.
CAUSE	The tool geometry makes it impossible for the tool to machine the whole programmed profile.
CONSEQUENCE	The cycle cannot machine the whole programmed profile.
SOLUTION	Change the profile or the tool so it can machine the whole profile.

1700   'CENTER PUNCHING: F = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter F is 0. No feedrate has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1701   'CENTER PUNCHING: S = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter S is 0. No speed has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1702   'CENTER PUNCHING: T = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter T is 0. No tool has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1703   'CENTER PUNCHING: P = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter P is 0. No depth has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1704 **'CENTER PUNCHING: ALPHA = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter alpha is 0. No angle has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1705 **'CENTER PUNCHING: DIAMETER = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of the parameter to define the diameter is 0.
SOLUTION   Assign a value greater than 0 to the parameter.

1706 **'DRILLING 1: F = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter F is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1707 **'DRILLING 1: S = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter S is 0. No speed has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1708 **'DRILLING 1: T = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T is 0. No tool has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1709 **'DRILLING 1: P = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1710 **'DRILLING 2: F = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter F is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1711 **'DRILLING 2: S = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter S is 0. No speed has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1712 **'DRILLING 2: T = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T is 0. No tool has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1713 **'DRILLING 2: P = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1714 **'DRILLING 2: B = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter B is 0. No drilling peck has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1715 **'THREADING: F = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter F is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1716   'THREADING: S = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter S is 0. No speed has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1717   'THREADING: T = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T is 0. No tool has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1718   'THREADING: P = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1719   'REAMING: F = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter F is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1720   'REAMING: S = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter S is 0. No speed has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1721   'REAMING: T = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T is 0. No tool has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1722   'REAMING: P = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1723   'BORING 1: F = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter F is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1724   'BORING 1: S = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter S is 0. No speed has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1725   'BORING 1: T = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T is 0. No tool has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1726   'BORING 1: P = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1727   'DRILLING 3: F = 0'

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter F is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

FAGOR AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

1728   **'DRILLING 3: S = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter S is 0. No speed has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1729   **'DRILLING 3: T = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T is 0. No tool has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1730   **'DRILLING 3: P = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1731   **'BORING 2: F = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter F is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1732   **'BORING 2: S = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter S is 0. No speed has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1733   **'BORING 2: T = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T is 0. No tool has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1734   **'BORING 2: P = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1735   **'RECTANGULAR POCKET: F = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.
                   The value of some F parameter is 0. No feedrate has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1736   **'RECTANGULAR POCKET: S = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of some S parameter is 0; the speed for roughing or finishing is missing.
SOLUTION   Assign a value greater than 0 to the parameter.

1737   **'RECTANGULAR POCKET: T = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.
SOLUTION   Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1738   **'RECTANGULAR POCKET: P = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter P is 0. No depth has been programmed.
SOLUTION   Assign a value greater than 0 to the parameter.

1739   **'RECTANGULAR POCKET: tool diameter smaller than DELTA'**

DETECTION   During the execution of the cycle.
CAUSE   In the roughing or finishing operation, the tool diameter is smaller than the milling pass (parameter Δ).
SOLUTION   Reduce the milling pass or use a tool with a larger diameter.

1740   'RECTANGULAR POCKET: tool diameter greater than the pocket'

DETECTION	During the execution of the cycle.
CAUSE	In the roughing or finishing operation, the tool diameter is greater than the pocket dimensions (parameter L or H).
SOLUTION	Choose a tool with a smaller diameter.

1741   'RECTANGULAR POCKET: Finishing tool diameter smaller than delta'

DETECTION	During the execution of the cycle.
CAUSE	The diameter of the finishing tool is smaller than the residual stock on the side walls (parameter δ).
SOLUTION	Choose a tool of a greater diameter for the finishing pass.

1742   'PRE-EMPTIED POCKET: F = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1743   'PRE-EMPTIED POCKET: S = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of some S parameter is 0; the speed for roughing or finishing is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1744   'PRE-EMPTIED POCKET: T = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.
SOLUTION	Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1745   'PRE-EMPTIED POCKET: P = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter P is 0. No depth has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1746   'PRE-EMPTIED POCKET: tool diameter smaller than DELTA'

DETECTION	During the execution of the cycle.
CAUSE	In the roughing or finishing operation, the tool diameter is smaller than the milling pass (parameter Δ).
SOLUTION	Reduce the milling pass or use a tool with a larger diameter.

1747   'PRE-EMPTIED POCKET: tool diameter greater than the pocket'

DETECTION	During the execution of the cycle.
CAUSE	In the roughing or finishing operation, the tool diameter is greater than the pocket dimensions (parameter L or H).
SOLUTION	Choose a tool with a smaller diameter.

1748   'PRE-EMPTIED POCKET: Finishing tool diameter smaller than delta'

DETECTION	During the execution of the cycle.
CAUSE	The diameter of the finishing tool is smaller than the residual stock on the side walls (parameter δ).
SOLUTION	Choose a tool of a greater diameter for the finishing pass.

1749   'PRE-EMPTIED POCKET: R <r'

DETECTION	During the execution of the cycle.
CAUSE	The pocket radius (parameter R) is smaller than the pre-emptying radius (parameter r).
SOLUTION	The pocket radius (outside radius must be larger than the pre-emptying radius (inside radius).

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·89·

1750   **'RECTANGULAR BOSS: F = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1751   **'RECTANGULAR BOSS: S = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of some S parameter is 0; the speed for roughing or finishing is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1752   **'RECTANGULAR BOSS: T = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.
SOLUTION	Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1753   **'RECTANGULAR BOSS: P = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter P is 0. No depth has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1754   **'RECTANGULAR BOSS: tool diameter smaller than DELTA'**

DETECTION	During the execution of the cycle.
CAUSE	In the roughing or finishing operation, the tool diameter is smaller than the milling pass (parameter Δ).
SOLUTION	Reduce the milling pass or use a tool with a larger diameter.

1755   **'RECTANGULAR BOSS: Finishing tool diameter smaller than delta'**

DETECTION	During the execution of the cycle.
CAUSE	The diameter of the finishing tool is smaller than the residual stock on the side walls (parameter δ).
SOLUTION	Choose a tool of a greater diameter for the finishing pass.

1756   **'CIRCULAR BOSS: F = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1757   **'CIRCULAR BOSS: S = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of some S parameter is 0; the speed for roughing or finishing is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1758   **'CIRCULAR BOSS: T = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.
SOLUTION	Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1759   **'CIRCULAR BOSS: P = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter P is 0. No depth has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1760   **'CIRCULAR BOSS: R = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter R is 0. No radius has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1761   'CIRCULAR BOSS: tool diameter smaller than DELTA'

DETECTION   During the execution of the cycle.

CAUSE   In the roughing or finishing operation, the tool diameter is smaller than the milling pass (parameter Δ).

SOLUTION   Reduce the milling pass or use a tool with a larger diameter.

1762   'CIRCULAR BOSS: Finishing tool diameter smaller than delta'

DETECTION   During the execution of the cycle.

CAUSE   The diameter of the finishing tool is smaller than the residual stock on the side walls (parameter δ).

SOLUTION   Choose a tool of a greater diameter for the finishing pass.

1763   'SURFACE MILLING: F = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.

SOLUTION   Assign a value greater than 0 to the parameter.

1764   'SURFACE MILLING: S = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of some S parameter is 0; the speed for roughing or finishing is missing.

SOLUTION   Assign a value greater than 0 to the parameter.

1765   'SURFACE MILLING: T = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.

SOLUTION   Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1766   'SURFACE MILLING: P = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter P is 0. No depth has been programmed.

SOLUTION   Assign a value greater than 0 to the parameter.

1767   'SURFACE MILLING: L = 0 and H = 0'

DETECTION   During the execution of the cycle.

CAUSE   The length (parameter L) and/or the width of the surface milling (parameter H) are 0.

SOLUTION   Define both dimensions of the surface milling with a value other than zero.

1768   'POINT-TO-POINT PROFILE: F = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.

SOLUTION   Assign a value greater than 0 to the parameter.

1769   'POINT-TO-POINT PROFILE: S = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of some S parameter is 0; the speed for roughing or finishing is missing.

SOLUTION   Assign a value greater than 0 to the parameter.

1770   'POINT-TO-POINT PROFILE: T = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.

SOLUTION   Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1771   'POINT-TO-POINT PROFILE: P = 0'

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter P is 0. No depth has been programmed.

SOLUTION   Assign a value greater than 0 to the parameter.

1772   'POINT-TO-POINT PROFILE: Null profile'

DETECTION	During the execution of the cycle.
CAUSE	The first two points of the profile (P1 and P2) are the same; the cycle considers that no profile has been defined.
SOLUTION	Define the points of the profile correctly. Two identical points indicate the end of the profile.

1773   'POINT-TO-POINT PROFILE: Tool radius equal to or greater than the tangential entry/exit radius'

DETECTION	During the execution of the cycle.
CAUSE	The radius of one of the tools used in the cycle is equal to or greater than the tangential entry radius (radius of point P1) or of the tangential exit (radius of P12).
SOLUTION	Increase the radius of the tangential entry/exit or execute the cycle with a tool of smaller radius.

1774   'PROFILE: F = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1775   'PROFILE: S = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of some S parameter is 0; the speed for roughing or finishing is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1776   'PROFILE: T = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.
SOLUTION	Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1777   'PROFILE: P = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter P is 0. No depth has been programmed.
SOLUTION	Assign a value greater than 0 to the parameter.

1778   'PROFILE: (FINISHING): Tool's cutting length < P'

DETECTION	During the execution of the cycle.
CAUSE	The cutting length of the finishing tool is smaller than the profile depth (parameter P).
SOLUTION	Choose as the finishing tool, a tool with greater cutting length.

1779   'SLOT MILLING: F = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1780   'SLOT MILLING: S = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of some S parameter is 0; the speed for roughing or finishing is missing.
SOLUTION	Assign a value greater than 0 to the parameter.

1781   'GROOVING: T = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.
SOLUTION	Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1782   **'GROOVING: P = 0'**

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter P is 0. No depth has been programmed.

SOLUTION   Assign a value greater than 0 to the parameter.

1783   **'SLOT MILLING: L = 0'**

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter L is 0. The slot (groove) has no length.

SOLUTION   Assign a value greater than 0 to the parameter.

1784   **'SLOT MILLING: tool diameter smaller than DELTA'**

DETECTION   During the execution of the cycle.

CAUSE   In the roughing or finishing operation, the tool diameter is smaller than the milling pass (parameter Δ).

SOLUTION   Reduce the milling pass or use a tool with a larger diameter.

1785   **'GROOVING: tool diameter greater than the pocket'**

DETECTION   During the execution of the cycle.

CAUSE   In the roughing or finishing operation, the tool diameter is greater than the groove dimensions (parameter L or H).

SOLUTION   Choose a tool with a smaller diameter.

1786   **'GROOVING: FINISHING tool diameter smaller than delta'**

DETECTION   During the execution of the cycle.

CAUSE   The diameter of the finishing tool is smaller than the residual stock on the side walls (parameter δ).

SOLUTION   Choose a tool of a greater diameter for the finishing pass.

1787   **'CIRCULAR POCKET: F = 0'**

DETECTION   During the execution of the cycle.

CAUSE   The value of some F parameter is 0; the feedrate for roughing, finishing or Z-axis penetration is missing.

SOLUTION   Assign a value greater than 0 to the parameter.

1788   **'CIRCULAR POCKET: S = 0'**

DETECTION   During the execution of the cycle.

CAUSE   The value of some S parameter is 0; the speed for roughing or finishing is missing.

SOLUTION   Assign a value greater than 0 to the parameter.

1789   **'CIRCULAR POCKET: T = 0'**

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter T of roughing and finishing operations is 0. No tool has been programmed.

SOLUTION   Program a tool for roughing or finishing operations. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1790   **'CIRCULAR POCKET: P = 0'**

DETECTION   During the execution of the cycle.

CAUSE   The value of parameter P is 0. No depth has been programmed.

SOLUTION   Assign a value greater than 0 to the parameter.

1791   **'CIRCULAR POCKET: tool diameter smaller than DELTA'**

DETECTION   During the execution of the cycle.

CAUSE   In the roughing or finishing operation, the tool diameter is smaller than the milling pass (parameter Δ).

SOLUTION   Reduce the milling pass or use a tool with a larger diameter.

1792   **'CIRCULAR POCKET: tool diameter greater than the pocket'**

DETECTION   During the execution of the cycle.

CAUSE   In the roughing or finishing operation, the tool radius is greater than the pocket radius (parameter R).

SOLUTION   Choose a tool with a smaller diameter.

1793   'CIRCULAR POCKET: Finishing tool diameter smaller than delta'

DETECTION   During the execution of the cycle.

CAUSE   The diameter of the finishing tool is smaller than the residual stock on the side walls (parameter δ).

SOLUTION   Choose a tool of a greater diameter for the finishing pass.

1794   'RECTANGULAR POCKET: Tool diameter = 0'

DETECTION   During the execution of a cycle.

CAUSE   The radius of the roughing or finishing tool is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1795   'CIRCULAR POCKET: Tool diameter = 0'

DETECTION   During the execution of a cycle.

CAUSE   The radius of the roughing or finishing tool is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1796   'PRE-EMPTIED POCKET: Tool diameter = 0'

DETECTION   During the execution of a cycle.

CAUSE   The radius of the roughing or finishing tool is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1797   'RECTANGULAR BOSS: Tool diameter = 0'

DETECTION   During the execution of a cycle.

CAUSE   The radius of the roughing or finishing tool is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1798   'CIRCULAR BOSS: Tool diameter = 0'

DETECTION   During the execution of a cycle.

CAUSE   The radius of the roughing or finishing tool is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1799   'SURFACE MILLING: Tool diameter = 0'

DETECTION   During the execution of a cycle.

CAUSE   The radius of the roughing or finishing tool is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1800   'SLOT MILLING: Tool diameter = 0'

DETECTION   During the execution of a cycle.

CAUSE   The radius of the roughing or finishing tool is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1801   'RECTANGULAR POCKET: Beta or theta < 0 or Beta or theta > 90'

DETECTION   During the execution of a pocket.

CAUSE   In the pocket cycle, the lateral penetration angle for roughing (parameter β) or for finishing (parameter θ) is not between 0º and 90º.

SOLUTION   Correct the lateral penetration angle; program a value between 0º and 90º.

1802   'CIRCULAR POCKET: Beta or theta < 0 or Beta or theta > 90'

DETECTION   During the execution of a pocket.

CAUSE   In the pocket cycle, the lateral penetration angle for roughing (parameter β) or for finishing (parameter θ) is not between 0º and 90º.

SOLUTION   Correct the lateral penetration angle; program a value between 0º and 90º.

1803   'PRE-EMPTIED POCKET: Beta or theta < 0 or Beta or theta > 90'

DETECTION   During the execution of a pocket.

CAUSE   In the pocket cycle, the lateral penetration angle for roughing (parameter β) or for finishing (parameter θ) is not between 0º and 90º.

SOLUTION   Correct the lateral penetration angle; program a value between 0º and 90º.

1804   **'G87: Depth = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The resulting depth of the pocket is zero. The programmed depth (parameter I) is the same as the sum of parameters Z and D; if parameter Z has not been programmed, the programmed depth (parameter I) is the same as parameter D.
SOLUTION   Correct the programming of parameters I Z D.

1805   **'G87: tool diameter greater than the pocket'**

DETECTION   During the execution of the cycle.
CAUSE   The tool radius is greater than the dimensions of the pocket (parameters J and/or K).
SOLUTION   Select a tool with a smaller diameter.

1806   **'G87: tool diameter smaller than L'**

DETECTION   During the execution of the cycle.
CAUSE   The tool diameter is smaller than the finishing stock (parameter L).
SOLUTION   Select a tool with a larger diameter.

1807   **'G87: Tool diameter = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The tool diameter is 0.
SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1808   **'G87: Tool missing'**

DETECTION   During the execution of the cycle.
CAUSE   Tool missing at the spindle.
SOLUTION   Program a tool before the cycle or load a tool at the spindle.

1809   **'G87: tool diameter smaller than C'**

DETECTION   During the execution of the cycle.
CAUSE   The tool diameter is smaller than the milling pass (parameter C).
SOLUTION   Reduce the milling pass or use a tool with a larger diameter.

1810   **'G88: Depth = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The resulting depth of the pocket is zero. The programmed depth (parameter I) is the same as the sum of parameters Z and D; if parameter Z has not been programmed, the programmed depth (parameter I) is the same as parameter D.
SOLUTION   Correct the programming of parameters I Z D.

1811   **'G88: tool diameter greater than the pocket'**

DETECTION   During the execution of the cycle.
CAUSE   The tool radius is greater than the dimensions of the pocket (parameter J).
SOLUTION   Select a tool with a smaller diameter.

1812   **'G88: tool diameter smaller than L'**

DETECTION   During the execution of the cycle.
CAUSE   The tool diameter is smaller than the finishing stock (parameter L).
SOLUTION   Select a tool with a larger diameter.

1813   **'G88: Tool diameter = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The tool diameter is 0.
SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1814   **'G88: Tool missing'**

DETECTION   During the execution of the cycle.
CAUSE   Tool missing at the spindle.
SOLUTION   Program a tool before the cycle or load a tool at the spindle.

1815   **'G88: tool diameter smaller than C'**

DETECTION   During the execution of the cycle.
CAUSE   The tool diameter is smaller than the milling pass (parameter C).
SOLUTION   Reduce the milling pass or use a tool with a larger diameter.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

1816   **'The X parameter must be multiple of the I parameter'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   The length of the machining operation (parameter X) must be multiple of the step between machining operations (parameter I).

SOLUTION   Assign a correct value to the parameters.

1817   **'One of parameters I and K must be programmed'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   The step between machining operations (parameter I) or the number of machining operations (parameter K) has not been programmed.

SOLUTION   Assign a correct value to the parameter.

1818   **'J = 0'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   In a multiple machining operation, parameter J (machining step along the ordinate axis) is 0.

SOLUTION   Assign a correct value to the parameter.

1819   **'K = 0'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   In a multiple machining operation, parameter K (number of machining operations) is 0.

SOLUTION   Assign a correct value to the parameter.

1820   **'Two of parameters X, I and K must be programmed'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   Two of these parameters have not been programmed: X (machining length), I (step between machining operations), K (number of machining operations).

SOLUTION   Assign a correct value to the parameters.

1821   **'B must be multiple of I'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   The angular length of the machining operation (parameter B) must be multiple of the angular step between machining operations (parameter I).

SOLUTION   Assign a correct value to the parameters.

1822   **'The Y parameter must be multiple of the J parameter'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   The length of the machining operation (parameter Y) must be multiple of the step between machining operations (parameter J).

SOLUTION   Assign a correct value to the parameters.

1823   **'Two of parameters Y, J and D must be programmed'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   Two of these parameters have not been programmed: Y (machining length), J (step between machining operations), D (number of machining operations).

SOLUTION   Assign a correct value to the parameters.

1824   **'One of parameters I and K must be programmed'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   One of these parameters has not been programmed: I (step between machining operations), K (number of machining operations).

SOLUTION   Assign a correct value to the parameters.

1825   **'One of parameters I and A must be programmed'**

DETECTION   During the execution of a multiple machining cycle.

CAUSE   Two of these parameters have not been programmed: I (angular step between machining operations), A (number of machining operations).

SOLUTION   Assign a correct value to the parameter.

1826 **'Parameter I multiplied by K must be equal to 360'**

DETECTION	During the execution of a multiple machining cycle.
CAUSE	The angular step between machining operations (parameter I) multiplied by the number of machining operations (parameter K) must be 360º.
SOLUTION	Assign a correct value to the parameter.

1827 **'Parameters X and Y are equal to 0'**

DETECTION	During the execution of a multiple machining cycle.
CAUSE	Parameters X Y (center of the machining arc) are 0.
SOLUTION	Assign a correct value to the parameter.

1828 **'360 must be multiple of parameter I'**

DETECTION	During the execution of a multiple machining cycle.
CAUSE	The angular step (parameter I) must be sub-multiple of 360º.
SOLUTION	Assign a correct value to the parameter.

1829 **'I = 0'**

DETECTION	During the execution of a multiple machining cycle.
CAUSE	In a multiple machining operation, parameter I (machining step) is 0.
SOLUTION	Assign a value other than 0 to the parameter.

1830 **'Tool diameter = 0'**

DETECTION	During the execution of a cycle.
CAUSE	The tool diameter is 0.
SOLUTION	Correct the value of the tool radius in the tool table or select another tool.

1831 **'G82: C = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter C (approach coordinate) is 0.
SOLUTION	Assign a value other than 0 to the parameter.

1832 **'ROUGHING: I = 0'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter I (maximum penetration step) is 0.
SOLUTION	Assign to the parameter a value other than zero and smaller than the cutting length of the tool.

1833 **'FINISHING: N = 0 and undefined tool's cutting length.**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter N (number of penetration passes) is 0 and the cutting length of the finishing tool has not been defined.
SOLUTION	Program the number of penetrating passes (parameter N) or define the cutting length in the tool table.

1834 **'FINISHING: Z delta greater than the tool's cutting length'**

DETECTION	During the execution of the cycle.
CAUSE	Parameter δz (finishing stock at the bottom) exceeds the cutting length of the finishing tool.
SOLUTION	Program a smaller finishing stock (parameter δz) or use another tool.

1835 **'Pocket with islands: Wrong safety Z'**

DETECTION	During the execution of the cycle.
CAUSE	The safety plane (parameter Zs) is located inside the part.
SOLUTION	Assign a correct value to the parameter.

1836 **'Pocket with islands: Wrong Z profile'**

DETECTION	During the execution of the cycle.
CAUSE	Some depth profile is closed or the cycle cannot machine some depth profile with the programmed tool.
SOLUTION	Check that the depth profiles are correct.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

1837   'Pocket with islands: Profile in XY intersects itself'

DETECTION	During the execution of the cycle.
CAUSE	Some surface profile forms more than one closed profile.
SOLUTION	Check that the surface profiles do not intersect each others and that only their starting point is the same.

1838   'Pocket with islands: The XY profile is not closed'

DETECTION	During the execution of the cycle.
CAUSE	Some surface profile is not a closed profile; its initial and final points are not the same.
SOLUTION	Check that all the surface profiles are closed.

1839   'Pocket with islands: Wrong intersection of XY profiles'

DETECTION	During the execution of the cycle.
CAUSE	Two surface profiles have some straight or arc section in common.
SOLUTION	Check that the surface profiles do not have sections in common, the profiles must intersect in points.

1840   'Pocket with islands: Not enough memory to resolve it'

DETECTION	During the execution of the cycle.
CAUSE	The system has not enough memory to continue resolving the pocket.
SOLUTION	Contact FAGOR.

1841   'Wrong tool position before the canned cycle'

DETECTION	During the execution of the cycle.
CAUSE	When executing the canned cycle, the tool tip is located between the reference plane and the surface of the part.
SOLUTION	To execute the cycle, position the tool above the reference plane.

1842   'Tool diameter smaller than DELTA'

DETECTION	During the execution of the cycle.
CAUSE	The finishing tool diameter is smaller than the milling pass (parameter Δ).
SOLUTION	Reduce the milling pass or use a tool with a larger diameter.

1843   'Pocket with islands: Wrong roughing tool radius'

DETECTION	During the execution of the cycle.
CAUSE	The roughing tool radius is 0 or the tool is too large for the geometry of the pocket considering the lateral stock (parameter δ).
SOLUTION	Correct the value of the tool radius in the tool table or select another tool. Check the value of the lateral residual stock.

1844   'Pocket with islands: Wrong finishing tool radius'

DETECTION	During the execution of the cycle.
CAUSE	The finishing tool radius is 0 or the tool is too large for the geometry of the pocket.
SOLUTION	Correct the value of the tool radius in the tool table or select another tool.

1845   'G165: Chord length (I) greater than the diameter'

DETECTION	During the execution of a multiple machining cycle.
CAUSE	The length of the chord (parameter I) is greater than the diameter of the circle.
SOLUTION	Assign a correct value to the parameter.

1846   'Zero arc radius'

DETECTION	During the execution of a multiple machining cycle.
CAUSE	The possible causes are: • There is an arc with zero radius in the profile. • In a multiple machining operation in an arc or a circle, both center coordinates are the same as the those of the starting point.
SOLUTION	Check the program.

1847   'RECTANGULAR BOSS: Q = 0'

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter Q (stock to be removed) is 0.
SOLUTION	Assign a value other than 0 to the parameter.

1848 **'CIRCULAR BOSS: Q = 0'**

DETECTION   During the execution of the cycle.
CAUSE   The value of parameter Q (stock to be removed) is 0.
SOLUTION   Assign a value other than 0 to the parameter.

1849 **'RECTANGULAR POCKET: Finishing stock DELTA greater than the pocket'**

DETECTION   During the execution of the cycle.
CAUSE   The size of the pocket minus the tool diameter is smaller than the lateral residual stock (twice parameter δ).
SOLUTION   Assign a correct value to the parameter.

1850 **'Lx must be multiple of Ix'**

DETECTION   During the execution of a multiple machining cycle.
CAUSE   The length of the machining operation (parameter Lx) must be multiple of the step between machining operations (parameter Ix).
SOLUTION   Assign a correct value to the parameters.

1851 **'Ly must be multiple of Iy'**

DETECTION   During the execution of a multiple machining cycle.
CAUSE   The length of the machining operation (parameter Ly) must be multiple of the step between machining operations (parameter Iy).
SOLUTION   Assign a correct value to the parameters.

1852 **'No tool has been programmed'**

DETECTION   During the execution of a canned cycle.
CAUSE   There is neither a tool nor a probe at the spindle so the cycle can be executed.
SOLUTION   Put a tool or a probe at the spindle, depending on the canned cycle.

1853 **'The probe signal has not been received'**

DETECTION   During the execution of a #PROBE probing cycle.
CAUSE   The probe has not touched the part during the probing movement.
SOLUTION   Correct the definition of the canned cycle.

1854 **'The diameter J must be greater than zero'**

DETECTION   During the execution of a #PROBE 2 or #PROBE 9 probing cycle.
CAUSE   Parameter J (theoretical hole diameter) has the wrong value.
SOLUTION   Assign a positive value to the parameter.

1855 **'Tool diameter greater than that of the hole'**

DETECTION   During the execution of a #PROBE 2 or #PROBE 9 probing cycle.
CAUSE   The probe diameter is greater than the diameter of the hole to be measured or greater than the hole used to calibrate the probe.
SOLUTION   Correct the definition of the canned cycle.

1856 **'Wrong withdraw distance E'**

DETECTION   During the execution of a #PROBE 2 or #PROBE 9 probing cycle.
CAUSE   Parameter E (withdrawal distance) has the wrong value.
SOLUTION   Assign to the parameter a value between 0 and the diameter of the hole.

1857 **'There is no tool offset selected'**

DETECTION   During the execution of a #PROBE 1 or #PROBE 2 probing cycle.
CAUSE   There is no active tool offset for the tool calibration cycle.
SOLUTION   Program the tool offset (function D) before the cycle.

1858 **'Program I = 0/1'**

DETECTION   During the execution of a #PROBE probing cycle.
CAUSE   Parameter I (type of calibration) has the wrong value.
SOLUTION   Parameter I only admits values of 0 (single calibration) or 1 (double calibration).

1859 **'The active plane for calibrating the probe must be G17, G18 or G19'**

DETECTION   During the execution of a #PROBE 2 probing cycle.
CAUSE   When executing the cycle, there is an active work plane defined with G20.
SOLUTION   Activate a main plane G17, G18 or G19 to execute the cycle.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

1860   'Program K = 0/1/2'

DETECTION	During the execution of a #PROBE 3 probing cycle.
CAUSE	Parameter K (probing axis) has the wrong value.
SOLUTION	Parameter K only admits values of 0 (abscissa axis), 1 (ordinate axis) or 2 (longitudinal axis).

1861   'The safety distance B must be greater than zero'

DETECTION	During the execution of a #PROBE probing cycle.
CAUSE	The value of parameter B is negative.
SOLUTION	Assign a positive value to the parameter.

1862   '#PROBE 1: not allowed when G20 is active'

DETECTION	During the execution of a #PROBE 1 probing cycle.
CAUSE	When executing the cycle, there is an active work plane defined with G20.
SOLUTION	Activate a main plane G17, G18 or G19 to execute the cycle.

1863   Tool worn due to radius wear

DETECTION	During the execution of a #PROBE 1 probing cycle.
CAUSE	The cycle has detected a tool that is worn out. The measured wear is greater than the permissible tolerance (parameter 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	During the execution of a #PROBE 1 probing cycle.
CAUSE	The tabletop probe position has been defined wrong in the cycle. One of parameters X, Y, Z is greater than its corresponding parameter U, V, W. Parameters X Y Z refer to the minimum coordinates of the probe and parameters U V W refer to the maximum coordinates.
SOLUTION	Define the probe position so U > X, V > Y, W > Z.

1865   'Pocket with islands: Roughing T missing'

DETECTION	During the execution of the cycle.
CAUSE	The programmed tool does not exist in the tool table.
SOLUTION	Correct the tool in the table or select another tool.

1866   'Pocket with islands: Wrong roughing F'

DETECTION	During the execution of the cycle.
CAUSE	The roughing feedrate is 0.
SOLUTION	Define a feedrate higher than 0.

1867   'Pocket with islands: Wrong roughing S'

DETECTION	During the execution of the cycle.
CAUSE	The spindle speed for the roughing operation is 0.
SOLUTION	Define a speed higher than 0.

1868   'Pocket with islands: Finishing T missing'

DETECTION	During the execution of the cycle.
CAUSE	The programmed tool does not exist in the tool table.
SOLUTION	Correct the tool in the table or select another tool.

1869   'Pocket with islands: Wrong finishing F'

DETECTION	During the execution of the cycle.
CAUSE	The finishing feedrate is 0.
SOLUTION	Define a feedrate higher than 0.

1870   'Pocket with islands: Wrong finishing S'

DETECTION	During the execution of the cycle.
CAUSE	The spindle speed for the finishing operation is 0.
SOLUTION	Define a speed higher than 0.

1871   'Pocket with islands: Wrong roughing pass'

DETECTION	During the execution of the cycle.
CAUSE	The roughing pass (parameter Δ) is greater than tool diameter.
SOLUTION	Correct the location code (tool shape) or select another tool.

1872   **'Pocket with islands: Wrong finishing pass'**

DETECTION	During the execution of the cycle.
CAUSE	In a 2D pocket, the finishing pass (parameter Δ) is greater than tool diameter. In a 3D pocket, the finishing pass (parameter ε) is 0.
SOLUTION	Correct the machining pass.

1873   **'Pocket with islands: Wrong side stock (excess material)'**

DETECTION	During the execution of the cycle.
CAUSE	The lateral finishing stock (parameter δ) is greater than tool diameter.
SOLUTION	Correct the lateral finishing stock or select another tool.

1874   **'Pocket with islands: Wrong depth'**

DETECTION	During the execution of the cycle.
CAUSE	The value of parameter P (pocket depth) is 0.
SOLUTION	Assign a correct value to the parameter.

1875   **'Pocket with islands: XY profile does not exist'**

DETECTION	During the execution of the cycle.
CAUSE	The programmed surface profile does not exist.
SOLUTION	Select an existing profile or generate the programmed profile.

1876   **'Pocket with islands: Wrong XY profile'**

DETECTION	During the execution of the cycle.
CAUSE	Geometry of the surface profile programmed wrong.
SOLUTION	Correct the geometry of the profile from the profile editor.

1877   **'Pocket with islands: Z profile does not exist'**

DETECTION	During the execution of the cycle.
CAUSE	The programmed depth profile does not exist.
SOLUTION	Select an existing profile or generate the programmed profile.

1878   **'Pocket with islands: Wrong finishing tool tip (nose) radius'**

DETECTION	During the execution of the cycle.
CAUSE	The tip radius (Rp) of the finishing tool is greater than its radius (R).
SOLUTION	Correct the tool tip (nose) radius.

1879   **'Pocket with islands: Wrong semi-finishing tool radius'**

DETECTION	During the execution of the cycle.
CAUSE	The semi-finishing tool radius 0 or it is too large for the geometry of the pocket.
SOLUTION	Correct the tool in the table or select another tool.

1880   **'Pocket with islands: Semi-finishing T missing'**

DETECTION	During the execution of the cycle.
CAUSE	The programmed tool does not exist in the tool table.
SOLUTION	Correct the tool in the table or select another tool.

1881   **'Pocket with islands: Wrong semi-finishing F'**

DETECTION	During the execution of the cycle.
CAUSE	The semi-finishing feedrate is 0.
SOLUTION	Define a feedrate higher than 0.

1882   **'Pocket with islands: Wrong semi-finishing S'**

DETECTION	During the execution of the cycle.
CAUSE	The spindle speed for the semi-finishing operation is 0.
SOLUTION	Define a speed higher than 0.

1883   **'Pocket with islands: Pocket already in execution'**

DETECTION	During the execution of the cycle.
CAUSE	The CNC cannot simulate or execute the pocket because it is already in execution or simulation.
SOLUTION	Wait for the CNC to finish the simulation or execution.

1884 **'Pocket with islands: Arc programmed wrong in the profile'**

DETECTION	During the execution of the cycle.
CAUSE	In some profile of the pocket, there is an arc that has been defined wrong or the profile has been modified with a program other than the profile editor.
SOLUTION	Correct the geometry of the profile from the profile editor.

1885 **'Pocket with islands: Wrong pocket geometry'**

DETECTION	During the execution of the cycle.
CAUSE	In some profile of the pocket, there is a section that has been defined wrong (rounding, chamfer, etc.).
SOLUTION	Correct the geometry of the profile from the profile editor.

1886 **'Pocket with islands: Wrong roughing tool's penetration angle'**

DETECTION	During the execution of the cycle.
CAUSE	The roughing penetration angle (parameter β) is not between 0º and 90º.
SOLUTION	Correct the penetration angle; program a value between 0º and 90º.

1887 **'Pocket with islands: Wrong penetration angle of the semi-finishing tool'**

DETECTION	During the execution of the cycle.
CAUSE	The finishing penetration angle (parameter θ) is not between 0º and 90º.
SOLUTION	Correct the penetration angle; program a value between 0º and 90º.

1888 **'Pocket with islands: Wrong type of finishing'**

DETECTION	During the execution of the cycle.
CAUSE	The type of finishing (parameter Q) is wrong.
SOLUTION	Assign a correct value to the parameters.

1889 **'Back-and-forth movement: Distance traveled too short'**

DETECTION	Execution of the back-and-forth movement for adjusting the inertia.
CAUSE	The distance programmed in the back-and-forth movement is not long enough.
SOLUTION	Program a longer distance.

1890 **'Back-and-forth movement: Distance traveled too long'**

DETECTION	Execution of the back-and-forth movement for adjusting the inertia.
CAUSE	The distance programmed in the back-and-forth movement is too long.
SOLUTION	Program a shorter distance.

1891 **Tool worn out due to length wear.**

DETECTION	During execution.
CAUSE	The cycle has detected a tool that is worn out. The measured wear is greater than the permissible tolerance (parameter M).
SOLUTION	Repair the tool or use an equivalent tool.

1900 **'Required data not programmed**

DETECTION	During the execution of the cycle.
CAUSE	One or more required parameters have not been programmed. The wrong parameter will be indicated in the error message.
SOLUTION	Program all the mandatory parameters of the cycle.

1901 **'Wrong character'**

DETECTION	During the execution of the cycle.
CAUSE	Some cycle parameter has the wrong value. The wrong parameter will be indicated in the error message.
SOLUTION	Correct the program.

1902 **'Incompatible starting position and location code (shape) of the tool**

DETECTION	During the execution of the cycle.
CAUSE	The initial position of the tool is not compatible with the location code (tool shape) and the geometry of the tool.
SOLUTION	Position the tool properly or change the tool.

1903 'Arc programmed wrong'

DETECTION   During the execution of the cycle.
CAUSE   Arc defined wrong in the geometry of the canned cycle.
SOLUTION   Correct the program.

1904 'No tool has been programmed'

DETECTION   During the execution of the cycle.
CAUSE   No tool has been programmed.
SOLUTION   Program a tool before the cycle or load a tool at the spindle.

1905 'SLOT MILLING: Zero cutter width'

DETECTION   During the execution of the cycle.
CAUSE   The tool width is 0.
SOLUTION   Correct the tool width or select another tool.

1906 'Wrong tool location code (shape)

DETECTION   During the execution of the cycle.
CAUSE   The location code (tool shape) is not valid for the programmed cycle.
SOLUTION   Correct the location code (tool shape) or select another tool.

1907 'Zero feedrate F before the cycle

DETECTION   During the execution of the cycle.
CAUSE   The machining feedrate is 0.
SOLUTION   Define a feedrate higher than 0.

1908 'No residual stock has been programmed'

DETECTION   During the execution of the cycle.
CAUSE   The finishing stock is 0.
SOLUTION   Define a finishing stock greater than 0.

1909 'Zero roughing pass'

DETECTION   During the execution of the cycle.
CAUSE   The value of the machining pass is 0.
SOLUTION   Define a machining pass greater than 0.

1910 'Roughing pass greater than cutter width'

DETECTION   During the execution of the cycle.
CAUSE   The roughing pass is greater than cutter width.
SOLUTION   Program a roughing pass smaller or equal to the cutter width.

1911 'Wrong location code (shape code) in profile roughing cycle'

DETECTION   During the execution of the cycle.
CAUSE   The location code (tool shape) is not valid for the programmed cycle.
SOLUTION   Correct the location code (tool shape) or select another tool.

1912 'Wrong cutter angle'

DETECTION   During the execution of the cycle.
CAUSE   The cutter angle is not valid for the programmed cycle.
SOLUTION   Correct the cutter angle or select another tool.

1913 'Wrong cutting angle of the cutter'

DETECTION   During the execution of the cycle.
CAUSE   The cutting angle is not valid for the programmed cycle.
SOLUTION   Correct the cutting angle or select another tool.

1914 'Wrong roughing F'

DETECTION   During the execution of the cycle.
CAUSE   The roughing feedrate is 0.
SOLUTION   Define a feedrate higher than 0.

1915   'Wrong finishing F'

DETECTION	During the execution of the cycle.
CAUSE	The finishing feedrate is 0.
SOLUTION	Define a feedrate higher than 0.

1916   'Wrong roughing S'

DETECTION	During the execution of the cycle.
CAUSE	The spindle speed for the roughing operation is 0.
SOLUTION	Define a speed higher than 0.

1917   'Wrong finishing S'

DETECTION	During the execution of the cycle.
CAUSE	The spindle speed for the finishing operation is 0.
SOLUTION	Define a speed higher than 0.

1918   'The programmed profile is closed'

DETECTION	During the execution of the cycle.
CAUSE	The profile defined in the canned cycle is not valid because it is a closed profile.
SOLUTION	Program a profile that is not closed.

1919   'The programmed profile intersects itself'

DETECTION	During the execution of the cycle.
CAUSE	The profile defined in the canned cycle is not valid because it intersects itself.
SOLUTION	Program a profile that does not intersect itself.

1920   'No profile has been programmed'

DETECTION	During the execution of the cycle.
CAUSE	The profile that defines the geometry of the canned cycle has not been programmed.
SOLUTION	Define the profile that defines the geometry of the cycle.

1921   'The programmed geometry is not an outside geometry'

DETECTION	During the execution of the cycle.
CAUSE	The geometry programmed in the canned cycle does not correspond to an external geometry, as indicated in the cycle.
SOLUTION	Change the geometry or define it as internal geometry in the cycle.

1922   'The programmed geometry is not an inside geometry'

DETECTION	During the execution of the cycle.
CAUSE	The geometry programmed in the canned cycle does not correspond to an internal geometry, as indicated in the cycle.
SOLUTION	Change the geometry or define it as external geometry in the cycle.

1923   'Wrong geometry'

DETECTION	During the execution of the cycle.
CAUSE	The geometry of the canned cycle is wrong.
SOLUTION	Correct the program.

1924   'Insufficient memory'

DETECTION	During the execution of the cycle.
CAUSE	The system has not enough memory to continue resolving the pocket.
SOLUTION	Contact FAGOR.

1925   'Neither roughing nor finishing has been programmed'

DETECTION	During the execution of the cycle.
CAUSE	No operation has been defined in the canned cycle.
SOLUTION	The canned cycle must have the definition of the roughing or finishing operation. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1926   'Vertex programmed wrong'

DETECTION	During the execution of the cycle.
CAUSE	The geometry of the canned cycle is wrong.
SOLUTION	Correct the program.

1927 **'Wrong cutter for programmed profile'**

DETECTION   During the execution of the cycle.
CAUSE   The tool is not valid for the geometry of the programmed cycle.
SOLUTION   Select another tool.

1928 **'Grooving pass greater than cutter width'**

DETECTION   During the execution of the cycle.
CAUSE   The slot milling (grooving) pass is greater than cutter width.
SOLUTION   Program a grooving (slot milling) pass smaller or equal to the cutter width.

1929 **'No spindle speed has been programmed'**

DETECTION   During the execution of the cycle.
CAUSE   The spindle speed is 0.
SOLUTION   Define a feedrate higher than 0.

1930 **'A zero drilling peck has been programmed'**

DETECTION   During the execution of the cycle.
CAUSE   The value of the drilling peck is 0.
SOLUTION   Define a drilling peck other than 0.

1931 **'No depth has been programmed'**

DETECTION   During the execution of the cycle.
CAUSE   The machining depth is 0.
SOLUTION   Define a machining depth other than 0.

1932 **'Spindle missing to execute the cycle'**

DETECTION   During the execution of the cycle.
CAUSE   The channel that executes the cycle does not a spindle.
SOLUTION   Yield a spindle to the channel or execute the cycle in a channel with spindle.

1933 **'Live tool missing to execute the cycle'**

DETECTION   During the execution of the cycle.
CAUSE   The channel that executes the cycle does not a spindle for the live tool.
SOLUTION   Yield a live tool to the channel or execute the cycle in a channel with live tool.

1934 **'Wrong cutter width'**

DETECTION   During the execution of the cycle.
CAUSE   The cutter width is larger than the slot (groove) or the cycle cannot leave the programmed residual stock.
SOLUTION   Check that the selected tool can machine the programmed slot (groove), leaving the programmed finishing stock.

1935 **'No axis has been defined as LONGAXIS in C axis cycle'**

DETECTION   During the execution of the cycle.
CAUSE   No longitudinal axis has been defined in the channel.
SOLUTION   Set machine parameter LONGAXIS = TRUE for the longitudinal axis.

1936 **'No axis has been defined as FACEAXIS in C axis cycle'**

DETECTION   During the execution of the cycle.
CAUSE   No face axis has been defined in the channel.
SOLUTION   Set machine parameter FACEAXIS = TRUE for the face axis.

1937 **'No axis or spindle has been defined for C axis'**

DETECTION   During the execution of the cycle.
CAUSE   A canned cycle has been programmed for the C axis, but there is no C axis in the channel.
SOLUTION   Define some axis or spindle of the channel as C axis (parameter CAXIS).

1938 **'No axis has been defined for Y axis'**

DETECTION   During the execution of the cycle.
CAUSE   A canned cycle has been programmed for the Y axis, but there is no Y axis in the channel.
SOLUTION   Define a trihedron type axis configuration in the channel (parameter GEOCONFIG) with a Y axis.

1950 **'The Z coordinate of the part surface has not been programmed'**

DETECTION   During the execution of the cycle.

CAUSE   'The coordinate of the part surface has not been programmed (parameter Z).

SOLUTION   Correct the program.

1951 **'The safety Z coordinate has not been programmed'**

DETECTION   During the execution of the cycle.

CAUSE   The safety plane has not been programmed (parameter Zs).

SOLUTION   Correct the program.

1952 **'The part depth has not been programmed'**

DETECTION   During the execution of the cycle.

CAUSE   The part depth has not been programmed (parameter P).

SOLUTION   Correct the program.

1953 **'The penetration step has not been programmed'**

DETECTION   During the execution of the cycle.

CAUSE   The penetration step (parameter I) has not been programmed.

SOLUTION   Correct the program.

1954 **'The roughing pass has not been programmed'**

DETECTION   During the execution of the cycle.

CAUSE   The roughing pass (parameter D) has not been programmed.

SOLUTION   Correct the program.

1955 **'Zero roughing tool radius'**

DETECTION   During the execution of the cycle.

CAUSE   The roughing tool radius is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1956 **'Zero finishing tool radius'**

DETECTION   During the execution of the cycle.

CAUSE   The finishing tool radius is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1957 **'Zero semi-finishing tool radius'**

DETECTION   During the execution of the cycle.

CAUSE   The semi-finishing tool radius is 0.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1958 **'Roughing tool radius too large'**

DETECTION   During the execution of the cycle.

CAUSE   The roughing tool radius is too large for the geometry of the pocket.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1959 **'Finishing tool radius too large'**

DETECTION   During the execution of the cycle.

CAUSE   The finishing tool radius is too large for the geometry of the pocket.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1960 **'Semi-finishing tool radius too large'**

DETECTION   During the execution of the cycle.

CAUSE   The semi-finishing tool radius is too large for the geometry of the pocket.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1961 **'Wrong finishing tool tip radius'**

DETECTION   During the execution of the cycle.

CAUSE   The tip radius (Rp) of the finishing tool is greater than its radius (R).

SOLUTION   Correct the tool tip (nose) radius.

1962   **'Wrong roughing penetration angle'**

DETECTION   During the execution of the cycle.
CAUSE   The roughing penetration angle is not between 0º and 90º.
SOLUTION   Correct the penetration angle; program a value between 0º and 90º.

1963   **'Wrong semi-finishing penetration angle'**

DETECTION   During the execution of the cycle.
CAUSE   The semi-finishing penetration angle is not between 0º and 90º.
SOLUTION   Correct the penetration angle; program a value between 0º and 90º.

1964   **'A roughing pass has been programmed greater than tool diameter'**

DETECTION   During the execution of the cycle.
CAUSE   The roughing pass is greater than tool diameter.
SOLUTION   Correct the location code (tool shape) or select another tool.

1965   **'A finishing pass has been programmed greater than tool diameter'**

DETECTION   During the execution of the cycle.
CAUSE   The finishing pass is greater than tool diameter.
SOLUTION   Correct the location code (tool shape) or select another tool.

1966   **'Wrong type of finishing'**

DETECTION   During the execution of the cycle.
CAUSE   The type of finishing is wrong.
SOLUTION   Assign a correct value to the parameters.

1967   **'Wrong Z coordinate of the starting plane'**

DETECTION   During the execution of the cycle.
CAUSE   The Z coordinate of the beginning of the cycle is located inside the part.
SOLUTION   The Z coordinate of the beginning of the cycle must be above the reference plane.

1968   **'Wrong safety Z coordinate**

DETECTION   During the execution of the cycle.
CAUSE   The safety plane is located inside the part.
SOLUTION   Assign a correct value to the parameter.

1969   **'The programmed XY profile does not exist'**

DETECTION   During the execution of the cycle.
CAUSE   The surface profile does not exist.
SOLUTION   Edit the profile. The CNC saves the profiles in the folder ..\Users\Profile.

1970   **'The programmed Z profile does not exist'**

DETECTION   During the execution of the cycle.
CAUSE   The depth profile does not exist.
SOLUTION   Edit the profile. The CNC saves the profiles in the folder ..\Users\Profile.

1971   **'An open XY profile has been programmed'**

DETECTION   During the execution of the cycle.
CAUSE   Some surface profile is not a closed profile; its initial and final points are not the same.
SOLUTION   Check that all the surface profiles are closed.

1972   **'Wrong XY profile'**

DETECTION   During the execution of the cycle.
CAUSE   The surface profile does not exist.
SOLUTION   Edit the profile. The CNC saves the profiles in the folder ..\Users\Profile.

1973   **'Wrong Z profile'**

DETECTION   During the execution of the cycle.
CAUSE   Some depth profile is closed or the cycle cannot machine some depth profile with the programmed tool.
SOLUTION   Check that the depth profiles are correct.

1974   'Wrong profile intersection'

DETECTION   During the execution of the cycle.

CAUSE   Two surface profiles generate an invalid intersection.

SOLUTION   Check that the surface profiles do not have sections in common, the profiles must intersect in points.

1975   'Probing has not taken place'

DETECTION   During the execution of a probing cycle.

CAUSE   The probe has not touched the part during the probing movement.

SOLUTION   Correct the definition of the canned cycle.

1976   'Neither ROUGHING nor FINISHING has been programmed'

DETECTION   During the execution of the cycle.

CAUSE   No operation has been defined in the canned cycle.

SOLUTION   The canned cycle must have the definition of the roughing or finishing operation. If an operation does not have a tool, the cycle will not executed it; at least one operation must have a tool.

1977   'Drilling tool radius too large'

DETECTION   During execution.

CAUSE   The drilling tool radius is too large for the geometry of the pocket.

SOLUTION   Correct the value of the tool radius in the tool table or select another tool.

1985   'The probe has not touched the ball'

DETECTION   During execution.

CAUSE   When executing the kinematics calibration cycle, the probe has not touched the ball.

SOLUTION   Check the program. Before executing the cycle, place the probe in the right position, about 10 mm (0.4 inch) over the ball.

1986   'The probe has not touched the part unexpectedly'

DETECTION   During execution.

CAUSE   When executing the kinematics calibration cycle, the probe has touched the ball when it shouldn't.

SOLUTION   Check the program. Before executing the cycle, place the probe in the right position, about 10 mm (0.4 inch) over the ball.

1987   There is no contact with the sphere

DETECTION   During execution.

CAUSE   There was no probe signal when the sphere was expected to be touched.

SOLUTION   Ensure that the kinematics parameters have a maximum error of 1 mm.

1988   'Probe type is not valid (P6)'

DETECTION   During execution.

CAUSE   The selected probe type is not valid.

SOLUTION   Parameter P6 only allows values of 0 and 1.

1989   'This type of kinematics cannot be calibrated'

DETECTION   During execution.

CAUSE   The calibration for this type of kinematics is not implemented.

1990   'There is no contact with the sphere. Ensure that the probe is above the sphere, at a maximum clearance of twice the safety distance'

DETECTION   During execution.

CAUSE   The probe is not located above the sphere or it is too far away.

SOLUTION   The probe must be above of the sphere, at a maximum clearance of twice the safety distance'.

1991   'Parameterization error. The start and end position cannot be the same'

DETECTION   During execution.

CAUSE   The start and final end of the rotary axes are the same.

SOLUTION   The end position of the rotary axes must be different from the start position.

1992 **'The rotary axes of the kinematics must be in their resting position'**

DETECTION   During execution.

CAUSE   The rotary axes are not in their resting position.

SOLUTION   Position the rotary axes into their resting position.

1993 **'The resting position of the rotary axes of the kinematics must be set at zero'**

DETECTION   During execution.

CAUSE   For the machine parameters of the kinematics, the resting position of the axes must be set at 0.

SOLUTION   Set the resting position at zero for the machine parameters.

1994 **The position of the manual rotary axes of the kinematics must be 0 and must be indicated in the variables V.G.POSROTx.**

DETECTION   During execution.

CAUSE   The variables V.G.POSROTF/V.G.POSROTS/V.G.POSROTT/V.G.POSROTO of the manual rotary axes of the kinematics are non-zero.

SOLUTION   The value of these variables must be zero.

1995 **No axes have been chosen for calibration.**

DETECTION   During execution.

CAUSE   No axes have been chosen for calibration.

SOLUTION   Choose an axis.

1996 **The COMPID is being used by another KINID.**

DETECTION   During execution.

CAUSE   The COMPID is being used by another KINID.

SOLUTION   Select another COMPID.

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 programmed profile; the tool will 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 form a loop (the two arcs 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, there are too many motionless blocks (parameter assignments P, variables, etc.) between two motion blocks.

SOLUTION   Reduce the number of motionless blocks programmed; for example, group several of these blocks 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 program has changed the type of tool radius compensation (from G41 to G42 or vice versa) and the next motion block is a circular block.

SOLUTION   The tool radius compensation cannot be changed if the next motion block is an arc. Change the type of radius compensation on a linear tool path.

2007   'While G138 is active, G40 is not allowed after the first compensation block'

DETECTION   During execution.

CAUSE   After activating tool radius compensation with the direct method (G138), a compensation cancellation has been programmed before the first motion block.

SOLUTION   To activate tool radius compensation with the direct method (G138), the CNC needs an additional block in the plane besides the activation block. The CNC allows canceling tool radius compensation after this movement. If this block cannot be programmed, use the indirect method (G139) to activate tool radius compensation.

2008   **'While G138 is active, no G41/G42 change is allowed after the first compensation block'**

DETECTION	During execution.
CAUSE	After activating tool radius compensation with the direct method (G138), a change of compensation type (from G41 to G42 or vice versa) has been programmed before the first motion block.
SOLUTION	To activate tool radius compensation with the direct method (G138), the CNC needs an additional block in the plane besides the activation block. The CNC allows changing the type of tool radius compensation after this movement. If this block cannot be programmed, use the indirect method (G139) to activate tool radius compensation.

2009   **'Too great a difference between the starting and end radius of the arc'**

DETECTION	During execution.
CAUSE	On the programmed arc, the difference between the initial radius and the final radius is greater than the tolerance allowed.
SOLUTION	Check how the arc has been programmed.

2010   **'Tool radius too large when calculating the compensation between arcs'**

DETECTION	During execution.
CAUSE	The tool radius is too large for the programmed arc.
SOLUTION	Use a tool with a smaller radius, eliminate the arch of the profile or program a larger arc.

2011   **'Compensated movement in circular path (damaged profile)'**

DETECTION	During execution.
CAUSE	The tool radius is too large for the programmed arc.
SOLUTION	Use a tool with a smaller radius, eliminate the arch of the profile or program a larger arc.

2012   **An irrelevant transition block has been eliminated.**

DETECTION	During execution.
CAUSE	The radius compensation has removed an unnecessary block.
SOLUTION	Accept the proposed modification and continue with the execution.

2013   **'Different tool radius between previous path and an arc'**

DETECTION	During execution.
CAUSE	When machining an arc while tool radius compensation is active, the tool radius has changed with respect to the previous block.
SOLUTION	Tool radius compensation being active, do not change the tool radius while executing an arc.

2014   **The line and the arc do not intersect: tool radius too large.**

DETECTION	During execution.
CAUSE	The tool radius is too large to machine the part of the indicated profile.
SOLUTION	Use a tool with a smaller radius.

2015   **The arc and the line do not intersect: Tool radius too large.**

DETECTION	During execution.
CAUSE	The tool radius is too large to machine the part of the indicated profile.
SOLUTION	Use a tool with a smaller radius.

2016   **'The profile has been changed to avoid a collision'**

DETECTION	During execution.
CAUSE	The process of detecting collisions during tool compensation has detected a tool path that would 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	The collision detection feature does not admit certain processes like home search, probing, etc.
SOLUTION	Do not activate collision detection if any of these processes has been programmed.

2021   CPU overload during collision detection calculation.

DETECTION	During block preparation.
CAUSE	The model of the machine and/or collisible elements is too complex.
SOLUTION	Simplify the machine model.

2051   'Incompatible spline and tangential control have been programmed'

DETECTION	During execution.
CAUSE	The CNC does not allow activating tangential control while the splines are active and vice versa.
SOLUTION	Deactivate the splines before activating the tangential control and vice versa.

2100   'Positive software limit overrun in G5, G60 or HSC'

DETECTION	During execution.
CAUSE	The path generated by the CNC in G05, G60 or HSC mode is beyond the active travel limits.
SOLUTION	Check that the programmed path is within the software limits and execute it in G7 or G50 to avoid overshooting. If necessary, cancel the HSC mode (#HSC OFF).

2101   'Negative software limit overrun in G5, G60 or HSC'

DETECTION	During execution.
CAUSE	The path generated by the CNC in G05, G60 or HSC mode is beyond the active travel limits.
SOLUTION	Check that the programmed path is within the software limits and execute it in G7 or G50 to avoid overshooting. If necessary, cancel the HSC mode (#HSC OFF).

2102   #ROUNDPAR: parameter too small.

DETECTION	During execution.
CAUSE	The programmed contour error is too small to be processed.
SOLUTION	Program the instruction with a contour error greater than 20 microns.

2103   #ROUNDPAR: wrong parameter.

DETECTION	During execution.
CAUSE	The programmed rounding type is not valid.
SOLUTION	Check the format of the instruction in the programming manual.

2104   #HSC: undetermined initial tangent.

DETECTION	During execution.
CAUSE	The possible causes are: • The point starting the #HSC coincides with the previous point and the tangent remains undetermined. • Only one tangent axis has been programmed.
SOLUTION	Program a different point before the entry point of the #HSC and for both tangent axes.

2105   #HSC: undetermined final tangent.

DETECTION	During execution.
CAUSE	The possible causes are: • The point finalizing the #HSC coincides with the previous point and the tangent remains undetermined. • Only one tangent axis has been programmed.
SOLUTION	Program a different exit point after the end point of the #HSC and for both tangent axes

2106   HSC: Internal error in HSC mode.

DETECTION	During execution.
CAUSE	Error when calculating the tool path to work in HSC mode.
SOLUTION	Cancel the HSC mode in the block or blocks that cause the error.

2108   'HSC: mode change not allowed'

DETECTION	During execution.
CAUSE	The HSC mode cannot be changed without canceling the previous mode.
SOLUTION	Cancel the HSC mode before programming another one.

2109 **'HSC: Contour error too small'**

DETECTION	During execution.
CAUSE	The programmed contour error is too small to be processed.
SOLUTION	Program the instruction with a contour error 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 **SPLINE: Undetermined initial tangent.**

DETECTION	During execution.
CAUSE	Either the starting point of the spline is not the same as the previous point, thus the tangent is undetermined or only one axis of the tangent has been programmed.
SOLUTION	Program a point before the spline entry point or both axes of the tangent.

2113 **SPLINE: Undetermined final tangent.**

DETECTION	During execution.
CAUSE	Either the end point of the spline is not the same as the previous point, thus the tangent is undetermined or only one axis of the tangent has been programmed.
SOLUTION	Program an exit point after the final point of the spline or both axes of the tangent.

2114 **SPLINE: Activation not allowed.**

DETECTION	During execution.
CAUSE	The spline mode cannot be activated without finishing a previous spline.
SOLUTION	Cancel the active spline mode before defining a new one.

2115 **#ASPLINE: wrong parameter.**

DETECTION	During execution.
CAUSE	The type of tangency is wrong.
SOLUTION	Check the correct format in the programming manual.

2116 **SPLINE: Activation error.**

DETECTION	During execution.
CAUSE	A spline mode cannot be activating without canceling the previous mode.
SOLUTION	Cancel the active spline mode before activating another one.

2118 **'SPLINE: Wrong type of spline'**

DETECTION	During execution.
CAUSE	The type of spline programmed is not allowed.
SOLUTION	Check the correct format in the programming manual.

2119 **'SPLINE: The type cannot be changed while splines are active'**

DETECTION	During execution.
CAUSE	A change of spline type has been programmed without canceling the spline.
SOLUTION	Cancel the active spline mode before activating another one.

2121 **'Circular paths (arcs) are not allowed while splines are active'**

DETECTION	During execution.
CAUSE	An arc has been programmed while the spline mode is 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 the spline mode is active.
SOLUTION	Cancel the spline mode before programming the polynomial.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·113·

2123   'Programming not allowed while splines are active'

DETECTION	During execution.
CAUSE	Being the spline mode active, a coordinate system change has been programmed or instructions that interrupt block preparation.
SOLUTION	Cancel the spline mode before changing the coordinate system or program instructions that interrupt block preparation.

2124   'SPLINE: It requires at least two main axes'

DETECTION	During execution.
CAUSE	The spline mode cannot be activated with only one axis in the channel.
SOLUTION	Either configure the channel with two axes or program the movements without spline mode.

2125   'SPLINE: Error when calculating the tangent'

DETECTION	During execution.
CAUSE	Duplicate points have been programmed.
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	Either one of the parameters of the helical path is missing or is wrong. The pitch has been programmed, but not the depth, both the pitch and the 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 the helical path makes full turns, program the pitch and the depth.

2129   'Error in the RTCP transformation'

DETECTION	During execution.
CAUSE	An RTCP transformation has been programmed, but the channel does not have all the necessary axes.
SOLUTION	Before activating the RTCP transformation, configure the channel with all the necessary axes.

2130   'Error in the inclined plane transformation'

DETECTION	During execution.
CAUSE	An inclined plane has been programmed and the channel does not have three axes.
SOLUTION	Before activating the inclined 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 or blocks that caused the error. Contact Fagor Automation.

2132   Machine parameter PROBERANGE not programmed.

DETECTION	During execution.
CAUSE	The parameter PROBERANGE has not be defined.
SOLUTION	Define the parameter PROBERANGE.

2133   The master axis does not exist.

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   The slave axis does not exist.

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   No FACEAXIS has been defined.

DETECTION   During execution.
CAUSE   No face axis has been defined.
SOLUTION   Set machine parameter FACEAXIS = TRUE for the face axis.

2138   No threading axis has been defined.

DETECTION   During execution.
CAUSE   The threading axis is not in the channel.
SOLUTION   Program the threading operation with the available axes or configure the axes of the
                   channel in order to be able to make the thread.

2139   'Null threading movement'

DETECTION   During execution.
CAUSE   No movement has been programmed for the threading axis.
SOLUTION   Program the movement for the threading axis.

2140   SLOPE: Invalid parameter.

DETECTION   During execution.
CAUSE   A parameter of the programming instruction #SLOPE is wrong.
SOLUTION   Check the correct format in the programming manual.

2141   Circular path not allowed without two axes in the main plane.

DETECTION   During execution.
CAUSE   Circular paths are not possible if there aren't at least two axes in the channel.
SOLUTION   Configure the channel with at least two axes.

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 the spline in another point or program another tangent.

2144   'Error when generating the spline'

DETECTION   During execution.
CAUSE   The first block has not been programmed to activate the spline.
SOLUTION   Program a movement before activating the spline.

2145   'End of the program without canceling splines'

DETECTION   During execution.
CAUSE   The CNC has reached the end of the program while the spline mode is active.
SOLUTION   Cancel the spline before finishing the program.

2146   **'The spline cannot be canceled'**

DETECTION	During execution.
CAUSE	Not enough blocks to cancel the spline.
SOLUTION	Do not program the spline to execute a single block.

2147   **'Splines cannot be programmed in HSC mode'**

DETECTION	During execution.
CAUSE	The spline cannot be activated while the HSC mode is active.
SOLUTION	Cancel the HSC mode before activating the spline mode.

2148   **'Too many motionless blocks between blocks in spline mode'**

DETECTION	During execution.
CAUSE	While the spline mode is active, there are too many motionless blocks (parameter assignments P, variables, etc.) between two motion blocks.
SOLUTION	Reduce the number of motionless blocks programmed; for example, group several of these blocks into a single block.

2149   **#TANGCTRL: Internal error in tangential control.**

DETECTION	During execution.
CAUSE	Tangential control is not possible.
SOLUTION	Eliminate the block that caused the error. Contact Fagor Automation.

2150   **#TANGCTRL: Additional block between two polynomials.**

DETECTION	During execution.
CAUSE	The CNC inserts an additional positioning block for the tangential axis between two polynomials.
SOLUTION	Contact Fagor Automation.

2151   **'Incompatible spline and tangential control have been programmed'**

DETECTION	During execution.
CAUSE	Tangential control cannot be activated while the splines mode is active and vice versa.
SOLUTION	Do not program splines while tangential control is active.

2152   **'Unreachable theoretical position in the FACE because the tool is out of alignment.**

DETECTION	During execution.
CAUSE	The C axis cannot reach the position because the tool is misaligned with respect to the turning axis.
SOLUTION	Change the machining path or work with tool alignment.

2153   **'Helical path near the rotation center of the FACE'**

DETECTION	During execution.
CAUSE	A helical tool path of more than one turn has been programmed for an area close to the turning center of the FACE.
SOLUTION	Change the machining tool path.

2154   **Probe delay incorrectly adjusted. Setting parameter PROBEDELAY %f.**

DETECTION	During execution.
CAUSE	The calculated value of PROBEDELAY does not match the value defined in the machine parameter.
SOLUTION	Define the parameter PROBEDELAY using the value indicated in the message. The calibration cycle for the probe (#PROBE 2) can be used to calculate the correct value.

2300   **'#CSROT orientation solution not valid or not found'**

DETECTION	During execution.
CAUSE	The rotary axes of the kinematics could not be oriented to the programmed position in the machine reference system.
SOLUTION	Deactivate function #CSROT or change the orientation.

2301   **#CSROT orientation solution not found according to the criteria set in #DEFROT'**

DETECTION	During execution.
CAUSE	The rotary axes of the kinematics could not be oriented to the programmed position in the machine reference system, according to the criteria selected in #DEFROT.
SOLUTION	Deactivate function #CSROT or change the orientation.

2302   Discontinued #CSROT orientation solution with respect to what has been programmed. Main rotary / Secondary rotary'

DETECTION	During execution.
CAUSE	The orientation of the rotary axes of the kinematics in the machine reference system requires a movement with respect to the programmed value greater than the angle defined in #DEFROT.
SOLUTION	Raise the value of the angle defined in #DEFROT.

2303   'Function CSROT can only be applied to kinematics of two spindle type, table type or spindle-table type axes'

DETECTION	During execution.
CAUSE	The type of kinematics is not compatible with function #CSROT.
SOLUTION	Deactivate function #CSROT.

2304   Program #PWMOUT ON<[FREQ/F value, DUTY/D value]>/OFF

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong or there is more data in the block.
SOLUTION	Check the program.

2305   Parameter PWM out of range.

DETECTION	During execution.
CAUSE	One of the commands of the instruction #PWMOUT has a wrong value.
SOLUTION	Check the program.

2306   Machine parameter PWMOUTPUT is 0.

DETECTION	During execution.
CAUSE	The CNC has attempted to activate the PWM, but it has no local digital output assigned.
SOLUTION	Define which is the local digital output associated with the PWM (parameter PWMOUTPUT) in the machine parameters.

2308   The virtual tool axis transformation, #VIRTAX, has been deactivated.

DETECTION	During execution.
CAUSE	There has been some change in the situation of the axes belonging to the virtual tool axis transformation (#VIRTAX).
SOLUTION	Check the program.

2309   #VIRTAX: Cannot be programmed while the virtual tool axis is active.

DETECTION	During execution.
CAUSE	The CNC has tried to execute one of the following functions while virtual tool axis transformation is active. • Home search; G74. • Modify software limits; G198 - G199. • #OSC instruction.
SOLUTION	First cancel virtual tool axis transformation (#VIRTAX), then execute functions #OSC, G74 or G198-G199, and finally activate #VIRTAX transformation back.

2310   #VIRTAX: The Virtual tool axis must be linear and belong to the channel.

DETECTION	During execution.
CAUSE	The CNC has tried to activate a virtual tool axis (#VIRTAX) and there is none defined (parameter VIRTAXISNAME), the axis is rotary or the axis does not belong to the channel.
SOLUTION	Check the configuration. The virtual axis of the tool must be defined (parameter VIRTAXISNAME), be a linear axis and belong to the channel.

2311   #VIRTAX: The virtual tool axis cannot be primary or be part of a kinematics.

DETECTION	During execution.
CAUSE	The CNC has tried to activate virtual tool axis transformation (#VIRTAX) with an axis that belongs to the main trihedron or to a kinematics.
SOLUTION	Check the program.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·117·

2312 **'#KIN ID[]: Program values between 0 and that of machine parameter NKIN'**

DETECTION   During execution.

CAUSE   The CNC has tried to activate a kinematics that has not been defined in the kinematics table of the machine parameters.

SOLUTION   Check the program.

2313 **Programming not allowed outside a profile PROGTL3 or MACRO.**

DETECTION   During execution.

CAUSE   Functions G810, G811, G813, G820, G821, G840, G841, G842, G850 and G841 are only allowed inside the definition of a profile ProGTL3.

SOLUTION   Program the G function after the beginning of a profile with G841 or G842.

2314 **Programming not allowed inside a profile PROGTL3.**

DETECTION   During execution.

CAUSE   The programmed function is not allowed inside the definition of a ProGTL3 profile.

SOLUTION   Delete that function, replace it with an equivalent one or end the definition of the profile.

2315 **'Error in the PROGTL3 profile resolving process'**

DETECTION   During execution.

CAUSE   Either the programmed ProGTL3 profile is wrong or has no solution.

SOLUTION   Check the program.

2316 **'Unresolved PROGTL3 profile has ended'**

DETECTION   During execution.

CAUSE   The programmed ProGTL3 profile has ended without being fully resolved.

SOLUTION   Check the program.

2317 **'The G809-G820 function cannot have any more points'**

DETECTION   During execution.

CAUSE   Too many points have been programmed in function G809 or G820.

SOLUTION   Check the program.

2318 **'Undefined geometrical entity'**

DETECTION   During execution.

CAUSE   The programmed entity has not been defined or initialized.

SOLUTION   An entity must be defined before it can be used.

2319 **'The definition of the geometrical entity does not admit any more elements'**

DETECTION   During execution.

CAUSE   Too many elements have been programmed when defining a geometrical entity.

SOLUTION   Modify the entity or use several.

2320 **'The element is not tangent'**

DETECTION   During execution.

CAUSE   The indicated element is not tangent to the previous one.

SOLUTION   Check the program.

2321 **'Tangency is not possible'**

DETECTION   During execution.

CAUSE   The programmed element cannot be tangent to the previous one.

SOLUTION   Check the program.

2322 **'The element is not valid'**

DETECTION   During execution.

CAUSE   The element has been wrongly programmed.

SOLUTION   Check the program.

2323 **'The arc is not possible'**

DETECTION   During execution.

CAUSE   The programmed arc is not possible with the given data.

SOLUTION   Check the program.

2324 **'The starting point does not belong to the circle'**

DETECTION   During execution.
CAUSE   The starting point does not belong to the circle.
SOLUTION   Check the program.

2325 **'The final point does not belong to the circle'**

DETECTION   During execution.
CAUSE   The final point does not belong to the circle.
SOLUTION   Check the program.

2326 **'Final point missing in G802/G803'**

DETECTION   During execution.
CAUSE   The final point does not belong to the circle.
SOLUTION   Check the program.

2327 **'G802/G803 XY previous element must be a circle'**

DETECTION   During execution.
CAUSE   The element before a G802/G803 must be a circle.
SOLUTION   Check the program.

2328 **'G808 without previous element'**

DETECTION   During execution.
CAUSE   A G808 has been programmed without previously programming any element.
SOLUTION   Check the program. Program the previous element.

2329 **'G808 undefined previous point'**

DETECTION   During execution.
CAUSE   A G808 has been programmed without previously programming any point.
SOLUTION   Check the program. Program the previous element.

2330 **'G809 undefined previous point'**

DETECTION   During execution.
CAUSE   A G809 has been programmed without previously programming any point.
SOLUTION   Check the program. Program the previous element.

2331 **'Data missing for properly defining the entity'**

DETECTION   During execution.
CAUSE   Data missing when programming the entity.
SOLUTION   Check the program.

2332 **'Invalid entity'**

DETECTION   During execution.
CAUSE   The programmed entity is not valid.
SOLUTION   Check the program.

2333 **'Programming not allowed outside a G8736 macro'**

DETECTION   During execution.
CAUSE   There is a G8737 or G8738 function programmed outside a G8736 macro.
SOLUTION   Check the program. Function G8737 or G8738 must be programmed after the G8736 macro.

2334 **'Programming not allowed outside a G8735 macro'**

DETECTION   During execution.
CAUSE   There is a G8734 function programmed outside a G8735 macro.
SOLUTION   Check the program. Function G8734 must be programmed after the G8735 macro.

2335 **'Programming not allowed outside a G8725 macro'**

DETECTION   During execution.
CAUSE   There is a G8724 function programmed outside a G8725 macro.
SOLUTION   Check the program. Function G8724 must be programmed after the G8725 macro.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

Ref. 2209

2336   'Programming not allowed outside a G8726 macro'

DETECTION	During execution.
CAUSE	There is a G8727 or G8728 function programmed outside a G8726 macro.
SOLUTION	Check the program. Function G8727 or G8728 must be programmed after the G8726 macro.

2337   'PROGTL3 programing not allowed in an ISO profile'

DETECTION	During execution.
CAUSE	The profile of the macro is written in two languages, ProGTL3 and ISO.
SOLUTION	The profiles of the macros may be defined in either ProGTL3 or ISO language, but they cannot be combined in the same profile. Program the profile for the macro in one language only, either ProGTL3 or ISO.

2338   'Section H cannot be programmed'

DETECTION	During execution.
CAUSE	H can only be programmed in the blocks of the plane profile defined in G8736 or as parameter of G8737.
SOLUTION	Check the program. Do not program H in other blocks.

2339   'The active tool radius compensation and 3D compensation are not compatible with each other'

DETECTION	During execution.
CAUSE	The CNC has tried activating tool radius compensation (G41/G42) while 3D tool compensation (#COMP3D ON) was active or vice versa.
SOLUTION	Check the program. The two types of compensation are incompatible with each other.

2340   'Vector defined twice'

DETECTION	During execution.
CAUSE	3D tool compensation. Two vectors N[a,b,c] are programmed in the same block.
SOLUTION	Check the program. Program a single vector per block.

2341   'Vector programmed wrong'

DETECTION	During execution.
CAUSE	3D tool compensation. The vector N[a,b,c] is wrong.
SOLUTION	Check the program. Program the three components of the vector.

2342   'The vector N[-,-,-] must be programmed for 3D compensation'

DETECTION	During execution.
CAUSE	When 3D tool compensation is active, the vector N[a,b,c] is missing in some block.
SOLUTION	Check the program. When 3D tool compensation is active, a vector must be programmed in each motion block.

2343   'Wrong active tool for 3D compensation'

DETECTION	During execution.
CAUSE	The tool is not flat, toric or spherical.
SOLUTION	3D tool compensation in "NORMAL" mode only admits flat, toric or spherical milling tools.

2344   'Program #COMP3D ON/OFF [PARAX/NORMAL]'

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the program.

2345   'Profile defined twice'

DETECTION	During execution.
CAUSE	More than one plane profile or more than a profile have been defined in a section.
SOLUTION	Check the program. Define a single profile

2346   'Too many blocks for 2D display'

DETECTION	During execution.
CAUSE	Either not enough memory reserved for 2D display of the program or more than 500 blocks.
SOLUTION	Modify the program to display it in parts.

2347   'PROGTL3 profile not allowed while G91 is active'

DETECTION	During execution.
CAUSE	A profile cannot be initiated while G91 is active and G91 cannot be programmed inside a G841-G842 profile.
SOLUTION	Program G90 before the beginning of the profile. To program incremental coordinates inside the profiles, use the specific programming for it.

2348   'Macro G8735/G8736/G8725/G8726 defined twice'

DETECTION	During execution.
CAUSE	An attempt has been made to start defining a new macro without finishing the previous one.
SOLUTION	Check the program. Finish the definition of the macros correctly.

2349   'Beginning of a G841/G842 profile defined twice'

DETECTION	During execution.
CAUSE	An attempt has been made to start defining a new profile without finishing the previous one.
SOLUTION	Check the program. Finish the definition of the profiles correctly.

2350   End of macro expected: G8738/G8734/G8728/G8724

DETECTION	During execution.
CAUSE	The CNC has detected the end of the program without finishing the defined macro.
SOLUTION	Check the program. Finish the definition of the macros correctly.

2351   'No profile has been programmed'

DETECTION	During execution.
CAUSE	Some profile has not been programmed in the ProGTL3 macro.
SOLUTION	All the profiles must be defined in order to execute the macro correctly.

2352   'Entity not initialized'

DETECTION	During execution.
CAUSE	The program has attempted to use an uninitialized entity.
SOLUTION	An entity must be initialized before it can be used.

2353   Program #LSFIT [type, pLocal <,data1, data2, data3>]

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the program.

2354   #LSIFT, wrong local parameter.

DETECTION	During execution.
CAUSE	Parameter TYPE of the #LSFIT instruction is not valid.
SOLUTION	Check the program.

2355   'Error in #LSFIT process'

DETECTION	During execution.
CAUSE	Generic internal error of the #LSFIT process.
SOLUTION	Check the program.

2356   'Program #SELECT ORI [HEAD1-2/TABLE1-2, HEAD1-2/TABLE1-2]'

DETECTION	During execution.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the program.

2357   'Selected rotary axes are either the same or the wrong ones'

DETECTION	During execution.
CAUSE	The selected two rotary axes are the same or they're wrong.
SOLUTION	Check the program. The possible values for selecting the rotary axes are HEAD1, HEAD2, TABLE1, TABLE2.

2358   '#DGWZ CYL: The programmed axis must be linear'

DETECTION	During execution.
CAUSE	The axis programmed in the instruction #DGWZ is not linear.
SOLUTION	Select a linear axis as a longitudinal axis.

2359 '#DGWZ CYL: The programmed axis must belong to the plane or to the trihedron'

DETECTION	During execution.
CAUSE	The axis programmed in the instruction #DGWZ does not belong to the plane or to the trihedron.
SOLUTION	Select an axis of the plane or of the trihedron as a longitudinal axis.

2360 'The function or instruction does not allow axis programming'

DETECTION	During execution.
CAUSE	An axis has been programmed in the function or instruction.
SOLUTION	Check the program.

2361 #DGWZ <RECT>: The rotation axis cannot be programmed.

DETECTION	During execution.
CAUSE	The axis of rotation has been programmed in DGWZ or DGWZ RECT.
SOLUTION	A rotation axis can only be programmed for DGWZ CYL. Remove the rotation axis from the DGWZ or DGWZ RECT programming.

2362 #DGWZ CYL: Rotation axis not programmed.

DETECTION	During execution.
CAUSE	For DGWZ CYL, it is necessary to program a rotation axis that has not been programmed.
SOLUTION	Program the axis of rotation for DGWZ CYL.

2363 G8/G9: End point of the arc incorrectly programmed.

DETECTION	During execution.
CAUSE	For functions G8 and G9, the end point of the arc is required.
SOLUTION	Angle of the end point of the arc.

2364 End point of non-existent movement.

DETECTION	During execution.
CAUSE	The coordinate programmed by a Cartesian coordinate and an angle is incorrect. The tool path beginning from the previous point cannot reach the programmed point.
SOLUTION	Correctly program the movement end point.

2365 G34: Dual leadscrew pitch increment/decrement programming.

DETECTION	During execution.
CAUSE	The increment/decrement of the leadscrew pitch (K1) has been programmed more than once
SOLUTION	Program the leadscrew pitch increment/decrement only once for the block.

2366 G34: Leadscrew pitch increment/decrease equal to zero.

DETECTION	During execution.
CAUSE	The leadscrew pitch increment/decrement (K1) for G34 is zero.
SOLUTION	Program the leadscrew pitch increment/decrement (K1) for G34.

2367 G34: Leadscrew pitch increase/decrease is out of range.

DETECTION	During execution.
CAUSE	The leadscrew pitch increment/decrement value for G34 is incorrect.
SOLUTION	Program a correct leadscrew pitch increment/decrement (K1).

2368 G34: Leadscrew pitch increment/decrease is too large.

DETECTION	During execution.
CAUSE	A leadscrew pitch increment has been programmed greater than twice the thread pitch.
SOLUTION	Program a correct leadscrew pitch increment (K1).

2369 G34: Negative or zero final ballscrew pitch.

DETECTION	During execution.
CAUSE	With the leadscrew pitch and the leadscrew pitch decrement programmed for G34, the final pitch calculated by the control is negative or zero.
SOLUTION	Correctly program the pitch and leadscrew pitch decrement for G34

2370   Thread axis non-existent.

DETECTION   During execution.

CAUSE   The thread axis is defined by the letter used to program the leadscrew pitch for the block. The programmed axis does not exist.

SOLUTION   Program the leadscrew pitch for a valid axis.

2371   #LSFIT, algorithm does not converge, check the input data.

DETECTION   During execution.

CAUSE   Kinematics calibration. The cycle cannot finish the adjustment with the palpation or input data.
          Correct input data or number and/or positions of palpations.

2372   #LSFIT, minimum amount of data exceeded.

DETECTION   During execution.

CAUSE   Kinematics calibration. The number of palpations or input data is greater than the minimum allowed.

SOLUTION   Correct input data or number and/or positions of palpations.

2373   #LSFIT, insufficient amount of data.

DETECTION   During execution.

CAUSE   Kinematics calibration. The cycle cannot finish the adjustment with the palpation or input data.

SOLUTION   Correct input data or number and/or positions of palpations.

2374   #LSFIT, calculation error (radius=0), check input data.

DETECTION   During execution.

CAUSE   Kinematics calibration. The cycle cannot finish the adjustment with the palpation or input data.

SOLUTION   Correct input data or number and/or positions of palpations.

2375   #LSFIT, calculation error (SVD), check input data.

DETECTION   During execution.

CAUSE   Kinematics calibration. The cycle cannot finish the adjustment with the palpation or input data.

SOLUTION   Correct input data or number and/or positions of palpations.

2376   #LSFIT, calculation error (SVBK), check input data.

DETECTION   During execution.

CAUSE   Kinematics calibration. The cycle cannot finish the adjustment with the palpation or input data.

SOLUTION   Correct input data or number and/or positions of palpations.

2377   #LSFIT, the instruction is invalid.

DETECTION   During execution.

CAUSE   The #LSFIT instruction has not been programmed correctly.

SOLUTION   Check the program.

2378   #GAPCTRL: The axis or GAP needs to be programmed.

DETECTION   During execution.

CAUSE   For the command #GAPCTRL, the axis or gap has not been programmed.

SOLUTION   Program the axis and/or the GAP.

2379   #GAPCTRL: Machine parameter GAPANAINID is zero or unassigned.

DETECTION   During execution.

CAUSE   The analog input to which the gap sensor is associated (parameter GAPANAINID) has not been defined.

SOLUTION   Assign the correct value to the machine parameter GAPANAINID.

2380   #LSFIT, internal error.

DETECTION   During execution.

CAUSE   Kinematics calibration. Inconsistent data in the cycle.

SOLUTION   Correct input data or number and/or positions of palpations.

2381 #ABORT OFF not programmed in the same subroutine as #ABORT.

DETECTION   During execution.

CAUSE   The command #ABORT OFF is not programmed. If the command #ABORT is only programmed in the block, when aborting the program, the execution continues for the command #ABORT OFF.

SOLUTION   Program #ABORT OFF.

2382 #ISO ON[]. Incorrect syntax.

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

2383 #ISO ON[]. Invalid filename.

DETECTION   During execution.

CAUSE   The destination folder does not exist or the file name is incorrect.

SOLUTION   Select an available folder and a correct file name.

2384 Movement using active modal cycle and call to a subroutine in the same block.

DETECTION   During execution.

CAUSE   A subroutine call and a modal cycle are scheduled in the same block, or an attempt was made to execute a canned cycle during the execution of another canned cycle.

SOLUTION   Check the program. Cancel the modal cycle or schedule the subroutine call in a later block. It is not possible to execute a canned cycle within another canned cycle.

2385 Work zone not programmed.

DETECTION   During block preparation.

CAUSE   The zone number has not been programmed in G120/G121/G122/G123.

SOLUTION   Program the zone number in parameter "K"; from 1 to 5.

2386 Work zone does not exist.

DETECTION   During block preparation.

CAUSE   The zone number programmed in G120/G121/G122/G123 does not have the right value.

SOLUTION   Program the zone number in parameter "K"; from 1 to 5.

2387 Tool monitoring not valid in work zone.

DETECTION   During block preparation.

CAUSE   Tool monitoring in the work zone does not have the right value. There are 2 possibilities.
- The value programmed in G122 is wrong.
- The value restored after some data restoring process (startup, reset, tool inspection) is wrong.

SOLUTION   Program the right value for tool monitoring; from 0 to 2.
- In the first case, program a correct value in parameter "I"of function G122.
- In the second case, write from the PLC the right value in the variable "G.ZONETOOLWATCH[zone]".

2388 Invalid work zone status.

DETECTION   During block preparation.

CAUSE   The work zone status does not have the right value. There are 2 possibilities.
- The value programmed in G122 is wrong.
- The value restored after some data restoring process (startup, reset, tool inspection) is wrong.

SOLUTION   Program the right value for the work zone; from 0 to 2.
- In the first case, program a correct value in parameter "E"of function G122.
- In the second case, write from the PLC the right value in the variable "G.ZONEST[zone]".

2389 'The radius can only be programmed with R'

DETECTION   During block preparation.

CAUSE   The programmed instruction only lets program the radius with "R"; neither with "R1" nor with G263.

SOLUTION   Program the radius with "R".

FAGOR ⬤
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·**124**·

2390   'The radius has not been programmed'

DETECTION   During block preparation.
CAUSE   The radius has not been programmed.
SOLUTION   Program the radius.

2391   'Circular zone: The center must be programmed with two axes of the channel'

DETECTION   During block preparation.
CAUSE   In functions G123, the center of the circular zone has not been programmed correctly.
SOLUTION   The center of the circular zone G123 must be defined with two axes of the channel. Function G123 only admits two axes that must be different and belong to the channel where the zone is activated.

2392   Linear zone: The lower limit must be smaller that than the upper limit for each axis'

DETECTION   During block preparation.
CAUSE   The CNC has tried to enable a work zone where the lower limit of an axis is greater than the upper limit.
SOLUTION   In a linear zone, the lower limit of the axis must be smaller than the upper limit. Change the lower limit (G120) or the higher limit (G121) of the axis. Checking is not carried out if the zone is disabled.

2393   'E function programmed twice'

DETECTION   During block preparation.
CAUSE   The "E" parameter has been programmed more than once in the same block.
SOLUTION   Program the "E" parameter only once in the block.

2394   Work zone status not programmed.

DETECTION   During block preparation.
CAUSE   The work zone status has not been programmed in G122.
SOLUTION   Program the work zone status in parameter "E"; from 0 to 2.

2395   'The center cannot be programmed with module-type rotary axes'

DETECTION   During block preparation.
CAUSE   The center cannot be programmed with module-type rotary axes.
SOLUTION   The center cannot be defined with module-type rotary axes. Check the program.

2396   #GAPCTRL programmed in another channel.

DETECTION   During execution.
CAUSE   An attempt has been made to activate the gap control on a channel while it is active on another channel.
SOLUTION   Only a gap control can be active in the CNC.

2397   Active #GAPCTRL ON. #GAPCTRL OFF required to change parameters.

DETECTION   During execution.
CAUSE   An attempt was made to change the parameters for #GAPCTRL while it was active.
SOLUTION   Deactivate #GAPCTRL before changing its parameters.

2398   #LEAP: requires the POSLIMIT to be programmed.

DETECTION   During execution.
CAUSE   The POSLIMIT command is missing in the #LEAP command.
SOLUTION   When programming #LEAP for the first time, the command POSLIMIT must be programmed.

2399   Active #LEAP. POS, final position of the leap, cannot be altered.

DETECTION   During execution.
CAUSE   Once the end value of the leap has been programmed (#LEAP[POS]), it cannot be changed with active #LEAP.
SOLUTION   Check the program.

2400   Logic number of the axis out of range.

DETECTION   During execution.
CAUSE   The logical number programmed in the variable is not correct.
SOLUTION   Program a correct logical axis; from 1 to the maximum number of axes (parameter NAXIS).

2401   '#FREE/SET AX: An axis that defines an active work zone cannot be freed (released)'

DETECTION	During block preparation.
CAUSE	The CNC has tried to remove from the configuration an axis that is part of an active work zone.
SOLUTION	Deactivate the work zone before freeing (releasing) the axis.

2402   '#PARK: An axis that defines an active work zone cannot be parked'

DETECTION	During block preparation.
CAUSE	The CNC has tried to park an axis that is part of an active work zone.
SOLUTION	Deactivate the work zone before parking the axis.

2403   'A zone cannot be activated while one of its axes is parked'

DETECTION	During block preparation.
CAUSE	The CNC has tried to activate a work zone; but one of the axes that define it is parked.
SOLUTION	Unpark the axis before activating the work zone.

2404   #GAPCTRL: The index has been changed for the controlled axis channel.

DETECTION	During block preparation.
CAUSE	The possible causes are: • When the active gap control (#GAPCTRL ON) is on, an instruction is programmed that changes the index for that axis in the channel (#SET AX, #CALL AX, #FREE AX). • If the machine parameter GAPCTRLCANCEL is set to "NO", the axis remains controlled after the program has finished or following a reset. This error persists if the axis index has been changed under these conditions.
SOLUTION	It is not possible to modify the position of the axes in the channel with the active gap control.

2405   Movement block canceled because an axis has #GAPCTRL.

DETECTION	During tool inspection or MDI mode.
CAUSE	An attempt has been made to move an axis in MDI that is being controlled by the command #GAPCTRL.
SOLUTION	Program #GAPCTRL OFF.

2406   #LEAP with #GAPCTRL ON: The leap must start and end at the same coordinate.

DETECTION	During block preparation.
CAUSE	There is an axis controlled by the instruction #GAPCTRL ON. In this circumstance, the #LEAP instruction has been programmed to execute a leap where the start and end coordinates (POS parameter for the #LEAP instruction) do not match.
SOLUTION	The possible solutions are: • Do not program POS for the #LEAP command • Remove the gap control (#GAPCTRL OFF).

2407   #LEAP with #GAPCTRL ON: POSLIMIT less than initial leap coordinate.

DETECTION	During block preparation.
CAUSE	There is an axis controlled by the instruction #GAPCTRL ON. In this circumstance, the #LEAP instruction has been programmed with a lower POSLIMIT (maximum leap value) than the start and end leap coordinates.
SOLUTION	The possible solutions are: • Program POSLIMIT for the #LEAP command with a coordinate greater than the initial leap position. • Remove the gap control (#GAPCTRL OFF).

2408   #LEAP, #GAPCTRL ON #SWTOUT ON: All three commands cannot be active.

DETECTION	During block preparation.
CAUSE	The commands #LEAP, #GAPCTRL ON and #SWTOUT ON cannot be active at the same time.
SOLUTION	Deactivate one of the other two active commands.

2409   The slave axis "LASER FOLLOW AXIS" cannot be programmed.

DETECTION	During block preparation.
CAUSE	The slave axis name for #FOLLOW ON [MASTER, slave axis] coincides with the machine parameter LASERFOLLOWAXIS.
SOLUTION	Do not program the slave axis for #FOLLOW [MASTER].

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·126·

2410   **#PWRCTRL   ON/OFF   [<DUTY/D/OUT/O>   <,OVRMINvalue>   <,OVRMAXvalue> <,FMINvalue> <,FMAXvalue>]**

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

2411   **#PWRCTRL OFF<[DUTY/D/OUT/O]>**

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

2412   **Parameter PWRCTRL out of range.**

DETECTION	During editing and execution.
CAUSE	A command in the instruction #PERCTRL has an invalid value.
SOLUTION	Check the syntax of the instruction in the programming manual.

2413   **DUTY/OUT must be programmed.**

DETECTION	During editing and execution.
CAUSE	The command DUTY/OUT is not programmed for the function #PWRCTRL.
SOLUTION	Check the syntax of the instruction in the programming manual. To activate the power control (#PWRCTRL), one of the commands DUTY or OUT must be programmed.

2414   **PWCTRL DUTY not allowed if there is no PWM software option.**

DETECTION	During execution.
CAUSE	To activate the PWM duty control, the "PWM" software option is required.
SOLUTION	Activate the power control (command OUT) to acquire the appropriate software option.

2415   **PWCTRL OUT not allowed without master spindle in the channel.**

DETECTION	During execution.
CAUSE	The power control is applied to the master spindle. An attempt was made to activate the power control and there is no master spindle active.
SOLUTION	Select a master spindle (# MASTER) before activating the power control.

2416   **#PWRCTRL ON [<DUTY/D/OUT/O> <,OVRMINvalue> <,OVRMAXvalue> <,FMINvalue> <,FMAXvalue>]**

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

2417   **#PWRCTRL ON: FMIN must be less than FMAX.**

DETECTION	During execution.
CAUSE	For the function #PWRCTRL, the minimum feedrate is greater than the maximum feedrate.
SOLUTION	The minimum feedrate (FZMIN) must be less than the maximum feedrate (FZMAX).

2418   **#PWRCTRL ON: OVRMIN must be less than OVRMAX.**

DETECTION	During execution.
CAUSE	For the function #PWRCTRL, the minimum override is greater than the maximum override.
SOLUTION	The minimum override (OVRMIN) must be less than the maximum override (OVRMAX).

2419   **#PWRCTRL ON: the first time, all parameters must be programmed.**

DETECTION	During execution.
CAUSE	The function #PWRCTRL has been programmed for the first time and a parameter is missing.
SOLUTION	The function #PWRCTRL has been programmed for the first time and all parameters must be programmed.

2420   #FOLLOW ON[MASTER,]. The slave axis cannot be the slave of another master.

DETECTION	During execution.
CAUSE	There are 2 possibilities.

- The slave axis name for #FOLLOW ON [MASTER, slave axis] coincides with the machine parameter LASERFOLLOWAXIS.
- The slave axis for the synchronization is already a slave for another synchronization.

SOLUTION	Do not program #FOLLOW ON with the slave axis appearing in the error message, or previously make a #FOLLOW OFF for the same slave axis. An axis cannot be the slave for two synchronizations.

2421   The master spindle cannot be changed or parked while PWRCTRL is active.

DETECTION	During execution.
CAUSE	The master spindle cannot be changed or parked while PWRCTRL is active.
SOLUTION	The power control is applied to the master spindle. To change or park the master spindle, deactivate the power control (#PWRCTRL).

2422   #TFOLLOW ON [MASTER] is not permitted. Permitted: #FOLLOW ON [MASTER]

DETECTION	During block preparation.
CAUSE	When an axis is programmed to be slave for the variable FLWMASTER, the synchronization can only done using real coordinates; it is not possible to synchronize theoretical coordinates.
SOLUTION	Perform a synchronization with real coordinates.

2423   Undefined #CUTTING or #PIERCING type.

DETECTION	During block preparation.
CAUSE	The type of piercing or cutting is zero. The parameter T is missing in the instruction and a T (type) value has not been previously programmed.
SOLUTION	Activate a technological table, program the material command or program the command #PIERCING or #CUTTING generating the error with a correct T parameter (value between 1 and 10).

2424   No active technological table (#MATERIAL).

DETECTION	During block preparation.
CAUSE	The command #PIERCING and #CUTTING have been programmed and there is no technological table active.
SOLUTION	Activate a technological table from the HMI or program the command #MATERIAL.

2425   #PLC[PLCmark=0/1,PLCmark=0/1,...]

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

2426   #PLC: non-modifiable mark.

DETECTION	During editing and execution.
CAUSE	The mark programmed in the #PLC instruction is not modifiable.
SOLUTION	The instruction #PLC can only activate or deactivate modifiable tags.

2427   #DMC ON[<PWRSP value><,OVRMIN value><,OVRMAX value><,FZMIN value><,FZMAX value>]

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

2428   #DMC OFF

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

2429   Parameter DMC out of range

DETECTION	During editing and execution.
CAUSE	A command in the instruction #DMC has an invalid value.
SOLUTION	Check the syntax of the instruction in the programming manual.

2430 #DMC ON: FZMIN must be less than FZMAX.

DETECTION   During execution.
CAUSE   For the function #DMC, the minimum feedrate is greater than the maximum feedrate.
SOLUTION   The minimum feedrate (FZMIN) must be less than the maximum feedrate (FZMAX).

2431 #DMC ON: OVRMIN must be less than OVRMAX.

DETECTION   During execution.
CAUSE   For the function #DMC, the minimum override is greater than the maximum override.
SOLUTION   The minimum override (OVRMIN) must be less than the maximum override (OVRMAX).

2432 #DMC ON: Programming not allowed with analog spindle.

DETECTION   During execution.
CAUSE   The CNC has attempted to activate DMC on an analog spindle.
SOLUTION   DMC can only be activated on digital spindles.

2433 ON/OFF must be programmed.

DETECTION   During execution.
CAUSE   The command ON/OFF is not programmed to activate or deactivate the function.
SOLUTION   Check the syntax of the instruction in the programming manual. Include the command ON to activate the function (for example, #DMC ON [...]) or the OFF command to cancel it (for example, #DMC OFF).

2434 Programming not allowed outside a G8077 macro.

DETECTION   During execution.
CAUSE   There is a G8078 function programmed outside a G8077 macro.
SOLUTION   Check the program. Functions G8078 must be programmed after the macro G8077.

2435 Programming not allowed outside a G8079 macro.

DETECTION   During execution.
CAUSE   There is a G8078 function programmed outside a G8079 macro.
SOLUTION   Check the program. Functions G8078 must be programmed after the macro G8079.

2436 Programming not allowed outside a G8777 macro.

DETECTION   During execution.
CAUSE   There is a G8778, G8701 or G8702 function programmed outside a G8777 macro.
SOLUTION   Check the program. Function G8778, G8701 or G8702 must be programmed after the G8777 macro.

2437 The profile of the G8077 macro is polygonal.

DETECTION   During execution.
CAUSE   The profile must be constructed with straight sections.
SOLUTION   Modify the profile.

2438 #LEAP [GAP] XY without previously programming #GAPCTRL ON.

DETECTION   During block preparation.
CAUSE   A leap has been programmed using the command #LEAP, which must finish at a certain distance along the plate (GAP parameter). The command #GAPCTRL ON is not programmed. It may be that the GAP parameter is not programmed for the block where the error occurs, but rather a previous #LEAP.
SOLUTION   The possible solutions are:
                   • Program #GAPCTRL ON before the #LEAP where error is detected
                   • Remove the parameter GAP from the #LEAP commands.

2439 Program the node number after the ".".

DETECTION   During execution.
CAUSE   The drive variable (DRV) associated with the Sercos node (drive or RCS-S) is incorrect. The node has not been programmed.
SOLUTION   Program a node between 1 and 32.

2440   Non-existent node.

DETECTION	During execution.
CAUSE	The drive variable (DRV) associated with the Sercos node (drive or RCS-S) is incorrect. The node does not exist.
SOLUTION	Program a node between 1 and 32.

2441   #FINALSUB programmed twice.

DETECTION	During block preparation.
CAUSE	The command #FINALSUB has been programmed more than once in the main program. Only a single programming is permitted. Within loops (There is no error for $IF, $FOR, etc. or in subroutines).
SOLUTION	Program #FINALSUB just once.

2442   Programming not allowed outside a G8754 macro.

DETECTION	During execution.
CAUSE	There is a G8753 function programmed outside a G8754 macro.
SOLUTION	Check the program. Functions G8753 must be programmed after the macro G8754.

2443   The CNC must be restarted to assume the changes to the materials table.

DETECTION	During the execution of the command #MATERIAL or when activating/validating a technology table.
CAUSE	The format of the table has been changed (units, new variables, listed range, etc.). A restart is required for the CNC to utilize the new format.
SOLUTION	Restart the CNC.

2444   #OPEN/#CLOSE/#WRITE: file handle value out of range.

DETECTION	During block preparation.
CAUSE	File identifier (command F) is wrong.
SOLUTION	The file identifier (command F) must have a value between 1 and 4.

2445   #OPEN/#CLOSE/#WRITE: programmed twice.

DETECTION	During block preparation.
CAUSE	A parameter in the instruction has been programmed more than once.
SOLUTION	Program each parameter in the instruction only once.

2446   Non-existent variable prefix

DETECTION	During execution.
CAUSE	The prefix of the variable does not exist.
SOLUTION	Check the documentation for the list of allowed variables.

2447   #PARALAX: The parallel axes must belong to the same channel.

DETECTION	During execution.
CAUSE	The programmed axes do not belong to the same channel.
SOLUTION	The programmed axes must belong to the same channel.

2448   Cannot be programmed while the transformation #PARALAX is active.

DETECTION	During execution.
CAUSE	The CNC does not allow executing the programmed function if parallel axes transformation is active.
SOLUTION	Cancel parallel axes transformation to execute the rest of the functions.

2449   The axis cannot be activated as Hirth axis because it is part of the #PARALAX transformation.

DETECTION	During execution.
CAUSE	Any of the programmed axes are Hirth axes.
SOLUTION	The programmed axes cannot be Hirth.

2450   #PARALAX: The axes of the active transformation cannot be excluded.

DETECTION	During execution.
CAUSE	An instruction #SET AX or #FREE AX has attempted to change the axis included in a transformation of parallel axes.
SOLUTION	Cancel the kinematics before changing the axes of the channel.

CNCelite
8058   8060
8065   8070

2451 Parallel axis transformation #PARALAX has been deactivated.

DETECTION	During execution.
CAUSE	The possible causes are:

- The axes required for the transformation are not suitable.
- A home search has been attempted with the function #PARALAX active.

SOLUTION	The possible solutions are:

- The axes of a transformation must be linear, belong to the channel, unparked and not slaves of a coupling or gantry axis.
- Deactivate the transformation before doing a home search.

2452 The axes of the parallel transformation, #PARALAX, must be linear.

DETECTION	During execution.
CAUSE	Any of the programmed axes are rotary axes.
SOLUTION	The programmed axes must be linear.

2453 "}" was expected, vector end programming.

DETECTION	During block preparation.
CAUSE	Closing parenthesis "}" missing in the programming of a vector.
SOLUTION	Check the program. Program the vector in the form V {i, j, k}.

2454 No memory for user variables associated with kinematics calibration.

DETECTION	During block preparation.
CAUSE	The user variables associated with the kinematics calibration only exist if the "Kinematics calibration" software option is active.
SOLUTION	Contract the corresponding software option.

2455 Variable tree management error.

DETECTION	During block preparation.
CAUSE	Error creating a new user variable.
SOLUTION	Contact Fagor Automation.

2456 While #GAPPOS is active, the first three axes cannot be modified.

DETECTION	During block preparation.
CAUSE	An attempt has been made to change the first three axes on the channel (#FREE AX or #SET AX) while the instruction #GAPPOS was still active.
SOLUTION	Do not change the first three axes on the channel before the #GAPPOS or #GAPCTRL instruction is active.

2457 #GAPPOS: too many parameters.

DETECTION	During block preparation.
CAUSE	More than three axes in the instruction #GAPPOTS have been defined.
SOLUTION	Only the first three axes in the channel need to be defined in the instruction #GAPPOS.

2458 #DMC ON: The master spindle must use the parameter DMCSPDL = Yes.

DETECTION	During execution.
CAUSE	An attempt was made to activate the DMC for a spindle that does not have permission to do so.
SOLUTION	To activate the DMC on a spindle, then it must have the machine parameter set to DMCSPDL=Yes.

2459 #OPEN: parameter value TYPE not valid.

DETECTION	During execution.
CAUSE	The command TYPE has a wrong value.
SOLUTION	Check the programming of the #OPEN instruction.

2461 The #GAPCTRL axis, cannot be a part of the transformed axis.

DETECTION	During execution.
CAUSE	The axis for gap control (#GAPCTRL) coincides with some axis of the #CS/#ACS, #RTCP, #ANGAX or #VIRTAX transformations.
SOLUTION	Program the gap control with a simulated axis.

2462   The programmed axis must belong to the plane or trihedron.

DETECTION   During execution.

CAUSE   The axis does not belong to the principal plane or trihedron.

SOLUTION   Check the program.

2463   #LEAP incompatible with GAP3D.

DETECTION   During execution.

CAUSE   The #LEAP command does not support multi-axis gap control.

SOLUTION   Do not program #LEAP or perform gap control on a surface.

2464   Movement not allowed for Piercing selection with G92. Program #MOVE.

DETECTION   During execution.

CAUSE   In the piercing selection, the option to repeat the program at the same point as the previous run has been chosen, and the program performs a pre-selection (G92) in the initial blocks. The program has a previous movement block in #INITIALSUB or before the first G1 of the programme.

SOLUTION   Perform another piercing selection or replace the movement with an independent movement (#MOVE).

2465   #RPT: the N label and the number of repetitions must be whole numbers.

DETECTION   During execution.

CAUSE   In the #RPT instruction a non-integer value is programmed for the label N or for the number of repetitions.

SOLUTION   The value of the N label and the number of repetitions must be integers.

2466   Subroutine name expected.

DETECTION   During execution.

CAUSE   A local subroutine (%L) has been programmed without defining its name.

SOLUTION   Define the start of the local subroutine with "%L name".

2467   Existing local subroutine.

DETECTION   During execution.

CAUSE   More than one local subroutine (%L) with the same name is defined.

SOLUTION   No two local subroutines in a program can have the same name.

2468   COROT=FIX programming not allowed in spindle-table kinematics, TYPE=13..16.

DETECTION   During execution.

CAUSE   An RTCP without rotation of the coordinate system has been programmed for a spindle+table kinematics (TYPE=13..16).

SOLUTION   This type of kinematics does not support such programming. Program the RTCP with coordinate system rotation or redefine the kinematics as vector (TYPE 52).

2469   HEAD programming not allowed in spindle-less kinematics.

DETECTION   During execution.

CAUSE   An RTCP has been programmed with the HEAD option for spindle-less kinematics.

SOLUTION   Check the program.

2470   TABLE or COROT programming not allowed in non-table kinematics.

DETECTION   During execution.

CAUSE   An RTCP has been programmed with the option TABLE or COROT for a kinematics without a table.

SOLUTION   Check the program.

2471   It is mandatory to program the name of the material file and/or the selected table.

DETECTION   During execution.

CAUSE   The #TECHTABLE instruction has been programmed without indicating the material file or the selected table.

SOLUTION   Check the program. Both the material file and the technology table are optional, but one of them should be programmed.

2472   There is no active material file.

DETECTION   During execution.

CAUSE   The #TECHTABLE instruction has been programmed without indicating the material file, and there is no active material file.

SOLUTION   Activating a material file. The programming of the material file is only optional when a material file is active.

2473   Set number out of range.

DETECTION   During execution.

CAUSE   The set number (parameter group) programmed in the #TECHTABLE instruction is invalid or does not exist.

SOLUTION   Check the program. Check that the programmed parameter group exists in the table. The valid range is 1..256.

2474   Non-existent variable without an active material that has a Piercing table.

DETECTION   During execution.

CAUSE   An attempt was made to read a "TTPIR.name" variable but there is no active material file, or the active file has a Piercing table.

SOLUTION   Activate a material file that has a Piercing table.

2475   Non-existent variable without an active material that has a Cutting table.

DETECTION   During execution.

CAUSE   An attempt has been made to read a "TTCUT.name" variable but there is no active material file, or the active file has a Cutting table.

SOLUTION   Activate a material file that has a Cutting table.

2476   Table does not exist in the active material file.

DETECTION   During execution.

CAUSE   A technology table has been programmed which does not exist in the active material file.

SOLUTION   Review the programming and what tables exist.

2477   Table name not allowed.

DETECTION   During execution.

CAUSE   The #TECHTABLE instruction tried to activate the Piercing and Cutting tables.

SOLUTION   The #TECHTABLE instruction allows you to activate a generic technology table. This instruction does not allow the Piercing and Cutting tables to be loaded, which are managed with the #PIERCING and #CUTTING instructions.

2478   Instruction not allowed without PIERCING table in the active material file.

DETECTION   During execution.

CAUSE   The #PIERCING instruction has been programmed but the active material file has no Piercing table.

SOLUTION   Upload a material file with Piercing table.

2479   Instruction not allowed without CUTTING table in the active material file.

DETECTION   During execution.

CAUSE   The #CUTTING instruction has been programmed but the active material file has no cutting table.

SOLUTION   Load a material file with cutting table.

2480   START not allowed to continue profile due to invalid data.

DETECTION   Pressing [START] after selecting a partial continuation mode of the previous machining; "Restart point", "Restart PP", "Interrupted PP" or "BreakPP".

CAUSE   Error in the file that saves the data to continue.

SOLUTION   Repeat the process (program interruption and select continuation type). If the error persists, contact Fagor Automation.

2481   Invalid TIP value.

DETECTION   During execution.

CAUSE   The #KIN ID instruction has been programmed with an invalid TIP value.

SOLUTION   Check the program. The supported range of values for TIP is 0/1.

2482 Invalid origin number.

DETECTION	During execution.
CAUSE	The #KINORG instruction has been programmed with an invalid origin.
SOLUTION	Check the program. The allowed range of values for the origin is 1..99.

2483 #KINORG: active origin is not supported.

DETECTION	During execution.
CAUSE	In the #KINORG instruction, the active origin is not allowed to be programmed.
SOLUTION	Check the program. Program an origin other than the one that is active. The allowed range of values for the origin is 1..99.

2484 Rotational axis movement not allowed with active KIN ID TIP and active radius compensation (G41/G42)

DETECTION	During execution.
CAUSE	An attempt has been made to move a rotational axis that affects the tool position, and #KIN ID[TIP] and radius compensation (G41/G42) are active.
SOLUTION	Deactivate the radius compensation to move the rotary axis.

2485 It is not possible to program linear and rotary axis in the same block, with active KIN ID TIP.

DETECTION	During execution.
CAUSE	An attempt has been made to move a linear axis next to a rotary axis that affects the tool position, and #KIN ID[TIP] is active.
SOLUTION	Move the rotary axis and the linear axis individually.

2486 Spindle #RENAME is not supported in C-axis mode.

DETECTION	During execution.
CAUSE	An attempt has been made to rename the working spindle as C-axis.
SOLUTION	Renaming the active axis as C-axis is not allowed.

2487 The CNC has tried to activate #FACE/#CYL with an incorrect kinematics type.

DETECTION	During execution.
CAUSE	The kinematics programmed in the #FACE/#CYL instructions are not of the right type.
SOLUTION	Activate #FACE/#CYL without kinematics or with 41/42/43 kinematics.

2488 Program #LOAD GAPTABLE before #GAPCTRL ON or #LEAP.

DETECTION	During execution.
CAUSE	A non-linear gap calibration has been defined for each machine parameter. An attempt has been made to use the gap probe (#GAPCTRL ON or #LEAP) without activating the gap table.
SOLUTION	Activate a gap calibration table using the #LOAD GAPTABLE command.

2489 It is not possible to program #RTCP in INSPECTION, with KINID TIP active.

DETECTION	During execution.
CAUSE	During tool inspection, an attempt was made to redo the coordinate system in a different order than the one you activated.
SOLUTION	Maintain #KIN ID [TIP] or #RTCP in the system during tool inspection.

2490 #RTCP ON/OFF: Programming not allowed while CS/ACS is active.

DETECTION	During execution.
CAUSE	The RTCP and CS/ACS activation and deactivation sequence in the kinematics of tables without rotation of the coordinate system is fixed; #RTCP ON, #CS/ACS ON, #CS/ACS OFF, #RTCP OFF.
SOLUTION	Check the program. Program the correct sequence.

2491 Programming not allowed inside the subroutine associated with #CAX.

DETECTION	During execution.
CAUSE	The #CAX instruction has been programmed with call parameters, within the subroutine associated with #CAX.
SOLUTION	Check the program. Within the subroutine, program #CAX without parameters.

FAGOR AUTOMATION

CNCelite
8058 8060
8065 8070

REF. 2209

·134·

2492   Wildcard not allowed outside the subroutine associated with #CAX.

DETECTION   During execution.

CAUSE   The @S or @C wildcard has been programmed outside the subroutine associated with the #CAX instruction (CAXSUB machine parameter).

SOLUTION   Check the program. The use of these wildcards is not allowed outside the subroutine associated with #CAX.

2493   #FACE/CYL: "Q", C-axis origin programming, in degrees, is expected.

DETECTION   During execution.

CAUSE   The origin of the C axis has not been programmed in the #FACE CYL instruction.

SOLUTION   Check the program.

2494   #DMC ON [<PWRSP> <,OVRMIN> <,OVRMAX> <,FZMIN> <,FZMAX> <,ENTRYOVR> <,ENTRYDIST>]

DETECTION   During execution.

CAUSE   Syntax error in the instruction.

SOLUTION   Check the program.

2495   #GRAPHCMD command <Pi>.

DETECTION   During execution.

CAUSE   Syntax error in the instruction.

SOLUTION   Check the program.

2496   Unknown command.

DETECTION   During execution.

CAUSE   An unknown command has been programmed in the #GRAPHCMD instruction.

SOLUTION   Check the program.

2497   Only one command is allowed to be programmed.

DETECTION   During execution.

CAUSE   Too many commands are scheduled in the #GRAPHCMD instruction.

SOLUTION   Check the program. The #GRAPHCMD instruction supports only one command.

2498   The programmed gap must be greater than the machine parameter GAPSMINLEVEL.

DETECTION   During execution.

CAUSE   The programmed gap value is less than or equal to the machine parameter GAPSMINLEVEL.

SOLUTION   Check the program.

2499   GAP simulation is not possible with parameter SIMMODE = FALSE.

DETECTION   During execution.

CAUSE   Gap control (#GRAPCTRL) has been programmed on an axis that has the SIMMODE=False parameter.

SOLUTION   The gap can only be activated on axes with SIMMODE=True.

2500   Rotary axis movement not permitted with RTCP STATIC active and radius compensation active (G41/G42).

DETECTION   During execution.

CAUSE   An attempt has been made to move a rotating axis of the kinematics, with RTCP STATIC and active radius compensation.

SOLUTION   Deactivate the radius compensation to move the rotary axis.

2501   It is not possible to program in the same linear and rotary axis block with RTCP STATIC active.

DETECTION   During execution.

CAUSE   An attempt has been made to move a linear axis together with a rotary axis of the kinematics, with RTCP STATIC active.

SOLUTION   Move the rotary axis and the linear axis individually.

2502   Rotational axis movement not permitted, in inspection, with RTCP STATIC active.

DETECTION   During execution.

CAUSE   During tool inspection, an attempt was made to move a rotary axis of the active RTCP STATIC kinematics.

SOLUTION   Disable the RTCP STATIC functionality to be able to move the rotary axis.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

2503   G36/37/38/39: The radius is programmed with I.

DETECTION	During execution.
CAUSE	The rounding radius (G36), chamfer (G39) or tangential input/output (G37/G38) has been incorrectly programmed.
SOLUTION	Check the program. Program the radius or chamfer with the "I" command.

2504   G107: The second point (XYZ) has yet to be programmed.

DETECTION	During execution.
CAUSE	One or more of the XYZ coordinates must be programmed to define the second point of G107.
SOLUTION	Check the program.

2505   G107: The first point (IJK) has yet to be programmed.

DETECTION	During execution.
CAUSE	One or more of the IJK coordinates must be programmed to define the first point of G107.
SOLUTION	Check the program.

2506   G107 incompatible with G106 and #SPLINE ON.

DETECTION	During execution.
CAUSE	It is not possible to program splines with G107 active. It is also not possible to program G107 without first deactivating G106 by means of a G0, G1.
SOLUTION	Check the program.

2507   G107: RQ polar coordinates or Cartesian coordinate/angle (X..ZQ) that is not allowed.

DETECTION	During execution.
CAUSE	It is not possible to program the dimensions of a block in G107 with polar coordinates or by Cartesian coordinates and angle.
SOLUTION	Programming dimensions in Cartesian coordinates.

2508   Write value out of range.

DETECTION	During execution.
CAUSE	It is not possible to write that value to the variable.
SOLUTION	Change the value to be written.

2509   #ORBIT ON[Rvalor, Jvalor ,Kvalor].

DETECTION	During editing or block preparation.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the program.

2510   #ORBIT OFF.

DETECTION	During editing or block preparation.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the program.

2511   Parameter ORBIT out of range.

DETECTION	During editing or block preparation.
CAUSE	One of the parameters of the #ORBIT instruction has an incorrect value.
SOLUTION	The value of R/J must be positive and not zero. The value of R must be greater than the tool diameter. The value of K must be 1 or 2.

2512   #ORBIT: the width of the slot must be greater than twice the radius of the active tool.

DETECTION	During editing or block preparation.
CAUSE	The slot width (R) is less than or equal to the tool diameter.
SOLUTION	Program a width greater than the tool radius.

2513   #ORBIT ON now active.

DETECTION	During editing or block preparation.
CAUSE	An attempt was made to activate #ORBIT ON while it was already active.
SOLUTION	To modify any parameter of the #ORBIT instruction, it is necessary to disable it first.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·136·

2514   #ORBIT ON: movement in G0 not allowed.

DETECTION   During editing or block preparation.

CAUSE   A movement has been programmed in G0 with #ORBIT active.

SOLUTION   Remove the block in G0 or disable #ORBIT.

2515   #ORBIT ON: movement of the axis perpendicular to the plane not allowed.

DETECTION   During editing or block preparation.

CAUSE   A movement in the axis perpendicular to the working plane has been programmed with #ORBIT active.

SOLUTION   Remove movement in the perpendicular axis or disable #ORBIT.

3000-3999

3000   'An arc cannot be programmed with a slave axis'

DETECTION	During execution.
CAUSE	The CNC tries to individually move the slave axis of a coupling or of a gantry 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	The CNC tries to move an axis that is in DRO mode.
SOLUTION	The CNC cannot move the axes that are active in DRO mode. To move the axis, cancel the DRO mode [PLC mark DRO(axis)].

3002   'Positive software travel limit overrun'

DETECTION	While repositioning an axis.
CAUSE	During the repositioning movement, the axis has reached the software travel limits. If the repositioning point is beyond the software travel limits, the axis cannot reach that point.
SOLUTION	The axes must always be within the software limits. Check that the software limits are correct, that they have not been modified via part-program or by PLC.

3003   'Negative software travel limit overrun'

DETECTION	While repositioning an axis.
CAUSE	During the repositioning movement, the axis has reached the software travel limits. If the repositioning point is beyond the software travel limits, the axis cannot reach that point.
SOLUTION	The axes must always be within the software limits. Check that the software limits are correct, that they have not been modified via part-program or by PLC.

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. An axis has been programmed that is not valid or is 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.
SOLUTION	Check that the programmed axes are valid and available.

3009   The probe signal has not been received.

DETECTION	During execution.
CAUSE	No signal from the sensor after a palpation movement.
SOLUTION	Check the parameterisation of the probe and physical signals.

3010   'The axis has not been defined as probe (PROBEAXIS)'

DETECTION	During execution.
CAUSE	A probing movement has been programmed on an axis that is not defined as an axis involved in the probing movement.
SOLUTION	Parameter PROBEAXIS determines whether the axis can participate in probing movements or not.

3011 **'Too many axis programmed as probe'**

DETECTION	During execution.
CAUSE	The number of axes programmed in the probing movement exceeds the maximum number of axes of the channel.
SOLUTION	Correct the programming of the probing block.

3013 **'Braking distance greater than parameter PROBERANGE'**

DETECTION	During execution.
CAUSE	The distance required to brake with the active feedrate is greater than the maximum distance admitted (parameter PROBERANGE), maybe due to the following error.
SOLUTION	Reduce the probing feedrate or reduce the amount of following error.

3015 **'During block search, the home search is left out'**

DETECTION	During execution.
CAUSE	During a block search, the CNC has found a home search and it has ignored it.
SOLUTION	Search home outside the program.

3016 **'During the block search, the jog mode is left out'**

DETECTION	During execution.
CAUSE	The user has tried to activate the jog mode during a block search and the CNC has ignored it.
SOLUTION	The CNC does not allow activating the jog mode during a block search.

3017 **'Block search already activated'**

DETECTION	During block search.
CAUSE	The CNC has tried to activate a block search while another one was active.
SOLUTION	Contact your supplier.

3018 **'The constant surface speed has not been reached yet'**

DETECTION	During execution.
CAUSE	The spindle has not had the time to reach the programmed speed.
SOLUTION	Reduce the feedrate, decrease the spindle speed or program the speed before to give the spindle enough time reach its speed.

3019 **'The feedrate programmed in G95 is too low'**

DETECTION	During execution.
CAUSE	The resulting feedrate is very small for the programmed speed.
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 Fagor.

3023 **'Axes missing in the new coordinate system'**

DETECTION	During execution.
CAUSE	One of the first three axes of the new coordinate system is missing.
SOLUTION	Define the first three axes of the channel using the instruction #SET AX.

3025 **'Positive software travel limit overrun'**

DETECTION	During execution.
CAUSE	The CNC has tried to move an axis to a point located beyond the software limits.
SOLUTION	The axes must always be within the software limits. Check that the software limits are correct, that they have not been modified via part-program or by PLC.

3026   'Negative software travel limit overrun'

DETECTION	During execution.
CAUSE	The CNC has tried to move an axis to a point located beyond the software limits.
SOLUTION	The axes must always be within the software limits. Check that the software limits are correct, that they have not been modified via part-program or by PLC.

3027   'Error when calculating the reverse RTCP transformation'

DETECTION	During execution.
CAUSE	The CNC cannot execute the transformation from part-coordinates to machine coordinates.
SOLUTION	Contact Fagor.

3029   'Error when calculating the direct RTCP transformation'

DETECTION	During execution.
CAUSE	The CNC cannot execute the transformation from machine-coordinates to part coordinates.
SOLUTION	Contact Fagor.

3030   'Simulated axes cannot be mixed with non-simulated axes'

DETECTION	During execution.
CAUSE	There are simulated axes and non-simulated ones in the process of measuring with a probe
SOLUTION	All the axes involved in a process of measuring with a probe must be of the same type.

3031'(RTCP/TLC) compensation not allowed in the current status.

DETECTION	During execution.
CAUSE	The CNC has tried to activate one of the RTCP or TLC compensations while the other one was active.
SOLUTION	Both compensations cannot be active at the same time; cancel one of them (#RTCP OFF / TLC OFF) before activating the other one.

3032   'Home search not possible in DRO mode'

DETECTION	During execution.
CAUSE	The CNC has tried to do a home search on a DRO axis.
SOLUTION	The CNC does not allow doing a home search on 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	The CNC tries to activate the jog 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	The CNC tries to access a parameter set that does not exist.
SOLUTION	The set number must be between 1 and parameter NPARSET.

3035   'The difference between the following errors of slaved (coupled) axes is too large'

DETECTION	During execution.
CAUSE	In a gantry axis, the difference between the following error of the master axis and that of the slave axis is greater than the value defined in parameter MAXCOUPE; in an axis coupling, the difference is greater than the defined value.
SOLUTION	Adjust the dynamic behavior of both axes in a similar way or increase the amount of error allowed.

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 up'

DETECTION	During execution.
CAUSE	The CNC tries to move an axis, but the PLC has set the signal SERVO(axis)ON.
SOLUTION	Check the treatment of the SERVO(axis)ON signal in the PLC program or increase the value of the DWELL parameter of the axis. The CNC can issue this error when it works on block blending (G50, G5 or HSC) and it is necessary to force the axis stop (with an M or G7) before moving the "dead" axis.

3038 'Too many parameters pending to be reported'

DETECTION	During execution.
CAUSE	The system is overloaded.
SOLUTION	Close the applications that are not related to the CNC. If the error persists, contact Fagor.

3039 'No stop block found in block search'

DETECTION	While executing in dry run.
CAUSE	The block search 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/G34/G95'

DETECTION	During execution.
CAUSE	Functions G33, C34 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, the channel will use the latter. The CNC will show an error if none of the spindles has been defined.
SOLUTION	Define a master spindle in the channel or assign a spindle to PLC register SYNC.

3042 'The spindle indicated in SYNC register does not exist'

DETECTION	During execution.
CAUSE	A G33, G34 or G95 function is active and the value of PLC register SYNC is not valid.
SOLUTION	The value of PLC register SYNC must be a value between 1 and the number of spindles of the system. It can also have a zero value.

3043 'Spindle in M5 when activating electronic threading'

DETECTION	During execution.
CAUSE	The spindle used in an electronic threading (G33/G34) is stopped or an M5 has been programmed in the same block as the G33/G34.
SOLUTION	The spindle used for electronic threading must be running. Set the spindle running in a previous block or in the same block as G33/G34.

3044 'An axis active as an independent axis cannot be homed (referenced)'

DETECTION	During execution.
CAUSE	The CNC tries to do a home search on an independent axis.
SOLUTION	If the axis has an independent movement left to make, wait until it completes it or abort the movement from the PLC (mark IABORT) or with a reset in the channel. If the axis is synchronized, cancel the synchronization or do a reset in the channel.

3045 'M6 not admitted'

DETECTION	During execution.
CAUSE	After interrupting a program and changing to MHFS mode, the user has edited an M6 that is not allowed.
SOLUTION	The tool change does not permit resuming the execution of the interrupted program. Press reset and execute M6 from the MDI mode.

3046 'The associated spindle does not belong to the channel'

DETECTION	During execution.
CAUSE	After interrupting a program and changing to MHFS mode, the user has edited an M or an S referred to a spindle that does not exist in the channel or in the system.
SOLUTION	Associate the function to the correct spindle. Access the spindle from the channel where it is.

3047 **'M3/M4/M5/M19/M41-M44 not allowed while G63 is active'**

DETECTION	During execution.
CAUSE	After interrupting a program having G63 active and changing to MHFS mode, the user has edited an M3, M4, M5, M19 or M41 through M44 that is not allowed for G63.
SOLUTION	Cancel function G63 or wait until this function is completed before interrupting the program.

3048 **'The programmed gear does not exist'**

DETECTION	During execution.
CAUSE	After interrupting a program and changing to MHFS mode, the user has edited an M function associated with a gear that does not exist in the channel.
SOLUTION	Program the right gear.

3050 **'The S cannot be changed if the spindle works as C axis'**

DETECTION	During execution.
CAUSE	The user has tried to change the speed of a spindle that is active as C axis.
SOLUTION	Program a feedrate F for the C axis or deactivate the C axis (#CAX) to program a speed.

3051 **'Incompatible M functions for the same spindle'**

DETECTION	During execution.
CAUSE	After interrupting a program and changing to MHFS mode, the user has edited some functions M3, M4, M5, M19 or M41 through M44 that are incompatible for the same spindle.
SOLUTION	Decide which M function to execute or execute one after the other, but not at the same time.

3052 **'The spindle status must be the same as the one at the time of the interruption'**

DETECTION	During execution.
CAUSE	After interrupting a program and changing to MHFS mode, the user tries to exit this mode while the spindle is in an M3, M4, M5 or M19 state different from the initial one.
SOLUTION	Program a new spindle state change so it gets into the same state as before the interruption.

3056 **'Error when calculating the reverse transformation of the incline axis #ANGAX'**

DETECTION	During execution.
CAUSE	In an angular transformation, the incline axis and the orthogonal axis form a 90º angle.
SOLUTION	If the axes form an angle of 90º, an angular transformation is not necessary. Cancel the angular transformation (#ANGAX OFF) or define an angle that does not measure 90º.

3057 **'Error when calculating the direct transformation of the incline axis #ANGAX'**

DETECTION	During execution.
CAUSE	In an angular transformation, the incline axis and the orthogonal axis form a 90º angle.
SOLUTION	If the axes form an angle of 90º, an angular transformation is not necessary. Cancel the angular transformation (#ANGAX OFF) or define an angle that does not measure 90º.

3058 **'Spindle functions M3/M4/M5/M19 and M41-M44 must be sent out to the PLC'**

DETECTION	During execution.
CAUSE	After a block search and before resuming the execution of the program, the CNC has detected that some function of the spindle has not been sent to the PLC.
SOLUTION	The program being interrupted, switch to MHFS mode and send the necessary functions to the PLC.

3059 **'M3/M4/M5/M19/M41-M44 not allowed while M19 or G74 is active'**

DETECTION	During execution.
CAUSE	The user has interrupted the program during the execution of M19 or G74 at a spindle. The CNC cannot send functions M3/M4/M5/M19 and M41-M44 out to the spindle.
SOLUTION	Go on with the execution until function M19 or G74 is completed. Then, interrupt the program and send the desired functions to the spindle.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·142·

3060   'The S value cannot exceed the programmed gear'

DETECTION	During execution.
CAUSE	After interrupting a program and changing to MHFS mode, the user has programmed a speed and a new gear, and the speed is higher than the maximum for that gear.
SOLUTION	Change the programmed speed or gear or program only one of them so they don't contradict each other.

3061   A master spindle that has not been homed cannot be synchronized in closed loop.

DETECTION	During execution.
CAUSE	The CNC cannot synchronize a master spindle in closed loop if it has not been previously homed.
SOLUTION	Home the master spindle before synchronizing the spindle.

3062   'The master spindle of the synchronization goes into open loop'

DETECTION	During execution.
CAUSE	The user has programmed an open-loop synchronization (OLOOP) for the master spindle, but it was working in closed loop.
SOLUTION	The CNC informs of the change; no action is required.

3063   'The master spindle of the synchronization goes into closed loop'

DETECTION	During execution.
CAUSE	The user has programmed an closed-loop synchronization (CLOOP) for the master spindle, but it was working in open loop.
SOLUTION	The CNC informs of the change; no action is required.

3064   'The spindle for threading with G33/G34 must be homed'

DETECTION	During execution.
CAUSE	The spindle used in G33/G34 threading has not been homed.
SOLUTION	If the CNC uses the master spindle of the channel, it always carries out a home search before the G33/G34. If the PLC selects another spindle (SYNC mark), perform a spindle home search.

3065   'The spindle turning speed programmed for G33 exceeds the encoder's threshold'

DETECTION	During execution.
CAUSE	The speed programmed for the G33/G34 function exceeded the threshold allowed by the encoder installed on the spindle. Exceeding the speed threshold of the encoder results in erroneous reading (counting) of the spindle feedback pulses.
SOLUTION	The CNC does not permit executing the G33/G34 with the programmed speed because it cannot ensure that the validity of the thread. If the spindle has exceeded the threshold speed, the CNC needs to home the spindle in order to recover the position value.

3066   'Error in the way the tangential axis follows the path'

DETECTION	During execution.
CAUSE	The tangential axis is not faithfully following the path in the work plane.
SOLUTION	Contact FAGOR.

3067   'G33/G34: 'Programming not allowed without a spindle being controlled in position'

DETECTION	During execution.
CAUSE	The programmed function needs a spindle controlled in position. The CNC cannot control the spindle in position because the spindle does not have an encoder.
SOLUTION	The spindle needs an encoder. Contact your supplier.

3068   'RETRACE completed, deactivate mark to continue'

DETECTION	During execution.
CAUSE	After ending the execution in retrace mode, the RETRACE mark of the PLC stays active.
SOLUTION	The PLC must cancel the RETRACE mark in order to resume the program.

3069   Wrong gear change, the drive does not respond.

DETECTION	During execution.
CAUSE	The CNC has not completed the gear change of an axis or spindle because the drive does not respond to the request for a gear change.
SOLUTION	Check the drive status.

3070   **'Zero spindle speed associated with G33/G34/G95'**

DETECTION	During execution.
CAUSE	The spindle associated with function G33, G34 or G95 is stopped.
SOLUTION	Check the speed of the spindle associated with these functions, that will be the master spindle of the channel or the spindle indicated in register SYNC, associated with the channel.

3071   **'S, M or H functions while a gear change process is active are not admitted'**

DETECTION	During execution.
CAUSE	The user has interrupted the program during the gear change and has tried to change the functions S, M or H in MHFS mode.
SOLUTION	Go on with the execution until the gear change is completed. Then, interrupt the program and send the desired functions to the spindle.

3072   **'The programmed pitch and spindle values for G33/G34 exceed the maximum feedrate allowed'**

DETECTION	During execution.
CAUSE	When starting a G33/G34 threading, the CNC has detected that the programmed pitch and speed result in a feedrate that exceeds the maximum permitted. The maximum feedrate may be limited by machine parameters or by PLC.
SOLUTION	Check how function G33/G34 has been programmed. To keep the programmed pitch, modify the programmed speed or, if possible, increase, via PLC, the maximum feedrate of the channel. Otherwise, check the machine parameters related to the maximum feedrate of the axes involved in the thread.

3073   **'Corner speed limit exceeded'**

DETECTION	During execution.
CAUSE	The CNC has exceeded the dynamics of some axis during the transition between two blocks.
SOLUTION	Contact Fagor.

3074   **'Function incompatible when Motion Control is active'**

DETECTION	During execution.
CAUSE	The CNC has tried changing the coordinate system or the coordinate transformation while a control function is active and parameter IMOVEMACH disabled.
SOLUTION	Disable the motion control functions before changing the coordinate system or the coordinate transformation.

3075   **'Multiple axis deactivate'**

DETECTION	During execution.
CAUSE	The axis cannot be moved because it belongs to a multi-axis group and it is deactivated.
SOLUTION	Activate the axis to be able to move it (PLC mark SWITCH).

3076   **'It is not possible get out of MHFS while Retrace is active. Cancel'**

DETECTION	During execution.
CAUSE	The user has tried to get out of MHFS mode while the PLC mark RETRACE was active.
SOLUTION	Cancel the RETRACE mark of the PLC before getting out of the MHFS mode.

3077   **'Single-block mode not allowed in this RETRACE process, cancel Single-block mode'**

DETECTION	During execution.
CAUSE	An attempt was made to activate the block-by-block mode during the retrace with a program with many blocks with very short movements.
SOLUTION	To avoid saturating system resources, the CNC does not allow the block-by-block function for these types of programs.

3078   **'The master spindle of the synchronization does not go into open loop'**

DETECTION	During execution.
CAUSE	The user has programmed an open-loop synchronization (#SYNC) for the master spindle, but it was working in closed loop for another previous synchronization and will stay in closed loop.
SOLUTION	The spindle does not change the loop. Leave the spindle in closed loop or cancel the previous synchronization so the spindle can switch to open loop.

3079 **'The master spindle of the synchronization does not go into closed loop'**

DETECTION During execution.

CAUSE The user has programmed a closed-loop synchronization (#SYNC) (command OLOOP) for the master spindle, but it was working in open loop for another previous synchronization and will stay in open loop.

SOLUTION The spindle does not change the loop. Leave the spindle in open loop or cancel the previous synchronization so the spindle can switch to closed loop.

3080 **'The offset of the not-repositioned axis cannot be assumed'**

DETECTION During execution.

CAUSE The CNC has tried to move an axis that has not been repositioned in a circular path.

SOLUTION It is not possible to move an axis that has not been repositioned in a circular path.

3081 **'Speed too high'**

DETECTION During laser cutting (synchronized switching).

CAUSE The cutting speed is too high for the size of the grid.

SOLUTION Lower the speed or shorten the cycle time.

3082 **'Too much anticipation to turn the digital output on'**

DETECTION During laser cutting (synchronized switching).

CAUSE The delay calculated for the laser is negative; i.e. there is no enough margin of cycles for the anticipation.

SOLUTION Decrease the cycle time or lower the value of the anticipation parameters.

3083 **'Too much delay to turn the digital output on'**

DETECTION During laser cutting (synchronized switching).

CAUSE Too much delay programmed for the laser.

SOLUTION Check the value of the anticipation parameters TON / TOF.

3084 **'Driver error when turning laser ON/OFF'**

DETECTION During laser cutting (synchronized switching).

CAUSE The necessary hardware and parameter setting conditions to use the laser cutting feature are not met.

SOLUTION Check the following conditions:
- There must be a Sercos type expansion board, with software ID equal to or higher than 1.2.
- The value of parameter NLOCOUT must be 8.
- The value of parameter PWMOUTPUT must not be higher than 2 (0, 1, 2).

3086 **The axes of the active plane cannot be changed in Simulated Execution mode: Main plane.**

DETECTION During the simulated execution.

CAUSE A change of plane has been made during the simulated execution.

SOLUTION Check the program.

3087 **'PWM activated in another channel'**

DETECTION During laser cutting (PWM).

CAUSE The PWM feature is already activated in a channel other than than desired one.

SOLUTION Check how the arc has been programmed.

3088 **'Error when programming the laser mask'**

DETECTION During laser cutting (synchronized switching).

CAUSE The necessary hardware and parameter setting conditions to use the laser cutting feature are not met.

SOLUTION Check the following conditions:
- There must be a Sercos type expansion board, with software ID equal to or higher than 1.2.
- The value of parameter NLOCOUT must be 8.
- The value of parameter PWMOUTPUT must not be higher than 2 (0, 1, 2).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF. 2209

3089 **'Error when calculating the reverse transformation of the virtual axis of the tool, #VIRTAX'**

DETECTION   During execution.

CAUSE   The transformation from part coordinates to machine coordinates cannot be executed.

SOLUTION   Contact Fagor.

3090 **'Error when calculating the direct transformation of the virtual axis of the tool, #VIRTAX'**

DETECTION   During execution.

CAUSE   The transformation from machine coordinates to part coordinates cannot be executed.

SOLUTION   Contact Fagor.

3092 **'Work zone limit exceeded'**

DETECTION   During execution.

CAUSE   The CNC has tried to move an axis to a position located outside the no-exit work zone. The zone may be defined either by program or by PLC (using the work-zone related variables).

SOLUTION   Keep the axes within the limits of the no-exit work zone. If an axis is outside the limits, jog it in the proper direction using the jog keys.

3093 **'Point inside the forbidden zone'**

DETECTION   During execution.

CAUSE   The CNC has tried to move an axis to a position located inside the no-entry work zone. The zone may be defined either by program or by PLC (using the work-zone related variables).

SOLUTION   Keep the axes out of the limits of the no-entry work zone. If the tool is inside the forbidden zone, deactivate that zone temporarily, jog the axis in the proper direction until it is outside the forbidden zone and activate the no-entry zone again.

3094 **The sensor had detected a negative distance.**

DETECTION   During execution.

CAUSE   With gap control active, the sensor shows negative distance.

SOLUTION   Check sensor calibration.

3095 **End point incorrectly programmed in #GAPCTRL or #LEAP.**

DETECTION   During execution.

CAUSE   During the #GAPCTRL or #LEAP instruction, the CNC cannot correct the position after receiving a sensor signal.

SOLUTION   Check that the end point indicated in the #GAPCTRL or #LEAP instruction is correct.

3096 **PWRCTRL DUTY activated in another channel.**

DETECTION   During execution.

CAUSE   An attempt has been made to activate #PWRCTRL DUTY on a channel while it active on another channel.

SOLUTION   The #PWRCTRL DUTY can only be active on one channel.

3097 **PWRCTRL OUT activated in another channel.**

DETECTION   During execution.

CAUSE   An attempt has been made to activate #PWRCTRL OUT on a channel while it active on another channel.

SOLUTION   The #PWRCTRL OUT can only be active on one channel.

3098 **PWRCTRL DUTY requires active PWM in the same channel.**

DETECTION   During execution.

CAUSE   An attempt has been made to activate #PWRCTRL DUTY on a channel without it first being activated for the same channel.

SOLUTION   Activate the PWM before activating #PWRCTRL DUTY.

3099 **LEAP not activated due to jump height below a minimum.**

DETECTION   During execution.

CAUSE   The maximum height of the leap is very close to the starting position.

SOLUTION   Check the program.

3100   'Excessive power on the spindle detected during DMC.'

DETECTION	During execution.
CAUSE	DMC has detected excessive power consumption (drive variable TV51) during a period of time. This may indicate that the tool has worn down, been damaged or it has collided with the part.
SOLUTION	Check the tool and/or the programming of the deepening pass sequence and the radial pass sequence.

3101   'Excessive power on the spindle detected during DMC. Press START to continue'

DETECTION	During execution.
CAUSE	DMC has detected that the instantaneous power (drive variable TV51) has exceeded the target power in a preset range. This can indicate that the tool has collided with the part, and so the CNC stops the axes, keeping the spindle turning.
SOLUTION	Check the status of the tool and the machining. The CNC allows the execution to be resumed by pressing [START], to enter in the tool inspection mode or to finish the execution.

3102   Non-compatible instruction with active independent movement.

DETECTION	During execution.
CAUSE	An attempt was made to execute the #HSC or #SWT instruction on an axis with independent motion.
SOLUTION	You can not program these instructions on an axis with an independent movement.

3103   Target power insufficient for the proper operation using DMC.

DETECTION	During execution.
CAUSE	The values of the no-load power and the target power during machining are similar. It is possible that the DMC cannot accurately distinguish the part inputs and outputs.
SOLUTION	The target power (programmed or obtained by learning) must be at least 20% of the no-load power. In this case, it is recommended to deactivate the DMC function for this machining process. Programming a higher target power, which is not actually going to be achieved, can mean that the DMC never detects part inputs and performs all the machining as if the tool were working in a vacuum.

3104   A collision has been detected.

DETECTION	During the execution or simulation of a part-program. While executing in MDI mode
CAUSE	FCAS (Fagor Collision Avoidance System) has detected a collision in one of the movements to be executed or simulated.
SOLUTION	Check the program and modify the part-program.

3105   Tapping or interrupted rigid tapping. Withdrawal allowed in manual mode. Press ENTER + RESET to confirm.

DETECTION	During CNC start-up or after interrupting tapping or rigid tapping.
CAUSE	A shutdown or a serious error has occurred during tapping or rigid tapping.
SOLUTION	Enter manual mode and move the tapping spindle until the tool is removed.

3400   'The independent axis does not exist or is not available'

DETECTION	During execution.
CAUSE	The user has programmed an operation of independent axis for an axis that is not assigned to any channel or is changing channels.
SOLUTION	Assign the independent axis to any channel or wait for the CNC to finish swapping the axis.

3401   'The cam does not exist or is not available'

DETECTION	During execution.
CAUSE	The cam number programmed in the #CAM instruction does not exist.
SOLUTION	Program the right cam or define the cam in the machine parameters.

3402   No more operations are allowed with independent axes. Table full.

DETECTION	During execution.
CAUSE	The CNC tries to execute an operation of independent axis, but there are two operations left to carry out.
SOLUTION	From the PLC, abort the pending instructions (ABORT mark) or wait until there is enough room to store it (FREE mark). Check the BUSY mark to know whether the independent interpolator has finished all the operations or not.

3403   'No more operations are allowed with independent axes. In synchronization'

DETECTION	During execution.
CAUSE	The CNC tries to execute an operation of independent axis, but there is an active synchronization.
SOLUTION	Neither program operations of independent axis with active cams nor program two synchronizations in a row. From the PLC, abort the pending instructions (ABORT mark) or wait until the synchronization is finished.

3404   '#FOLLOW OFF not admitted with another operation pending'

DETECTION	During execution.
CAUSE	The CNC tries to execute a #FOLLOW OFF instruction, but there is an operation pending.
SOLUTION	From the PLC, abort the pending instructions (ABORT mark) or wait until there is enough room to store it (FREE mark).

3405   '#MOVE INF not admitted for an axis with travel limits'

DETECTION	During execution.
CAUSE	Travel limits have been set for the axis programmed in the #MOVE INF instruction.
SOLUTION	Eliminate the software travel limits or interrupt the movement before the axis reaches the limit.

3406   'The speed needed to synchronize exceeds the maximum'

DETECTION	During execution.
CAUSE	The programmed synchronization means exceeding the maximum feedrate of the axis.
SOLUTION	Program the synchronization with the master axis moving at a lower feedrate or program a smaller synchronism ratio.

3407   'The coordinate programmed for the independent axis exceeds the positive travel limit'

DETECTION	During execution.
CAUSE	The coordinate programmed in the #MOVE instruction is beyond the software travel limits.
SOLUTION	The axes must always be within the software limits. Check that the software limits are correct, that they have not been modified via part-program or by PLC.

3408   'The coordinate programmed for the independent axis is smaller than the negative travel limit'

DETECTION	During execution.
CAUSE	The coordinate programmed in the #MOVE instruction is beyond the software travel limits.
SOLUTION	The axes must always be within the software limits. Check that the software limits are correct, that they have not been modified via part-program or by PLC.

3409   'The axis is not valid as independent axis'

DETECTION	During execution.
CAUSE	The CNC tries to execute an operation of independent axis involving an axis that depends on others or on which others depend (e.g. gantry axes, coupled axes or axes involved in a transformation) or because it is a DRO axis or it is a module and the lower limit of the module is negative.
SOLUTION	Cancel the existing coupling (slaving) or the transformations.

3410   'The maximum amount of time allowed for synchronizing has been exceeded'

DETECTION	During execution.
CAUSE	The synchronization does not reach the desired position in the established amount of time.
SOLUTION	Increase the synchronization time, start the synchronization at more similar speeds or increase the acceleration of the slave axis.

3411   'An attempt has been made to exceed to the maximum feedrate MAXFEED'

DETECTION	During execution.
CAUSE	An independent axis movement has been programmed with an active synchronization and the sum of both feedrate components exceeds the maximum permitted for that axis.
SOLUTION	If the feedrate of the master axis is high, the additional movement must be programmed with a lower feedrate.

3412 **'A "0" denominator cannot be programmed in synchronization'**

DETECTION During execution.

CAUSE In the synchronization, the denominator of the gear ratio has a zero value.

SOLUTION The denominator of the ratio cannot be zero.

3413 **'It is not possible to synchronize in position with module and ratio other than 1'**

DETECTION During execution.

CAUSE In position synchronization, the master axis has module and the gear ratio is other than 1.

SOLUTION In position synchronization where the master axis has module, the gear ratio must be 1. Program a gear ratio of 1 or a speed synchronization.

3414 **'A cam cannot be programmed for axes whose module value is different'**

DETECTION During execution.

CAUSE When synchronizing the cam, the master and slave axes have different modules.

SOLUTION Both axes must have the same module (the same parameters MODUPLIM and MODLOWLIM).

3415 **'Independent home search not allowed in execution'**

DETECTION During execution.

CAUSE The PLC tries to do a home search while the CNC is executing another one.

SOLUTION The PLC must wait until the CNC finishes its home search.

3417 **LEAPFROG not possible with given coordinates.**

DETECTION During execution.

CAUSE El CNC cannot execute the leap (#LEAP/#LEAPBEGIN) with the programmed data.

SOLUTION Check the program.

3500 **'Linear acceleration of section 1 smaller than or equal to zero'**

DETECTION On startup or during execution.

CAUSE The programmed acceleration is 0.

SOLUTION Program a positive value for the acceleration.

3501 **'Linear acceleration of section 1 greater than the maximum'**

DETECTION During startup or execution.

CAUSE The programmed acceleration is greater than the maximum.

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 The programmed acceleration is 0.

SOLUTION Program a positive value for the acceleration.

3503 **'Linear acceleration of section 2 greater than the maximum'**

DETECTION During startup or execution.

CAUSE The programmed acceleration is greater than the maximum.

SOLUTION Program a smaller value than the maximum for the acceleration.

3504 **'Acceleration changing speed higher than the maximum'**

DETECTION During startup or execution.

CAUSE The speed programmed for the acceleration change speed is higher than the maximum.

SOLUTION Program a lower speed than the maximum.

3505 **'Jerk limit overshoot'**

DETECTION During execution.

CAUSE Jerk overshoot on this path.

SOLUTION Contact Fagor.

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 Fagor.

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.

3509 **Error limit exceeded in HSC mode.**

DETECTION	During execution.
CAUSE	The theoretical trajectory exceeds the maximum programmed tolerance.
SOLUTION	The accuracy of the machined part may be below the programmed tolerance. Changing the machining conditions.

3510 **The programmed movement will collide with the probe.**

DETECTION	During execution.
CAUSE	The movement of the axis will exceed the limits set by the previous palpation movement. The CNC stops the movement before there is a collision.
SOLUTION	Check post-palpation movement.

3511 **Canceled M3 or M4 from PLC on G96 with no S programmed.**

DETECTION	During execution.
CAUSE	M3/M4 spindle rotation has been requested on G96 without programmed S. The CNC ignores the M3/M4 functions.
SOLUTION	Check programming of G96 and S.

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	The speed programmed for the spindle exceeds the maximum speed defined in the machine parameters for the active gear.
SOLUTION	Check spindle parameter G00FEED.

3602 **'The spindle cannot be moved in DRO mode'**

DETECTION	During execution.
CAUSE	The CNC attempts to control a spindle that is in DRO mode.
SOLUTION	The CNC cannot control the spindles that are active in DRO mode. To move the axis, cancel the DRO mode [PLC mark DRO(axis)] and check the status of the SERVOON mark of the spindle.

3603 **'The commanded spindle position exceeds the module range'**

DETECTION	During execution.
CAUSE	The CNC is trying to position the spindle in a position that is out of the permitted range (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	The programmed positioning movement requires a spindle movement in the opposite direction to what has been set by machine parameter UNIDIR.
SOLUTION	Check the program and spindle machine parameter UNIDIR.

3605 **'Spindle turning direction contrary to the setting of the machine parameter'**

DETECTION	During execution.
CAUSE	The programmed rotation requires a spindle movement in the opposite direction to what has been set by machine parameter UNIDIR.
SOLUTION	Check the program and spindle machine parameter UNIDIR.

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.

3607   'The spindle cannot be synchronized while PLCCNTL is active

DETECTION	During execution.
CAUSE	The CNC has tried to do a synchronization while the PLCCNTL mark was active.
SOLUTION	Deactivate the PLCCNTL mark of the PLC.

3608   'PLCCNTL is not possible if the spindle is synchronized'

DETECTION	During execution.
CAUSE	The PLC has activated the PLCCNTL mark in a spindle that is synchronized, either as master or as slave.
SOLUTION	Deactivate the PLCCNTL mark of the PLC.

3609   '#SERVO ON is not possible if the spindle has not been previously homed'.

DETECTION	During execution.
CAUSE	A #SERVO ON instruction has been programmed for a spindle that has not been homed.
SOLUTION	Home the spindle, with M19 or G74, before using the #SERVO ON instruction.

3610   'The loop cannot be opened because it is in M19 or in synchronization'

DETECTION	During execution.
CAUSE	A #SERVO OFF instruction is programmed but the loop cannot be opened because the spindle was at M19 before #SERVO ON.
SOLUTION	The spindle is kept in a closed loop. The #SERVO OFF instruction cancels the #SERVO ON instruction and, if possible, opens the loop. If the spindle is in M19, the #SERVO OFF instruction cannot open the loop; to open the loop, program M3, M4, M5 or do a reset.

3611   'The master spindle of the synchronization is not in the right gear (SYNCSET or CAXSET)'

DETECTION	During execution.
CAUSE	The master or slave spindle is not in the right gear, parameter SYNCSET or parameter CAXSET if the master spindle is as C axis.
SOLUTION	Change the gear of the master and/or slave spindle in its channel or change the master spindle to the channel of the slave spindle so the gear change is automatic.

3612   SPDLEREV is not possible if the spindle is synchronized or in M19.

DETECTION	During execution.
CAUSE	The PLC has activated the SPDLEREV mark in a spindle that is synchronized, either as master or as slave.
SOLUTION	Deactivate the SPDLEREV mark of the PLC.

3700   'Axis travel limit overrun (PLC)'

DETECTION	During the movement of the axis.
CAUSE	The PLC has tried to overrun the travel limits.
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 software travel limits of the axis.
SOLUTION	Check parameter REFVALUE and the software travel limits.

3702   'Axis following error beyond limits'

DETECTION	During execution.
CAUSE	The axis has exceeded the admitted amount of following error set by machine parameters (MAXFLWE, MINFLWE, FEDYNFAC, ESTDELAY). It may be because the axis has not been adjusted properly, because the axis has not been enabled, because there has been a failure at the motor, at the drive, in the feedback system and/or in the mechanics.
SOLUTION	Check the machine parameters, the axis adjustment, the connections, the status of the drive and motor-feedback system.

3703 **'Positive software travel limit overrun'**

DETECTION   During the movement of the axis.

CAUSE   The axis has exceed the position value set by machine parameter LIMIT+ or by the V.A.RTPOSLIMIT.xn variable.

SOLUTION   Check the program.

3704 **'Negative software travel limit overrun'**

DETECTION   During the movement of the axis.

CAUSE   The axis has exceed the position value set by machine parameter LIMIT- or by the V.A.RTNEGLIMIT.xn variable.

SOLUTION   Check the program.

3705 **'Home search required'**

DETECTION   During execution.

CAUSE   The CNC has lost the spindle reference (home) when switching from open loop to closed loop.

SOLUTION   Contact your supplier.

3706 **'Error when operating with a probe'**

DETECTION   During probe initialization.

CAUSE   The digital input assigned to the probe is not valid.

SOLUTION   Check the probe parameters.

3707 **'Error when searching home'**

DETECTION   During home search.

CAUSE   Error in the home search process. It may be because the axis parameters have not been properly set, because there has been a failure at the counter module, at the Sercos drive, in the feedback system or due to a connection error.

SOLUTION   Check the parameters related with home search, status of the counter module or Sercos drive, feedback system or the connections.

3708 **'Maximum time to get into the in-position zone exceeded'**

DETECTION   During execution.

CAUSE   The time the axis needs to get into the in-position zone so the CNC considers it to be in position, exceeds the value set in parameter INPOMAX.

SOLUTION   Adjust parameter INPOMAX.

3709 **'Error when refreshing analog inputs'**

DETECTION   When reading analog inputs.

CAUSE   Failure in the cyclic reading process for analog intputs. This may 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 at the position command variable.

SOLUTION   Check the settings of the axis parameters.

3711 **'Error when refreshing CAN feedback reading'**

DETECTION   When reading the feedback of analog axes.

CAUSE   If the error shows the name of an axis, the CNC did not complete the reading of that axis feedback. If the error does not show the name of any axis, the CNC did not complete the cyclic refresh of the counters. The error may be due to an error at the CAN bus, to a counting (feedback reading) failure, in the feedback module or due to the saturation of the CAN bus.

SOLUTION   Check the connection and configuration of the CAN bus, the counter module or increase the LOOPTIME if the bus is saturated.

3712 **'Home switch of the master axis is pressed when homing the slave axis'**

DETECTION   During execution.

CAUSE   While homing a gantry axis, the CNC has detected the home switch of the master axis before that of the slave axis.

SOLUTION   Adjust the position of the home switches of the gantry axes; the slave axis must be the first one to press the home switch.

3713   'Tandem control is not active'

DETECTION   During execution.

CAUSE   The CNC has not been able to initialize the additional torque control because the data of some parameter is not available.

SOLUTION   Set and validate the machine parameters of the tandem axis or enable the slave axis and reset the CNC.

3714   Gantry or tandem axes cannot be parked, nor axes enabled with DRENA or SPENA

DETECTION   When parking an axis or spindle from the PLC.

CAUSE   The PLC tries to park an axis of a gantry pair or an axis or spindle of a tandem pair, or an axis enabled (DRENA/SPENA).

SOLUTION   The possible solutions are:
              • The gantry or tandem axes cannot be parked, neither from the PLC nor from the CNC. Eliminate the signal for parking the axis or spindle from the PLC.
              • Disable the axis before parking it (DRENA/SPENA signals).

3715   'Overshooting in module compensation'

DETECTION   During the movement of a rotary axis or spindle.

CAUSE   On the position-Sercos axis, the CNC keeps track of the module compensation in an absolute way; if all the movements of the axis are made in the same direction, the CNC keeps accumulating the compensation.

SOLUTION   Do a home search.

3716   'Maximum velocity command exceeded'

DETECTION   During the movement of an axis or spindle.

CAUSE   On velocity-Sercos axes, the CNC sends the command to the drive in ten-thousandths of an rpm; the command value has exceeded the permitted range. Even if the programmed feedrate does not exceed G00FEED, the command sent to the drive after applying Kv exceeds the maximum permitted.

SOLUTION   Check machine parameters PITCH, INPUTREV, OUTPUTREV and PROGAIN. Limit the maximum feedrate (parameter G00FEED) to the value indicated by the warning in order to avoid overshooting.

3717   Encoder pulse counting threshold exceeded. Search home to restore the coordinate.

DETECTION   During execution.

CAUSE   The encoder of the axis or spindle has exceeded the maximum turning speed permitted. When exceeding this limit, the axis or spindle has lost the reference point (home) and consequently, the positions read are not correct.

SOLUTION   Do a home search to work in closed loop.

3718   'RESET to park axes in motion or of the active transformation/coupling'

DETECTION   When parking an axis from the PLC.

CAUSE   The PLC tries to park a moving axis, that belongs to the active kinematics, to an active angular axis, to the active tangential control or to a coordinate transformation #CS or #ACS.

SOLUTION   The PLC cannot part the axis in this situation. Press reset to stop the movement or cancel the active kinematics, the active angular axis, the active tangential control or the coordinate transformation #CS or #ACS.

3719   'RESET to park spindles in motion or G33/G34/G63/G95/G96/#CAX/#SYNC'

DETECTION   When parking an axis from the PLC.

CAUSE   The PLC tries to part a spindle that is moving, synchronized, activated as C axis or while some of the functions G33, G34, G63, G95 or G96 is active.

SOLUTION   The PLC cannot part the spindle in this situation. Press reset to stop the movement, the synchronization, deactivate the C axis or deactivate functions G33, G34, G63, G95 or G96.

3720   'Probe not available. Busy in another process'

DETECTION   During execution.

CAUSE   The CNC has tried to execute a probing action, but the probe is being used by a previous process (G100 or TOUCHPROBE command).

SOLUTION   Wait until a probing action is completed before executing the next one.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·153·

3721   Error when initializing the probe.

DETECTION	During execution.
CAUSE	Stylus parameters are not consistent.
SOLUTION	Check the palpator parameters (PROBETYPE, PRBDI, PRBPULSE).

3722   Error in the limits of the palpator window from PLC.

DETECTION	During execution.
CAUSE	In PLC palpator, the limits of the palpation window (lower limit > upper limit) are poorly defined.
SOLUTION	Review the programming of the TOUCHPROBE command in the PLC program.

3723   'G174 not possible for an axis in DRO mode'

DETECTION	During execution.
CAUSE	The CNC has detected function G174 for an axis that is in DRO mode (DRO mark).
SOLUTION	Check the program. In order to force the machine coordinate of an axis (G174), the axis must not be in DRO mode, deactivate its DRO(axis) mark and activate its SERVO(axis)ON mark.

3724   'G174 is not possible if the axis is not in position'

DETECTION	During execution.
CAUSE	The CNC has detected function G174 for an axis that is not in position.
SOLUTION	Check the program. In order to force the machine coordinate of an axis (G174), it must be in position; i.e. not moving and the PLC mark INPOS(axis) must be active.

3725   'G174 not possible for an axis that is synchronized'

DETECTION	During execution.
CAUSE	The CNC has detected function G174 for an axis that is synchronized.
SOLUTION	Check the program. To force the coordinate of an axis (G174), the axis must not be synchronized with another one. Desynchronize the axis or check, in the PLC program, the logic of the INSYNC mark of the axis.

3726   'The "Drive Status Ready (DRSTAF/DRSTAS) of the slave axis of the tandem is missing'

DETECTION	When starting the movement of a tandem axis.
CAUSE	The CNC has tried to move the master axis of the tandem without waiting for the slave axis to be enabled.
SOLUTION	In the PLC program, allow movement only if both tandem axes are enabled.

3727   'A multiple axis cannot be deactivated without getting in position'

DETECTION	When deactivating the SWITCH mark of the axis.
CAUSE	The CNC has tried to deactivate an axis of a multiple group before the axis reached position; i.e. while there is a movement pending.
SOLUTION	In the PLC program, condition the deactivation of the axis to being in position; i.e. if it is not moving.

3728   'Multiple axis with a command in execution at the drive , SWITCH not allowed'

DETECTION	When deactivating the SWITCH mark of the axis.
CAUSE	The CNC has tried to deactivate an axis of a multiple group before completing a Sercos command to park, unpark, change gears or change sets.
SOLUTION	In the PLC program, allow deactivating the axis only if the parking, unparking, gear change or set change has finished.

3729   'There cannot be more than one axis of the multiple group active'

DETECTION	When activating the SWITCH mark of the axis.
CAUSE	The CNC has tried to activate an axis of a multiple group, before deactivating the previous axis.
SOLUTION	In the PLC program, first deactivate the last active axis and then activate the new axis.

3730   'TOUCHPROBE: The selected flank will be ignored because it is different from the active one'

DETECTION	When the PLC is executing a TOUCHPROBE command.
CAUSE	The PLC has tried to execute a probing action, but the probe is being used by a previous TOUCHPROBE command or both commands use different flanks.
SOLUTION	In simultaneous probing operations on different axes, either the same probing flank must be used or the probing operations must be carried out with different probes.

3731   'G174: Invalid movement, value too high'

DETECTION   When executing G174

CAUSE   When calculating the movement programmed in units of the drive, the resulting value is out of the range allowed by the drive.

SOLUTION   Change value programmed in G174.

3732   Error in refreshing local counts.

DETECTION   In the reading of local analogue readings from the position loop.

CAUSE   Alarm at one of the local collection inlets.

SOLUTION   Check the connection of local counts and signal levels.

3733   Error when refreshing local feedback reading.

DETECTION   When reading local analogue inputs from the position loop.

CAUSE   Alarm on a local analogue input.

SOLUTION   Check the connection of the inputs and the connected probe.

3734   'PWM software option not allowed'

DETECTION   During CNC startup.

CAUSE   The software option for the PWM feature is not active at the CNC.

SOLUTION   Purchse the corresponding option.

3735   'Error when turning the PWM ON/OFF'

DETECTION   During laser cutting (PWM).

CAUSE   The PWM parameters are set wrong.

SOLUTION   Check the following conditions:
• The value of parameter NLOCOUT must be 8.
• The value of parameter PWMOUTPUT must not be higher than 2 (1, 2).

3736   'Machine parameter PWMOUTPUT = 0'

DETECTION   During laser cutting (PWM).

CAUSE   The value of parameter PWMOUTPUT cannot be 0.

SOLUTION   Set the value of parameter PWMOUTPUT within the range (1-2).

3737   'Expansion board not compatible with PWM'

DETECTION   During laser cutting (PWM).

CAUSE   The hardware is not ready for the PWM feature.

SOLUTION   Verify that there is a Sercos type expansion board with the first 2 local outputs ready for fast switching. The software ID of the FPGA expansion must be equal to or higher than 1.3, and its hardware ID equal to or higher than 1.2.

3738   'The axis has moved while the CNC was off'

DETECTION   During CNC startup.

CAUSE   The absolute coordinate (position value) of the drive does not match the one saved in the last CNC session. The axis has moved more than what was set in parameter MAXDIFREF.

SOLUTION   Verify that the axis position is correct.

3739   Error in the probe signal during the #GAPCTRL process.

DETECTION   During execution.

CAUSE   With the active gap control, the CNC reads a distance greater than the maximum possible (parameter GAPDISTLIMIT). The possible causes are:
• The probe is not working properly.
• The signal received by the CNC is not from the probe (probe connected incorrectly).
• The probe is incorrectly parameterized (parameters GAPDISTLIMIT and GAPVOLTLIMIT).

SOLUTION   Check the connection and parameterization of the probe. If the error persists, contact your supplier.

3740   ENABLEGAP: Cannot activate #GAPCTRL since it is outside the programmed GAP.

DETECTION   During execution.

CAUSE   The gap control is active (#GAPCTRL) but disabled by the PLC (mark ENABLEGAP). The CNC cannot enable the gap control because the probe is outside the gap.

SOLUTION   Bring the probe closer to the programmed gap.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

3741   #GAPCTRL: out of range [GAPMAX - GAPMIN].

DETECTION	During execution.
CAUSE	The gap has exceeded the maximum or minimum allowed values set by GAPMAX or GAPMIN. It is possible that the sensor is not reading correctly.
SOLUTION	Check the program. Change the parameter value of GAPERRORCANCEL so that the CNC does not prompt an error.

3742   #GAPCTRL: outside the GAPTOL range.

DETECTION	During execution.
CAUSE	The difference between the gap signal and the programmed gap is greater than the allowed tolerance (parameter GAPTOL).
SOLUTION	Check the program. Change the parameter value of GAPTOLCANCEL so that the CNC does not prompt an error.

3743   Palpator collision.

DETECTION	During execution.
CAUSE	A signal from the palpation probe arrived when it was not expected.
SOLUTION	Check that there are no obstacles in the path of the axes.

3744   #GAPCTRL: No probe signal.

DETECTION	During execution.
CAUSE	There is no signal from the sensor, which the CNC uses to calculate the distance.
SOLUTION	Check the probe connections and probe related machine parameters (GAPANAINTYPE and GAPAANAINID).

3745   The probe has collided.

DETECTION	During execution.
CAUSE	The PLC has indicated, by using the mark GAPCOLLISION, there has been a collision with the sensor.
SOLUTION	To avoid a collision with the plate, reduce the feedrate. If the error continues, the OEM will have to modify the machine parameter GAPCOLLISIONMODE.

3746   Initial position out of range.

DETECTION	On start-up or recovery of the operational phase of the bus.
CAUSE	The CNC has calculated a position that exceeds the allowed range.
SOLUTION	Review PITCH, NPULSES, INPUTREV, OUTPUTREV and ABSOFF parameters.

3747   Hardware not compatible with SWT.

DETECTION	During the execution of the grid cut.
CAUSE	The hardware is not ready for the "synchronized switching" option.
SOLUTION	Verify that the CNC has 2 local digital outputs prepared for fast switching.

3748   Hardware not compatible with PWM.

DETECTION	During the execution of the PWM.
CAUSE	The hardware is not ready for the PWM feature.
SOLUTION	Verify that the CNC has 2 local digital outputs prepared for fast switching.

3749   Probe signal not detected in #GAPCTRL or #LEAP.

DETECTION	During execution.
CAUSE	During #GAPCTRL or #LEAP, the axis reached the approach point without receiving the probe signal.
SOLUTION	Change the approach position to the workpiece.

3750   The axis reading (count) cannot be inhibited.

DETECTION	When deactivating an axis.
CAUSE	The axis that is deactivated does not have a feedback in the RCS-S module.
SOLUTION	Only one count of the RCS-S module can be inhibited. Check bit 0x80000000 of AXISOPTION of the axis involved in the disabling manoeuvre.

3751   Drive Feedback not valid.

DETECTION	During CNC power-up or after a reset.
CAUSE	The position feedback for the axis could not be calculated correctly. The drive has not been correctly initialised.
SOLUTION	Check the connection and parameterisation of the feedbacks, and restart the system.

3752   Gap out of range for the position limit.

DETECTION   During execution.

CAUSE   During #GAPCTRL or #LEAP, the axis reached the approach point without receiving the probe signal.

SOLUTION   Change the approach position to the workpiece.

3753   Difference of position greater than MAXDIFREF. Press ENTER + RESET to confirm.

DETECTION   During CNC startup.

CAUSE   The absolute coordinate (position value) of the feedback device does not match the one saved during the last CNC session.

SOLUTION   The possible solutions are:
- Verify that the position of the axis indicated in "Current Coordinate" is correct. If so, press [ENTER] and then [RESET] to clear the error and continue. If the axis position is not correct, there may be an error with the feedback device.
- If the position that the axis should have is not "Current Coordinate" but "Expected Coordinate", then clear the error by pressing [ENTER]" and then [RESET]". Then, execute G174 with the position indicated in "Expected Coordinate" and, finally, restart the CNC to recover this position.

3754   Incompatibility with the direction of the sensor and the direction of the gap compensation.

DETECTION   During execution.

CAUSE   With n-axis gap control active (#GAPCTRL), the angle programmed for gap compensation is incompatible with the sensor direction.

SOLUTION   Change the programming angle.

3755   Machine parameter PLOOPRESOL incompatible with drive version.

DETECTION   During CNC start-up or validation of machine parameters.

CAUSE   The drive does not have the right version for the PLOOPRESOL value.

SOLUTION   Check the value of the PLOOPRESOL parameter and the version of the drive.

3756   RPS, SSTIME parameter time reached without having a stable bus voltage.

DETECTION   When starting the RPS manoeuvre with PWSENAS.

CAUSE   Bus voltage is not charged within the estimated time.

SOLUTION   Increase the SSTIME time or check the mains voltage on the equipment performing the DC bus softstart.

3757   RPS, error in the execution of the sense board synchronisation command.

DETECTION   When starting the RPS manoeuvre with PWSENAS.

CAUSE   No voltage is detected on any of the sense plate branches during synchronisation.

SOLUTION   Check connection, sense board, X11 motor power cable. Drive not enabled.

3758   RPS, drive in error or in an incorrect state.

DETECTION   When starting the RPS manoeuvre with PWSENAS.

CAUSE   The drive is in error or in a state not expected by the CNC.

SOLUTION   Check the connection and parameterisation of the drive.

3759   RPS, contactor control fault, overcurrent on output X11 brake-contactor.

DETECTION   When starting the RPS manoeuvre with PWSENAS.

CAUSE   Brake output overcurrent.

SOLUTION   Check connection, missing 24 Vdc power supply or faulty contactor.

3760   RPS, bus voltage below the desired voltage.

DETECTION   When starting the RPS manoeuvre with PWSENAS.

CAUSE   Unexpected drop in the mains line supplying the manoeuvre.

SOLUTION   Check installation and operation.

3761   RPS, mains voltage high, reaches DC bus setpoint.

DETECTION   When starting the RPS manoeuvre with PWSENAS.

CAUSE   Mains voltage rise, which, when rectified, reaches the DC bus setpoint.

SOLUTION   Raise the setpoint voltage of the DC bus.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

3762   Error in the latching command of I0.

DETECTION   During the process of referencing the axis/head.
CAUSE   The I0 latch command has given an error.
SOLUTION   Contact Fagor Automation.

3763   Error in the controller error reset command.

DETECTION   After a CNC reset.
CAUSE   The error reset command of the controller has given an error.
SOLUTION   Contact Fagor Automation.

3764   Error in the command to set new position reference.

DETECTION   During the process of searching for I0 or G174.
CAUSE   The command to set new position reference has failed.
SOLUTION   Contact Fagor Automation.

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 jogging feedrate V.G.FMAN has been programmed and machine parameter JOGFEED is zero.
SOLUTION   Set a new feedrate from JOG mode. Check axis machine parameter JOGFEED.

3801   'Distance or feedrate in incremental jog equal to zero'

DETECTION   During execution.
CAUSE   The moving feedrate or distance in incremental jog is zero. No jogging feedrate has been programmed and machine parameter INCJOGFEED is zero or machine parameter INCJOGDIST is zero.
SOLUTION   Set a new feedrate from JOG mode. Check axis machine parameters INCJOGFEED and INCJOGDIST.

3802   'Incremental jog feedrate too low'

DETECTION   During execution.
CAUSE   The position increment calculated for a PLC cycle is too small.
SOLUTION   Check axis machine parameters INCJOGFEED and LOOPTIME.

3803   'The handwheel resolution cannot be zero'

DETECTION   While changing handwheel resolution from the operator panel.
CAUSE   The handwheel position selected from the switch or PLC has a zero resolution associated with it.
SOLUTION   Check axis machine parameter MPGRESOL.

3804   'Handwheel index out of range (switch positions 1-3)'

DETECTION   While changing handwheel resolution.
CAUSE   The PLC has tried to select a handwheel position that is out of the permitted range.
SOLUTION   Check in the PLC program the writing of the (V.)PLC.MPGIDX variable.

3805   'Zero feedrate or distance in incremental jog'

DETECTION   While changing the moving distance or feedrate in incremental jog.
CAUSE   The incremental jog position selected by the switch or by PLC 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   The PLC has tried to select an incremental jog position that is out of the permitted range.
SOLUTION   Check in the PLC program the writing of the (V.)PLC.INCJOGIDX variable.

3808   **'The axis does not exist or is not available'**

DETECTION   During execution.

CAUSE   The possible causes are:
- The axis was not in jog mode when quitting that mode.
- The CNC has detected function G101 or G102 for the slave axis of a gantry pair.
- Invalid axis name in the #TANGCTRL ON instruction.

SOLUTION   The possible solutions are:
- Press reset to exit the jog mode.
- Functions G101 and G102 must be programmed for the master axis of the gantry axis.
- Check the programming of the #TANGCTRL ON instruction.

3809   **'Zero spindle speed programmed in G95'**

DETECTION   During execution.

CAUSE   The CNC has tried to move an axis in continuous or incremental jog while function G95 was active and zero speed at the spindle used for synchronization.

SOLUTION   Program a speed for the master spindle of the channel or for the spindle used for synchronization (SYNC register).

3810   **'Handwheel mode not allowed with Hirth axis'**

DETECTION   During execution.

CAUSE   The CNC has tried to select a Hirth axis to move it in handwheel mode.

SOLUTION   A Hirth axis cannot be moved in handwheel mode; it may be moved in continuous and incremental jog modes.

3811   **'Handwheel increment limit exceeded'**

DETECTION   In jog mode, while reading (counting) the pulses of a handwheel.

CAUSE   Handwheel pulse reading (counting) increment too large.

SOLUTION   Check the feedback connected to the handwheel input; signal levels, frequency, etc.

3812   **'Handwheel refresh error'**

DETECTION   In jog mode, while reading (counting) the pulses of a handwheel.

CAUSE   Wrong handwheel reading (counting).

SOLUTION   Depending on the type of handwheel (CAN keyboard, CAN counter, local, RCS-S module, etc.), check for coherence and communication level of the corresponding bus as well as the feedback input connection.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·159·

4000-4999

4000(1)   'Error when initializing the Sercos ring'

CLASS   0.

CAUSE   Error when initializing the chip. The CNC does not detect the Sercos board or overflow at the DPRAM of the SERCON due to too many axes and data to be transmitted in the cyclic channel.

SOLUTION   Verify that the board is installed and that the CNC detects it correctly. If the error persists, contact the Service Department.

4000(2)   'Error when initializing the Sercos ring'

CLASS   1.

TRANSITION   0.

CAUSE   Timeout when requesting Sercos ring initialization.

SOLUTION   Contact our technical service department.

4000(3)   'Error when initializing the Sercos ring'

AXIS   Name of the axis issuing the error.

TRANSITION 1   Sercos ID that causes the error. Refer to the drive manual.

TRANSITION 2   Point of the initializing sequence where the error occurs.

Point.	Meaning.
0	Timeout.
1	Error when changing to Phase 0. (Trouble at the optical fiber)
2	Error when changing to Phase 1. (A drive does not respond; hardware failure or node selector set wrong)
3	Error when changing to Phase 2.
4	Error when reading the OEM version.
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.

Point.	Meaning.
24	Error when writing MDT Offset.
25	Error when writing RealTime Control Bit 1.
26	Error when writing RealTime Control Bit 2.
27	Error when writing the RealTime Status Bit1.
28	Error when writing OpMode.
29	Reset command error.
30	Park command error.
31	Phase 3 command error.
32	Error when changing to Phase 3.
33	Phase 4 command error.
34	Error when changing to Phase 4.
35	Error when reading Class Diagnostics 1.
36	Default error.
37	Error when reading Tncyc.
38	Error when reading OpMode.
39	Error when reading AxisType.
40	Error when reading G00Feed.
41	Error when reading Monit Window.
42	Error when reading SP100.
43	Error when reading KV.
44	Error when reading the Checksum.
45	Error when reading DV33.
46	Error when writing DV33.
47	Error when reading attributes of Sercos variables.
48	Error when reconfiguring the MDT and AT.
49	Error when reading the MP2.
50	Error when reading the MP3.
51	Error when writing the matching parameters.
52	Error when reading the PP55.
53	Error when reading the PP115.
54	Error when reading the PP147.
55	Error when reading the Checksum.
56	Error when reading MP44.
57	Error when reading RP77.
58	Error when reading PP76.
59	Error when reading GV9.
60	Error when reading SP10.

VALUE 1   Errors at the Sercos drive.

Error.	Meaning.
0	SERC_NO_ERROR
1	ERROR_PHASE_CHANGE
5	Request of Abort/Suspend/Resume of a command that is not active.
7	Wrong logic axis number.
0x0040	READY_FOR_SCDATA
0x0080	ERROR_DEFAULT
0x0101	NOT_READY

Error.	Meaning.
0x0102	BUSYTIMEOUT
0x0201	ERROR_ATMISS
0x0202	ERROR_NERR
0x0203	ERROR_MSTMISS
0x0204	ERROR_DISTORSION
0x0205	ERROR_FIBRA_ROTA
0x0400	ERROR_SCTRANS
0x0801	ERROR_SCTRANSNODATA
0x0802	ERROR_SCNODATA
0x0803	NOT_READY_FOR_SCDATA
0x1002	ERROR_DPRAMOVERFLOW
0x1004	ERROR_SCNOTINIT
0x1008	ERROR_WRONGCHANNELNUMBER
0x2001	ERROR_WRONGPHASE
0x2002	ERROR_WRONGADDRESS
0x2004	ERROR_WRONGATNUMBER
0x2008	ERROR_SCTRANSNOTREADY
0x4000	ERROR_CALCULATE_T1
0x4001	ERROR_CALCULATE_T2
0x4002	ERROR_CALCULATE_T3
0x4004	ERROR_CALCULATE_T4
0x4008	ERROR_CALCULATE_TEND
0x7002	Error in the service channel: Length of the smaller variable.
0x7003	Error in the service channel: Length of the bigger variable.
0x7004	Error in the service channel: Variable without reading permission.
0x7005	Error in the service channel: Variable without reading permission in the current phase.
0x7006	Error in the service channel: Value of the variable too low.
0x7007	Error in the service channel: Value of the variable too high.
0x7008	Error in the service channel: Invalid variable.
0x7009	Error in the service channel: Access to variable protected with password.
0x700A	Error in the service channel: Variable configured in cyclic channel.
0x8001	ERROR_HSTIMEOUT
0x8002	ERROR_SCHSTIMEOUT
0x8004	SERC_ERROR_TIMEOUT

VALUE 2   Not being used at this time

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.

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. Node selector of the drive (DriveID), parameters LOOPTIME, SERPOWSE, SERBRATE, OPMODE, etc.

4000(4)   'Error when initializing the Sercos ring'

CAUSE   Timeout when resetting errors after initializing the Sercos ring.

SOLUTION   Contact our technical service department.

4000(5)   **'Error when initializing the Sercos ring'**

VALUE   5.
TRANSITION   3.
VALUE   35.
CAUSE   Error when resetting errors after initializing the Sercos ring.
SOLUTION   Contact our technical service department.

4001   **'Parameter LOOPTIME different at the CNC and at the drive'**

VALUE   Parameter value at the CNC.
VALUE   Parameter value at the drive.
AXIS   Logic number of the axis
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'**

VALUE   Parameter value at the CNC.
VALUE   Parameter value at the drive.
AXIS   Logic number of the axis
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'**

VALUE   Parameter value at the CNC.
VALUE   Parameter value at the drive.
AXIS   Logic number of the axis
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.

4004   **'Parameter G00FEED higher at the CNC than at the drive'**

VALUE   Parameter value at the CNC.
VALUE   Parameter value at the drive.
AXIS   Logic number of the axis
CAUSE   Value of the parameter G00FEED 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'**

VALUE   0.
VALUE   PP159 parameter value at the drive.
AXIS   Logic number of the axis
CAUSE   'Following error monitoring not active at the drive'
SOLUTION   Activate parameter PP159 at the drive.

4006   **'Drive parameter SP100 must be 0'**

VALUE   0.
VALUE   SP100 parameter value at the drive.
AXIS   Logic number of the axis
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'**

VALUE   0.
VALUE   KV value at the drive.
AXIS   Logic number of the axis
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'**

AXIS   Logic number of the axis
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. If the error persists, contact the Service Department.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·163·

4009   'Too many variables in a Sercos telegram'

AXIS	Logic number of the axis affected by the error.
CAUSE	The maximum number of Sercos variables to process cyclically in a telegram is limited to 8. More than 6/7 synchronous Read/Write variables were defined in the table of Sercos variables for the same axis.
SOLUTION	Reduce the number of synchronous variables to treat on that axis. Define some of those variables for asynchronous access.

4010   'The parameter cannot be written at the drive'

DETECTION	When writing drive parameters while initializing the Sercos ring; CNC start-up, validation of machine parameters and reset.
CAUSE	Generic failure when writing the parameter.
SOLUTION	Check the communication status with the drive, software version and parameter setting options.

4011   'The parameter cannot be written at the drive: No IDN'

DETECTION	When writing drive parameters while initializing the Sercos ring; CNC start-up, validation of machine parameters and reset.
CAUSE	The parameter does not exist at the drive.
SOLUTION	Update the software version of the drive.

4012   'The parameter cannot be written at the drive: Out of range'

DETECTION	When writing drive parameters while initializing the Sercos ring; CNC start-up, validation of machine parameters and reset.
CAUSE	The value of the machine parameter is out of the permitted range.
SOLUTION	Correct the parameter value at the CNC.

4013   'The parameter cannot be written at the drive: Protected'

DETECTION	When writing drive parameters while initializing the Sercos ring; CNC start-up, validation of machine parameters and reset.
CAUSE	Write-protected parameter.
SOLUTION	Check the permissions and the access level at the drive.

4014   'Error when initializing the SERCON'

DETECTION	When initializing the Sercos chip; CNC start-up, validation of machine parameters and reset.
CAUSE	Wrong number of Sercos axes or Sercos times. Insufficient memory in the SERCON chip for the Sercos configuration that has been set.
SOLUTION	Check Sercos configuration and parameter settings.

4015   'TimeOut when initializing the Sercos ring'

DETECTION	When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.
SOLUTION	Check Sercos configuration and parameter settings.

4016   'The drive is not a Fagor drive. Software option not allowed'

DETECTION	When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.
CAUSE	The system has detected a non-Fagor drive without having the corresponding software option.
SOLUTION	The non-Fagor digital servo option must be activated at the CNC in order to be able to connect third-party Sercos drives.

4017   'It is necessary to recalculate the value of drive parameter PP4(HomingSwitchOffset)'

DETECTION	On CNC power-up.
CAUSE	The calculation of the module has switched from the drive to the CNC which means that the value of PP4 is not valid and it must be recalculated. On CNC power-up, it checks whether there is a non-integer gear ratio on spindles and rotary axes with module and whether parameter PP76(7)=1 or not. In this case, instead of writing a 0 in that bit, the CNC shows a message indicating that the drive parameters have been set wrong and the value of PP4 must be recalculated with PP76(7)=0.
SOLUTION	From the DDSsetup, set parameter PP76(7)=0, validate it and execute the GC6 command so the drive recalculates the new value of PP4. Finally, save the parameters in the flash memory of the drive.

4018   Remote Sercos feedback with incoherent FEEDBACKTYPE in various gears (ranges).

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   If there is any feedback input for the RCS-S module, the feedback (parameter FEEDBACKTYPE) is not the same for all sets.

SOLUTION   Correct the parameterization.

4019   Sercos ring error due to a counter reset.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   A reset has occurred for the RCS-S module, due to either a physical cause (power outage, etc.) or internally because of the node software.

SOLUTION   Remedy the cause of the problem. If the error persists, contact Fagor.

4020   Incoherent parameter settings.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   Incorrect parameterization of the RCS-S module or its resources.

SOLUTION   Correct the parameterization.

4022   Sercos ring error due to a counter reset by Watchdog.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   A watchdog reset has occurred for the RCS-S module, due to an internal cause regarding the node software.

SOLUTION   Contact Fagor.

4023   The trace configured in Sercos cannot be configured.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   Error during the configuration of the AT-MDT telegrams to capture the programmed trace. The programmed trace cannot be realized, either because it is a non-Fagor drive or because there is no space available for it.

SOLUTION   Reduce the number of variables in the telegram. If the bandwidth is limited, raise the SERBRATE frequency or remove drives from the ring.

4024   Invalid drive version for in-position TANDEM axis or for REFMODE CNC.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   The version of a drive does not support tandem torque control or CNC reference mode.

SOLUTION   To install the above features, install a version later than v6.28 or v8.08 on the drives.

4025   A Multi-Axis Box (MAB) has not been identified on the Sercos ring.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   A MAB has not been detected in the system. A MAB is required for the option MABSYSTEM when there is more than one drive on the Sercos ring.

SOLUTION   Delete the MABSYSTEM option or include a MAB in the Sercos configuration.

4026   Inconsistent FeeDat/EnDat encoder data. A new configuration must be validated.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   There is one or more FeeDat/EnDat feedback, where the data being read from the encoder is different from those stored on disk. The encoder of the feedback, therefore, is different from the one that was validated during the system setup.

SOLUTION   The encoder that was validated for each FeeDat/EnDat feedback must be reconnected or the new encoder must be connected.

4027   FeeDat/EnDat feedback with parameters set in the Sercos counter is incompatible with the protocol.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   A FeeDat/EnDat feedback has been parameterized in an RCS-S module, which does not permit it.

SOLUTION   The RCS-S module must be updated or to change it for another node that supports digital FeeDat/EnDat protocols.

4028   DRV variable whose IDN does not exist in the Sercos node.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   Sercos bus initialization has failed because a DRVVAR variable has incorrect IDN.

SOLUTION   Check the DRVVAR table and correct the parameterization. The initialization of the Sercos bus is not completed correctly as long as the parameterization is not consistent.

4029   DRV variable whose attributes cannot be read.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   The initialization of the Sercos bus has failed because the attributes of a DRVVAR variable cannot be read.

SOLUTION   Check the DRVVAR table and correct the parameterization. The initialization of the Sercos bus is not completed correctly as long as the parameterization is not consistent.

4030   Erroneous numbering of Sercos nodes.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   The parameterization of Sercos nodes (controllers and counters) is not consistent.

SOLUTION   Review the parameterization of Sercos axes and counters, and correct the parameterisation. The initialization of the Sercos bus is not completed correctly as long as the parameterization is not consistent.

4031   Incremental FeeDat/EnDat encoders are not supported.

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset.

CAUSE   A FeeDat/EnDat encoder of a Sercos counter has been parameterized as incremental.

SOLUTION   Review the parameterization of the FeeDat/EnDat encoders of the Sercos counter, ensuring that all are absolute. The initialization of the Sercos bus is not completed correctly as long as the parameterization is not consistent.

4032   It is recommended that the Sercos time-scheduling be changed (Diagnosis mode).

DETECTION   When initializing the Sercos bus; CNC start-up, validation of machine parameters and reset. Only in Setup mode.

CAUSE   The Sercos timing diagram is not optimal enough to guarantee the synchronism of the system.

SOLUTION   From the Sercos diagnosis page, carry out the time readjustment and accept the new proposed diagram.

4033   Gap Control setting for drive not supported.

DETECTION   When initialising the Sercos bus.

CAUSE   The GAPANAINID parameter indicates a Fagor drive whose version does not support gap control or the drive is not Fagor.

SOLUTION   Use a Fagor drive with software version v9.03 or higher, or override the drive's internal gap control.

4034   For Gap Control, the drive must be Sercos Position.

DETECTION   When initialising the Sercos bus.

CAUSE   The drive selected for gap control is not Sercos position.

SOLUTION   Correctly parameterise the drive that must perform gap control.

4035   For Gap Control, the drive must have REFMODE = Drive.

DETECTION   When initialising the Sercos bus.

CAUSE   The axis selected for gap control has its REFMODE parameter set to "CNC", not "Drive".

SOLUTION   Define the REFMODE parameter of the axis as "Drive".

4036   For Gap Control, the drive must have its external feedback free.

DETECTION   When initialising the Sercos bus.

CAUSE   The drive selected for gap control has no gap control card installed (connector X3).

SOLUTION   Install a gap control board in the drive (connector X3).

4037   The drive does not support synchronous writing of the DRV FFGAIN.

DETECTION   When initialising the Sercos bus.

CAUSE   In the DRVVAR table, the IDN296 (FFGAIN) is parameterised as synchronous write, and the drive is not AXD or the software version is incompatible.

SOLUTION   Remove the synchronous variable from the DRVVAR table or use an AXD drive with software version v9.10 or higher.

4200   'Error in the Sercos cyclic channel'

AXIS   Logic number of the axis

CAUSE   Error when reading or writing the cyclic channel (invalid telegram or phase other than 4).

SOLUTION   Contact our technical service department.

4201   'Error in the Sercos service channel'

AXIS   Logic number of the axis

VALUE 1   Errors at the Sercos drive. See error codes described in error 4000.

VALUE 2   Errors at the Sercos drive. See error codes described in error 4000.

VALUE 3   Status of the command that caused the error.

CAUSE   Error in the Sercos service channel in the processes launched from the loop:
  • Reading of variables. List of errors.
  • Writing of variables. Feedforward, ACforward, gear preset and KV.
  • Execution of commands. Axis or spindle parking, gear change.

SOLUTION   Analyze the error codes to try to determine the error source. Contact our technical service department.

4202   'Drive Enable (DRENA) missing'

AXIS   Logic number of the axis

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'

AXIS   Logic number of the axis

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'

AXIS   Logic number of the axis

VALUE 1   Errors at the Sercos drive. See error codes described in error 4000.

VALUE 2   Errors at the Sercos drive. See error codes described in error 4000.

CAUSE   Error when executing the command to reset the errors of a drive (ID 99). The loop executes this command 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.

Code.	Meaning.
0x00000002	Broken optical 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   Error 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'**

AXIS   Logic number of the axis

CAUSE   The drive reports an error.

SOLUTION   Analyze error codes. Refer to the drive manual.

4207   **'Drive Status Ready (DRSTAF/DRSTAS) is missing'**

DETECTION   During execution.

CAUSE   Monitoring of DRSTAF, DRSTAS or of the tandem, when the axis is moving.

SOLUTION   Check the enable signals and the status of the drive.

4209   **'Error in the SERCOS ring (optical fiber)'**

VALUE   Error codes that identify the source or sources of the problem.

Code.	Meaning.
0x00000002	Broken optical fiber.

CAUSE   Broken Sercos ring causing a loss of Phase 4.

SOLUTION   Check the following:
- Verify that the optical fiber is properly connected and in good condition.
- Check the parameter settings at the CNC and at the drives. Node selector of the drive (DriveID), parameters LOOPTIME, SERPOWSE, SERBRATE, OPMODE, etc.

4210   **'Error in the SERCOS ring (MST lost)'**

VALUE   Error codes that identify the source or sources of the problem.

Code.	Meaning.
0x00000200	MST loss.

CAUSE   A synchronism message got lost causing the loss of phase 4.

SOLUTION   Ensure connectivity through the whole ring (optical fiber, master and slaves).

4211   **'Error in the SERCOS ring (2 AT's lost)'**

AXIS   Logic number of the axis

VALUE   Error codes that identify the source or sources of the problem.

Code.	Meaning.
0x00000100	AT loss.

CAUSE   Some drive has lost responses causing the loss of phase 4.

SOLUTION   Determine the defective drive and replace the Sercos board or the drive itself. Contact our technical service department.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·168·

4212   'Error in the Sercos ring (AT reception)'

AXIS   Logic number of the axis

VALUE   Error codes that identify the source or sources of the problem.

Code.	Meaning.
0x00008000	AT transmission error. (Optic fiber or drive reset)

CAUSE   Error at the Sercos bus that causes a loss of Phase 4.

SOLUTION   Ensure connectivity through the whole ring (optical fiber, master and slaves).

4213   'SERCOS counter error'

NODE   Sercos ID of counter node.

CODE   Error code of the module. See *"RCS-S module."* on page 306.

CAUSE   The counter reports an error.

SOLUTION   Analyze error codes (See counter manual).

4214   'Error while searching home on the Sercos counter'

NODE   Sercos ID of counter node.

VALUE 1   Sercos drive error Codes:

VALUE 2   Sercos drive error Codes:

VALUE 3   Status of the command that caused the error.

VALUE 4   IDN of the command that caused the error.

CAUSE   Error while homing an axis with remote Sercos feedback.

SOLUTION   Analize the parameter settings of the process; parameter REFPULSE in particular.

4215   Error in the Sercos ring (MDT reception).

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4216   Error in the Sercos ring (synchronism).

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4217   Error in the Sercos ring (MDT write, buffer 0).

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4218   Error in the Sercos ring (MDT write, buffer 1).

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4219   errDoubleMSTMiss.

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4220   errMSTValidMiss.

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4221   errMDTValidMiss.

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4222   errATValidMiss.

DETECTION   On startup or normal operation of the system.

CAUSE   Error on the Sercos bus, due to connection problems, parameterization, problems on the Sercos boards, etc.

SOLUTION   Check Sercos III bus connection. If the error persists, contact Fagor Automation.

4300   'No Sercos variable server has been registered'

DETECTION   At the oscilloscope.

CAUSE   There is a Sercos variable defined in a channel of the oscilloscope, but there is no server of Sercos variables to respond to the request.

SOLUTION   Verify that the CNC powered up correctly.

4301   Sercos variables cannot be accessed.

DETECTION   From the interface.

CAUSE   A Sercos drive variable has been requested by the interface, however, the variable server has not been initialized correctly and it cannot access information on the Sercos variables.

SOLUTION   Verify that the CNC powered up correctly.

4302   Unidentified Sercos variable.

DETECTION   From the interface.

CAUSE   A Sercos drive variable has been requested by the interface, however, the variable is unknown.

SOLUTION   Check that the name of the Sercos variable is correct.

4303   'The Sercos variable has no sets"

DETECTION   At the oscilloscope.

CAUSE   There is a Sercos variable defined in a channel of the oscilloscope, whose syntax indicates the set, but the variable has no sets.

SOLUTION   Eliminate the set indicator from the name of the variable.

4500   'Error when initializing the Mechatrolink Hw'

DETECTION   During CNC startup.

CAUSE   The Mechatrolink communication initialization stage could not be completed successfully.

SOLUTION   Verify the setting of parameters for Mechatrolink communication (protocol, data size, number of axes, etc.) have been set properly and the physical condition of the bus (cables, terminating resistors, numbering of each station, etc.).

4501   'Mechatrolink command error'

DETECTION   During the execution of the CNC.

CAUSE   Failed execution of a Mechatrolink command from the master to a particular slave.

SOLUTION   Identify the command that has caused the error and the related slave. Analyze the conditions, if any, so the command can be completed successfully (drive status, voltage supply, power, motor connection, etc.).

4502   'Timeout when initializing the Mechatrolink'

DETECTION   During CNC startup.

CAUSE   The initialization of the Mechatrolink communication has not been completed.

SOLUTION   Verify the setting of parameters for Mechatrolink communication (protocol, data size, number of axes, etc.) have been set properly and the physical condition of the bus (cables, terminating resistors, numbering of each station, etc.).

4503   'Mechatrolink command alarm'

DETECTION   During the execution of the CNC.

CAUSE   The execution of a Mechatrolink command from the master to a particular slave causes an alarm situation in that device.

SOLUTION   Identify the reported alarm code and see the manual of the Mechatrolink slave for more detailed information.

CNCelite
8058   8060
8065   8070

4504 **'Communication error at the Mechatrolink Bus'**

DETECTION   During the execution of the CNC.

CAUSE   Failed cyclic data exchange between the Mechatrolink master and a slave.

SOLUTION   Check the hardware of the master and the physical condition of the bus (cables, terminating resistors, numbering of each station, etc.).

4505 **'Mechatrolink command warning'**

DETECTION   During the execution of the CNC.

CAUSE   The execution of a Mechatrolink command from the master to a particular slave causes a warning situation in that device.

SOLUTION   Identify the reported warning code and see the manual of the Mechatrolink slave for more detailed information.

4506 **'Power missing at the Mechatrolink slave'**

DETECTION   During CNC execution, once it gets into the interpolated movement state.

CAUSE   Some Mechatrolink slave cannot complete a motion command for lack of power.

SOLUTION   Check the power connection of the electrical cabinet and the PLC program for enabling the drives.

4507 **'Mechatrolink subcommand alarm'**

DETECTION   During the execution of the CNC.

CAUSE   The execution of a Mechatrolink subcommand from the master to a particular slave causes an alarm situation in that device.

SOLUTION   Identify the reported alarm code and see the manual of the Mechatrolink slave for more detailed information.

4508 **'Mechatrolink subcommand warning'**

DETECTION   During the execution of the CNC.

CAUSE   The execution of a Mechatrolink subcommand from the master to a particular slave causes a warning situation in that device.

SOLUTION   Identify the reported warning code and see the manual of the Mechatrolink slave for more detailed information.

4600 **Error when initializing bus hardware.**

DETECTION   During CNC startup.

CAUSE   Internal error in the initialization of the bus hardware.

SOLUTION   Contact Fagor Automation.

4601 **Error in bus node connection.**

DETECTION   During CNC startup.

CAUSE   Communication cannot be established during the bus start-up phase.

SOLUTION   Check the wiring, connections and power supply of the slaves. In the case of the EtherCAT bus, check the configuration file.

4602 **Error during node search.**

DETECTION   During CNC startup.

CAUSE   During the communication bus start-up, not all slaves expected to be detected could be detected.

SOLUTION   Check the wiring, connections and power supply of the slaves. In the case of the EtherCAT bus, check the configuration file.

4603 **Error during bus configuration phase.**

DETECTION   During CNC startup.

CAUSE   An internal error occurred during the configuration phase of slaves connected to the bus.

SOLUTION   Contact Fagor Automation.

4604 **Error in bus pre-operations phase.**

DETECTION   During CNC startup.

CAUSE   An internal error occurred during the configuration phase of slaves connected to the bus.

SOLUTION   If EtherCAT bus, check the configuration file. If this is not the case, please contact Fagor Automation.

4605   Error in bus operation phase.

DETECTION	During CNC startup.
CAUSE	An internal error occurred during the configuration phase of slaves connected to the bus.
SOLUTION	One of the slaves connected to the bus may be in an error state. Review additional errors that the CNC may be reporting to diagnose what may be causing the error. If there is no slave in error, please contact Fagor Automation.

4606   Error in bus preparation command.

DETECTION	During CNC startup.
CAUSE	An internal error occurred during the preparation of the bus parameters.
SOLUTION	Contact Fagor Automation.

4607   Error in bus update command.

DETECTION	During CNC startup.
CAUSE	An internal error occurred during activation of the bus parameters.
SOLUTION	Contact Fagor Automation.

4608   Error during feedback initialization.

DETECTION	During CNC startup.
CAUSE	An internal error occurred during catchment recognition.
SOLUTION	Contact Fagor Automation.

4609   Error during bus reset.

DETECTION	During CNC power-up or after a reset.
CAUSE	An internal error occurred during the bus reset process.
SOLUTION	Contact Fagor Automation.

4610   Bus Error, no connection.

DETECTION	During CNC start-up or execution.
CAUSE	Physical connection on the Sercos III bus has been lost; cable disconnected or slave switched off without power supply.
SOLUTION	Check the wiring, connections and power supply of the CNC and slaves.

4611   Bus Error, non-valid message for phase 0.

DETECTION	During CNC power-up or after a reset.
CAUSE	An error occurred during communication with slaves on the Sercos III bus.
SOLUTION	Check the wiring, connections and power supply of the slaves. If the error persists, contact Fagor Automation.

4612   Bus Error, message not received during phase 0.

DETECTION	During CNC power-up or after a reset.
CAUSE	An error occurred during slave detection on the Sercos III bus. One of the slaves has an error or is disconnected.
SOLUTION	Check the wiring, connections and power supply of the slaves. If the error persists, contact Fagor Automation.

4613   Bus Error, incorrect number of nodes detected.

DETECTION	During CNC startup.
CAUSE	An internal error occurred during the bus node detection phase.
SOLUTION	Correct the configuration of the simulated bus. If the error persists, contact Fagor Automation.

4614   Undefined Bus Error.

DETECTION	During CNC start-up, during execution or after a reset.
CAUSE	An unexpected error occurred on the Sercos III bus.
SOLUTION	Contact Fagor Automation.

4615   Bus Error, timeout.

DETECTION	During CNC power-up or after a reset.
CAUSE	A timeout error occurred during start-up or reset of the communications bus.
SOLUTION	Check the wiring, connections and power supply of the slaves. In the case of the EtherCAT bus, check the configuration file. If the error persists, contact Fagor Automation.

4616   Node Error, unidentified module.

DETECTION   During CNC startup.

CAUSE   An internal error occurred during the bus node detection phase.

SOLUTION   Correct the configuration of the simulated bus. If there is no simulated bus, please contact Fagor Automation.

4630   Serial number configuration error.

DETECTION   During CNC startup.

CAUSE   Error reading the serial numbers of a Sercos III node or incorrect serial numbers.

SOLUTION   Contact Fagor Automation.

4631   Connection name not found.

DETECTION   During CNC start-up, machine parameter validation or after a reset.

CAUSE   The connection specified in the parameter does not exist.

SOLUTION   Check the CNC connections section and make sure that all the connections to be used are present.

4632   Hardware node id unknown.

DETECTION   During CNC startup.

CAUSE   An internal error occurred in the detection of Sercos III nodes. The CNC version does not recognize one of the nodes.

SOLUTION   Contact Fagor Automation.

4633   Node communication error.

DETECTION   During CNC power-up or after a reset.

CAUSE   A communication error occurred with a Sercos III slave after a reset, reading of the version or reading of the Hardware ID.

SOLUTION   Check the wiring, connections and power supply of the slaves. If the error persists, contact Fagor Automation.

4634   Node not found in topology.

DETECTION   During CNC startup.

CAUSE   Sercos III node serial numbers do not match. The node has been exchanged or replaced.

SOLUTION   Contact Fagor Automation.

4635   Undefined node.

DETECTION   During CNC start-up, during execution or after a reset.

CAUSE   An unexpected error occurred in one of the Sercos III nodes.

SOLUTION   Contact Fagor Automation.

4636   Hardware data not found.

DETECTION   During CNC startup.

CAUSE   An internal error occurred in the reading of the hardware data file.

SOLUTION   Contact Fagor Automation.

4637   Node series number error.

DETECTION   During CNC startup.

CAUSE   Error reading the serial numbers of a Sercos III node or incorrect serial numbers.

SOLUTION   Contact Fagor Automation.

4638   Node not detected.

DETECTION   During CNC startup.

CAUSE   A topology node was not detected during CNC start-up.

SOLUTION   Update the topology according to the connected nodes (remove node from the topology) or check wiring, connections and power supply of the bus slaves. If the error persists, contact Fagor Automation.

4639   Undefined node found.

DETECTION   During CNC startup.

CAUSE   A node has been detected that is not defined in the system topology.

SOLUTION   Update the topology by adding the detected node or disconnect it from the bus if you do not want to use it.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

4640   Node structure version not compatible.

DETECTION   During CNC startup.

CAUSE   A Sercos III node has been detected with a version incompatible with the CNC version.

SOLUTION   Contact Fagor Automation.

4641   Integrated safety on node configuration error.

DETECTION   During CNC startup.

CAUSE   An error occurred in the configuration of a safety node.

SOLUTION   Contact Fagor Automation.

4642   Serial number of the powerControl board without engraving, sap-manuf.

DETECTION   During CNC startup.

CAUSE   A node has been detected that is not in topology and has no serial number engraved on the powerControl board.

SOLUTION   Contact Fagor Automation. A valid serial number must be engraved on the powerControl board.

4643   Serial number of powerModule board not engraved, sap-manuf.

DETECTION   During CNC startup.

CAUSE   A node has been detected that is not in topology and has no serial number engraved on the powerModule board.

SOLUTION   Contact Fagor Automation. A valid serial number must be engraved on the powerModule board.

4660   Hiperface status machine error.

DETECTION   During CNC startup.

CAUSE   Internal error in Hiperface communication state machine.

SOLUTION   Contact Fagor Automation.

4661   Hiperface driver communication error.

DETECTION   During CNC startup.

CAUSE   Internal error in Hyperface communication.

SOLUTION   Contact Fagor Automation.

4662   checksum Hiperface error.

DETECTION   During CNC startup.

CAUSE   Internal checksum error in Hyperface communication.

SOLUTION   Contact Fagor Automation.

4663   Hiperface encoder unknown.

DETECTION   During CNC startup.

CAUSE   Unknown hyperface feedback.

SOLUTION   Contact Fagor Automation.

4664   Feedback initialization error.

DETECTION   During CNC startup.

CAUSE   Internal error in feedback initialization.

SOLUTION   Contact Fagor Automation.

4665   Communication error with communications driver.

DETECTION   During CNC startup.

CAUSE   Internal error in feedback communications.

SOLUTION   Contact Fagor Automation.

4666   Virgin Hyperface Encoder without EEPROM format.

DETECTION   During CNC startup.

CAUSE   Unformatted hyperface encoder in EEPROM.

SOLUTION   Contact Fagor Automation.

4670   Communication error with SSI feedback driver.

DETECTION   During CNC startup.

CAUSE   Error in communication with SSI feedbacks.

SOLUTION   Contact Fagor Automation.

4671   SSI crc/checksum error.

DETECTION   During CNC start-up or execution.
CAUSE   Checksum error in communication with SSI feedbacks.
SOLUTION   Contact Fagor Automation.

4672   Internal SSI feedback error.

DETECTION   During CNC start-up or execution.
CAUSE   Internal error in an SSI feedback.
SOLUTION   Contact Fagor Automation.

4673   Error acknowledge feedback SSI.

DETECTION   During CNC start-up or execution.
CAUSE   Acknowledge error in an SSI feedback.
SOLUTION   Contact Fagor Automation.

4674   Error reading digital encoder parameters.

DETECTION   During CNC startup.
CAUSE   Error while reading parameters from a digital encoder or the data read is not correct.
SOLUTION   Contact Fagor Automation.

4675   Digital encoder not supported.

DETECTION   During CNC startup.
CAUSE   An unsupported digital encoder or an unsupported digital protocol has been detected.
SOLUTION   Contact Fagor Automation.

4680   Error configuring reference.

DETECTION   During CNC startup.
CAUSE   An internal error occurred during the configuration of a reference.
SOLUTION   Check the configuration of the CNC connections and make sure that the reference exists. If the error persists, contact Fagor Automation.

4681   Communications error with reference.

DETECTION   During CNC startup.
CAUSE   A communication error has occurred with a reference.
SOLUTION   Check the configuration of the CNC connections. If the error persists, contact Fagor Automation.

4682   Reference identification error.

DETECTION   During CNC startup.
CAUSE   An unsupported reference has been detected.
SOLUTION   Check the configuration of the CNC connections. If the error persists, contact Fagor Automation.

4683   SW Update Error for PwC devices connected to the bus.

DETECTION   During execution.
CAUSE   The execution of a Mechatrolink subcommand from the master to a particular slave causes a warning situation in that device.
SOLUTION   Identify the warning code in the Mechatrolink slave manual for more specific information.

4684   SW of AVR associated with the ordering number does not match the type of engine connected.

DETECTION   During commissioning, connection assignment and/or change in motor or RPS source types.
CAUSE   The motor type connected to the reference does not match the control type of the associated drive.
SOLUTION   Validate the topology.

4703   Axis reference not found.

DETECTION   During the preparation of real-time data.
CAUSE   Axis reference (node identifier) has not been defined correctly.
SOLUTION   Revise the definition of the reference and the node identifier.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·175·

4704   Axis feedback not found.

DETECTION   During the preparation of real-time data.
CAUSE   Axis feedback has not been defined correctly (node identifier).
SOLUTION   Revise the definition of the reference and the node identifier.

4705   A bus variable has not been found.

DETECTION   During the preparation of real-time data.
CAUSE   The bus variable has not been correctly defined.
SOLUTION   Check the definition of the variable.

4706   Node type not allowed.

DETECTION   During bus node detection.
CAUSE   The control does not have the software option required to initialise such a node type.
SOLUTION   You can check the active software options in .

4707   Maximum number of nodes exceeded.

DETECTION   During start-up or validation of machine parameters.
CAUSE   More nodes are defined than allowed.
SOLUTION   Delete nodes in the parameterisation.

4708   Bus configuration file wrong.

DETECTION   During CNC startup.
CAUSE   The actual bus topology does not match the configuration file.
SOLUTION   Check bus topology and connections.

4709   Node has communication problems with upstream node.

DETECTION   During CNC startup.
CAUSE   The connection between nodes is not correct.
SOLUTION   Check physical connections.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

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	Check the number of the timer; valid timers, T1 to 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	Check the number of the counter; valid counters, T1 to T256.

5002 **'CNCRD: Unidentified variable'**

DETECTION	When executing a CNCRD instruction.
CAUSE	The variable does not exist. If the error occurs during start-up, the PLC may be trying to execute a CNCRD instruction before the SERCOSRDY mark is active.
SOLUTION	Check the syntax of the variable. If the error occurs during start-up, condition the reading of the variable to the PLC mark SERCOSRDY being active.

5003 **'CNCWR: Unidentified variable'**

DETECTION	When executing a CNCWR instruction.
CAUSE	The variable does not exist.
SOLUTION	Check the syntax of the variable.

5004 **'Variable without reading permission for PLC'**

DETECTION	When executing a CNCRD instruction.
CAUSE	Reading of a variable that has no reading permission for the PLC.
SOLUTION	Check the permissions of the variable.

5005 **'Variable without writing permission for PLC'**

DETECTION	When executing a CNCWR instruction.
CAUSE	Writing of a variable that has no writing permission for the PLC.
SOLUTION	Check the permissions of the variable.

5006 **'Variable writing from the PLC out of range'**

DETECTION	When executing a CNCWR instruction.
CAUSE	The value assigned to the variable is not valid.
SOLUTION	Check the syntax of the variable and the valid values.

5007 **'Syntax error when writing the variable from the PLC'**

DETECTION	When executing a CNCWR instruction.
CAUSE	The variable does not exist or it has no permission to be written from the PLC.
SOLUTION	Check the syntax and the permissions of the variable.

5008 **'The variable could not be written from the PLC'**

DETECTION	When executing a CNCWR instruction.
CAUSE	Writing a global or local arithmetic parameter that does not exist.
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 a CNCRD instruction.
CAUSE	The variable does not exist or it has no permission to be read from the PLC.
SOLUTION	Check the syntax and the permissions of the variable.

5010 **'Division by zero at the PLC'**

DETECTION   When executing a DVS or MDS instruction.

CAUSE   The denominator of a DVS or MDS operation is 0.

SOLUTION   Check the PLC program. Do not divide by 0.

5013 **'Error when reading the digital inputs of the PLC'**

DETECTION   When reading the digital inputs of the PLC.

CAUSE   Failure to read PLC remote digital inputs.

SOLUTION   Check digital I/O configuration table (NDIMOD parameter) and check bus integrity. If the error persists, contact Fagor Automation.

5014 **'Error when writing the digital outputs of the PLC'**

DETECTION   When writing the digital outputs of the PLC.

CAUSE   Failure to write PLC remote digital outputs. The configuration table of the digital I/Os is invalid (parameter NDOMOD) or the PLC has received a write request before the previous one has been completed.

SOLUTION   Check digital I/O configuration table (parameter NDOMOD), check bus integrity or increase cycle time in parameter PLCFREQ. If the error persists, contact Fagor Automation.

5015 **'The variable could not be read from the PLC'**

DETECTION   When executing a CNCRD instruction.

CAUSE   Reading a global or local arithmetic parameter that does not exist.

SOLUTION   Check the settings of global and local parameters as well as the PLC program.

5016 **'Reading value out of range'**

DETECTION   When executing a CNCRD instruction.

CAUSE   Data read out of range.

SOLUTION   Check the syntax of the instruction.

5017 **'Null CNCEX block'**

DETECTION   When executing a CNCEX instruction.

CAUSE   The block programmed in the CNCEX instruction is null or the programmed channel 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 mark programmed in the CNCEX instruction is already active when executing the instruction. It may be because the instruction has been programmed wrong or because the channel is busy with another CNCEX instruction.

SOLUTION   Check, in the PLC program, the logic of the CNCEX instruction and of the marks used.

5019 **'CNCEX: Execution channel not available'**

DETECTION   When executing a CNCEX instruction.

CAUSE   The channel programmed in the CNCEX instruction is not available. The channel is executing another block, another program or is in a wrong state.

SOLUTION   Check the status of the channel programmed in the CNCEX instruction.

5020 **'CNCEX: Execution incomplete'**

DETECTION   When executing a CNCEX instruction.

CAUSE   The CNCEX instruction cannot execute the block in the indicated channel.

SOLUTION   Check the status of the channel programmed in the CNCEX instruction.

5021 **'CNCEX: The indicated channel is not a PLC channel'**

DETECTION   When executing a CNCEX instruction.

CAUSE   The channel programmed in the CNCEX instruction is not a PLC channel.

SOLUTION   Check the syntax of the instruction. Check the type of channel in the machine parameters (parameter CHTYPE).

5022   **'PLC: Too many CNCRD and CNCWR of asynchronous variables'**

DETECTION   While executing the CNCRD and CNCWR commands of asynchronous variables as well as while executing CNCEX commands.

CAUSE   Too many asynchronous requests from the PLC.
- CNCEX command.
- CNCRD and CNCWR commands of asynchronous variables of the drives.
- CNCWR command of tool variables.
- CNCRD command of variables of tools that are not in the tool magazine.

SOLUTION   The values shown in the warning are the numbers of the PLC marks that control the CNCRD, CNCWR and CNCEX processes that cause the error. To remove the error, check the PLC program so as to prevent so many asynchronous requests from happening at the same time.

5023   **'Error when reading local digital inputs'**

DETECTION   Error when reading local digital inputs.

CAUSE   Failure to read local PLC digital inputs.

SOLUTION   Check the installation of the CNC application and drivers. If the error persists, contact Fagor Automation.

5024   **'Error when writing local digital outputs'**

DETECTION   Error when writing local digital outputs.

CAUSE   Failure to write local PLC digital outputs.

SOLUTION   Check the installation of the CNC application and drivers. If the error persists, contact Fagor Automation.

5025   **'PLC counter disabled'**

DETECTION   During CNC startup.

CAUSE   The counter is disabled, from the PLC program or from monitoring.

SOLUTION   After this warning, the PLC forces the disabling of the counter (CEN = 1) automatically.

5026   **'CNCWR: The variable only admits a 0 value'**

DETECTION   While executing CNCWR commands.

CAUSE   The PLC has tried to write a value other than 0 in a variable that only admits a 0 (zero) value.

SOLUTION   Check the PLC program. Write 0 (zero) in the variable or eliminate the CNCWR command.

5027   **'Short-circuit at the local analog output (report from PLC)'**

DETECTION   The PLC has detected an over-current alarm when writing the local analog output.

CAUSE   Short-circuit at the local analog output of the CNC.

SOLUTION   Check the connection of the analog output of the CNC.

5028   **Remote resource error.**

DETECTION   When accessing the resources of the remote module.

CAUSE   Alarm in a remote module, when reading data.

SOLUTION   Check the connection of the remote modules.

5029   **Remote resource error.**

DETECTION   When accessing the resources of the remote module.

CAUSE   Alarm in a remote module when writing data.

SOLUTION   Check the connection of the remote modules.

5030   **Remote resource configuration error.**

DETECTION   During system startup.

CAUSE   Error when configuring remote modules.

SOLUTION   Check the configuration of the remote modules.

5031   **Remote resource internal configuration error.**

DETECTION   During system startup.

CAUSE   Error when configuring remote modules.

SOLUTION   Check the configuration of the remote modules.

5032 Error in the order of the configuration commands of the remote resource router.

DETECTION   During system startup.
CAUSE   Error when configuring the remote modules. The order of the initialization commands of the resource router is wrong.
SOLUTION   Check the configuration of the remote modules.

5033 Error in the configuration of the remote resource router.

DETECTION   During system startup.
CAUSE   Error when configuring the remote modules. A remote module is using a channel that does not exist.
SOLUTION   Check the configuration of the remote modules.

5034 Error in the setting of a remote resource.

DETECTION   During system startup.
CAUSE   Error when configuring the remote modules. A parameter of the remote resources is not correct.
SOLUTION   Check the configuration of the remote modules.

5035 Open circuit error in a remote analog output.

DETECTION   During PLC operation.
CAUSE   The circuit connected to a remote analogue output has been opened.
SOLUTION   Check the connection of the remote analog output.

5036 Short circuit error in a remote analog output.

DETECTION   During PLC operation.
CAUSE   A short circuit has occurred on a remote analogue output.
SOLUTION   Check the connection of the remote analog output.

5037 A remote analog output is incorrectly configured.

DETECTION   During PLC operation.
CAUSE   The configuration of a remote analog output is wrong.
SOLUTION   Check remote analogue output configuration.

5038 A remote analog output has detected a power failure.

DETECTION   During PLC operation.
CAUSE   The power supply of the remote analog output is wrong.
SOLUTION   Check the power supply of the remote analog output.

5039 A remote analog output has detected an error in the module.

DETECTION   During PLC operation.
CAUSE   The remote analogue output module is not working properly.
SOLUTION   Check remote module.

5040 A remote analog output has detected an unknown error.

DETECTION   During PLC operation.
CAUSE   The remote analogue output module has an error.
SOLUTION   Check remote module.

5041 A remote analog input has detected a range exceeded error.

DETECTION   During PLC operation.
CAUSE   The voltage applied to the analog input is greater than the permitted range.
SOLUTION   Check the circuit connected to the remote analog input.

5042 A remote analog input has detected a minimum range exceeded error.

DETECTION   During PLC operation.
CAUSE   The voltage applied to the analog input is greater than the permitted range.
SOLUTION   Check the circuit connected to the remote analog input.

5043 A remote analogue input has detected a conversion error.

DETECTION   During PLC operation.
CAUSE   The analogue input indicates that it cannot read correctly.
SOLUTION   Check remote module.

5044   A remote analog input is incorrectly configured.

DETECTION   During PLC operation.

CAUSE   The configuration of a remote analog input is wrong.

SOLUTION   Check the configuration of the remote analog input.

5045   A remote analog input has detected a power failure.

DETECTION   During PLC operation.

CAUSE   The power supply of the remote analog input is wrong.

SOLUTION   Check the remote analog input power supply.

5046   A remote analog input has detected an error in the module.

DETECTION   During PLC operation.

CAUSE   The remote analogue input module is not working properly.

SOLUTION   Check remote module.

5047   A remote analog input has detected an unknown error.

DETECTION   During PLC operation.

CAUSE   The remote analogue input module has an error.

SOLUTION   Check remote module.

5048   Open circuit error for a remote temperature sensor.

DETECTION   During PLC operation.

CAUSE   The circuit connected to a remote temperature sensor has been opened.

SOLUTION   Check the connection of the remote temperature sensor.

5049   A remote temperature sensor has detected a minimum range exceeded error.

DETECTION   During PLC operation.

CAUSE   Temperature sensor temperature is lower than the allowed range.

SOLUTION   Check the connection of the remote temperature sensor.

5050   A remote temperature sensor has detected a conversion error.

DETECTION   During PLC operation.

CAUSE   The remote temperature sensor cannot perform the reading correctly.

SOLUTION   Check the connection of the remote temperature sensor.

5051   A remote temperature sensor is incorrectly configured.

DETECTION   During PLC operation.

CAUSE   The configuration of a remote temperature sensor is wrong.

SOLUTION   Check the configuration of the remote temperature sensor.

5052   A remote temperature sensor has detected a power failure.

DETECTION   During PLC operation.

CAUSE   The power supply of the remote temperature sensor is wrong.

SOLUTION   Check the power supply of the remote temperature sensor.

5053   A remote temperature sensor has detected an error in the module.

DETECTION   During PLC operation.

CAUSE   The remote temperature sensor module is not working properly.

SOLUTION   Check remote temperature sensor module.

5054   A remote temperature sensor has detected an unknown error.

DETECTION   During PLC operation.

CAUSE   The remote temperature sensor module has an error.

SOLUTION   Check remote temperature sensor module.

5055   A remote temperature sensor has detected a range exceeded error.

DETECTION   During PLC operation.

CAUSE   Temperature sensor temperature is lower than the allowed range.

SOLUTION   Check the connection of the remote temperature sensor.

5056   Execution error in the handling of a remote analog output.

DETECTION   During PLC operation.
CAUSE   Data processing error.
SOLUTION   Internal error. If the error persists, contact Fagor Automation.

5057   Execution error in the handling of a remote analog input.

DETECTION   During PLC operation.
CAUSE   Data processing error.
SOLUTION   Internal error. If the error persists, contact Fagor Automation.

6000-6999

6000 **'Feedback alarm'**

DETECTION   During execution.

CAUSE   Feedback alarm on an analog axis (parameter FBACKAL).

For differential TTL signal.
- One or more cables broken (A, B signals or their inverted signals).
- Feedback input disconnected at the counter.

For differential sinusoidal signal.
- 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 (approximately).
- Input signal amplitude under 0.4 Vpp (approximately).
- Too much phase shift between A and B signals (in quadrature, in theory)

For non-differential signals, the feedback alarm must be disabled.

SOLUTION   Check the cables and the connection of the feedback inputs. If the error persists, contact the Service Department.

6001 **'Tendency test alarm activated'**

DETECTION   When an axis runs away and the tendency test monitoring is activated.

CAUSE   Positive feedback on the axis for a longer period of time than what is set by parameter ESTDELAY.

SOLUTION   Adjust the sign of the command (parameter AXISCHG), the feedback (LOOPCHG) and the time assigned to parameter ESTDELAY.

6002 **'The PLC program is not running'**

DETECTION   During CNC start-up or while monitoring the PLC program.

CAUSE   The PLC program is not running.
- Installation of a new software version.
- The user has stopped the PLC and forgot to set it running again.

SOLUTION   Set the PLC program running. Compile the PLC program if necessary.

6003 **'External emergency activated'**

DETECTION   During CNC start-up or while monitoring the PLC program.

CAUSE   The PLC mark _EMERGEN is deactivated.

SOLUTION   Check the status of the emergency buttons. Check the logic of the _EMERGEN signal in the PLC program.

6004 **'Error when initializing the BUS CAN'**

DETECTION   During CNC startup.

CAUSE   Error when initializing the CAN on PC start-up in order to start the CAN keyboard. The error codes being are different for the CANfagor bus and for the CANopen bus.

Error codes for the CANopen bus.

Code.	Meaning.
14	Failure when resetting the micro-processor of the COMPCI (poor micro-processor performance, bad contacts, etc.).
15	Problems in the connection or configuration of the CANopen slaves.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

Error codes for the CANfagor bus.

Code.	Meaning.
-1 / -2 / -3	165 in an error state or does not respond (problem in the COMPCI).
-4 / -5	Different memory structure for the PC and for the COMPCI (possibly different versions).
-6	Error in EnableCyclicRead(MD_JOG).
-7	Problems when identifying nodes (same node group and number).
-8	Problems when identifying nodes (an extra node has been found).
-9	Wrong IniCan accessing password.
-10	Failure during the hardware test of the COMPCI.
-11	Failure during the hardware test of the remote node.

SOLUTION   If the error persists, contact the Service Department.

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. If the error persists, contact the Service Department.
- Check that the length of the CAN cable is the right one for the frequency.
- Check in diagnosis mode that the CNC recognizes all the modules.
- Use the itfcCAN.exe tool to check the CAN bus (only service technicians).

6006   'One or several CAN nodes do not respond'

DETECTION   During execution.

CAUSE   One or several remote nodes do not respond due to a reset, shortcircuit, poor performance, etc. The error codes being are different for the CANfagor bus and for the CANopen bus. For the CANopen bus, the error window indicates which module is causing the error.

Error codes for the CANfagor bus.

Code.	Meaning.
-1 / -6	More than three nodes lost.
-3	1 node lost.
-4	2 node lost.
-5	3 node lost.

Error codes for the CANopen bus.

Code.	Meaning.
2	Timeout (the node does not respond).
3	Toggle bit error.
4	The node responds, but its state is not right.

SOLUTION   Check in diagnosis mode that the CNC recognizes all the modules. Check the pinout of the CAN cable. If the error persists, contact the Service Department.

6007   'Watchdog in the COMPCI'

DETECTION   During execution.

CAUSE   The COMPCI does not respond; maybe due to poor performance, contact, etc.

SOLUTION   If the error persists, contact the Service Department.

6008   'Watchdog in the PC + COMPCI'

DETECTION   During execution.

CAUSE   The COMPCI and the PC do not respond due to poor performance, bad contacts, losses of RT IT, etc.

SOLUTION   Check in diagnosis mode the CAN bus configuration and whether there are IT losses. If the error persists, contact the Service Department.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·184·

6009   **'Watchdog at the PC'**

DETECTION   During execution.

CAUSE   The PC does not respond due to loss of RT IT.

SOLUTION   Check in diagnosis mode whether there are IT losses and the configuration of the CAN bus. If the error persists, contact the Service Department.

6010   **'Error in COMPCI processes'**

DETECTION   During execution.

CAUSE   Various errors in the processes managed by the COMPCI.

SOLUTION   If the error persists, contact the Service Department.

6011   **'CAN error in remote node'**

DETECTION   During execution.

CAUSE   Error reported by a node. The error codes being are different for the CANfagor bus and for the CANopen bus. The error window indicates which module is causing the error.

Error codes for the CANfagor bus.

Code.	Meaning.
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, short-circuit, watchdog, etc.).
13	Message transmission failure.

Type of module that generates the error for the CANfagor bus.

Module.	Meaning.
1	Analog outputs.
2	Counters.
3	Digital outputs.
4	Digital inputs.
5	Analog inputs.
7	PT100 inputs.
8	CAN keyboard handwheel.
9	Probes.
10	Jog keyboard.
11	Alphanumeric keyboard.
12	One of the I/O nodes failed.

SOLUTION   Contact our technical service department.

FAGOR
AUTOMATION

CNCelite
8058   **8060**
8065   **8070**

REF. 2209

6011 **'CANopen node report: Undetermined emergency.'**
'CANopen node report: Generic emergency.'
'CANopen node report: Current emergency.'
'CANopen node report: Over-current at the outputs.'
'CANopen node report: Voltage emergency.'
'CANopen node report: Supply voltage missing.'
'CANopen node report: Temperature emergency.'
'CANopen node report: Hardware emergency.'
'CANopen node report: PT100 input broken or disconnected.'
'CANopen node report: Voltage missing at the outputs.'
'CANopen node report: Software emergency.'
'CANopen node report: Error in internal code.'
'CANopen node report: Emergency at additional modules.'
'CANopen node report: Communication emergency.'
'CANopen node report: Period too long between synchronisms.'
'CANopen node report: Overrun, lost message.'
'CANopen node report: Bus errors, passive bus.'
'CANopen node report: Presence control failure.'
'CANopen node report: Bus-off recovery.'
'CANopen node report: Protocol error.'
'CANopen node report: PDO too short.'
'CANopen node report: PDO too long.'
'CANopen node report: External emergency.'
'CANopen node report: OEM specific emergency.'

DETECTION   During execution.

CAUSE   This error indicates that a CANopen node has sent an emergency message. The error text depends on the specific code of the emergency being sent (value 3).

VALUE 1   Node number.

VALUE 2   Error codes for the CANopen bus (bit-based codes).

Code.	Meaning.
0x01	Generic error.
0x02	Current error.
0x04	Voltage error.
0x08	Temperature error.
0x10	Communication error.
0x20	Device error.
0x40	Reserved.
0x80	OEM specific error.

VALUE 3   Emergency table for the CANopen bus (bit-based codes).

Emergency.	Meaning.
0x0000	Corrected error.
0x1000	Generic error.
0x2000	Current.
0x2100	Current, device input side.
0x2200	Current inside the device.
0x2300	Current, device output side.
0x2310	Over-current at the outputs.
0x3000	Voltage.
0x3100	Mains voltage.
0x3200	Voltage inside the device.
0x3300	Output voltage.
0x4000	Temperature.
0x4100	Room (ambient) temperature.
0x4200	Temperature inside the device.
0x5000	Hardware of the device.

Emergency.	Meaning.
0x5030	PT100 broken or not connected.
0x5112	Output supply voltage.
0x6000	Software of the device.
0x6100	Internal software.
0x6200	User software.
0x6300	Data set.
0x7000	Additional modules.
0x8000	Monitoring.
0x8100	Communication.
0x8110	Lost messages.
0x8120	Too many errors in the bus.
0x8130	Presence control error detected by the node.
0x8140	Restored from BUS OFF.
0x8200	Protocol error.
0x8210	PDO not processed due to an error in its length.
0x8220	PDO with too many variables.
0x9000	External error.
0xF000	Additional functions.
0xFF00	Specific device.

SOLUTION   Contact our technical service department.

6012   'CAN controller error'

DETECTION   During execution.
CAUSE   The COMPCI detects BUSOFF and the bus drops.
Error codes for the CANopen bus.

Code.	Meaning.
0	System error.
1	CAN error.
2	Error in Tx/Rx queues.

SOLUTION   Check in diagnosis mode that the CNC recognizes all the modules and the CAN error counter; if there are errors, run the pertinent checks to ensure the integrity of the CAN bus. If the error persists, contact the Service Department.

6013   'Timeout when initializing the CAN'

DETECTION   During system startup.
CAUSE   CAN bus initialization failure 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 CAN bus:
  • There are no CAN groups with the same selected address.
  • The node selector of the COMPCI must be 0.
  • Line terminators.
  • Ground connection.
  • CAN cable continuity.
  • CAN cable connectors (if necessary, 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.
  • Use the itfcCAN.exe tool to check the CAN bus (only service technicians).
If the error persists, contact the Service Department.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·187·

6014   'Failed access to DPRAM of the COMPCI'

DETECTION	During execution.
CAUSE	Failed access to the common RAM memory of the COMPCI due to hardware error, bad contact, etc. The CNC and the COMPCI run Read/Write tests of the common RAM on start-up and cyclically.
SOLUTION	The diagnosis mode offers additional information about the failure. Contact our technical service department.

6015   'CAN error counter exceeded level 1'

DETECTION	During execution.
CAUSE	The (Rx/Tx) error frame counter exceeds level 1 (96).
SOLUTION	Make the relevant checks to assure the integrity of the CAN bus. If the error persists, contact the Service Department.

6016   'CAN error counter at a critical level'

DETECTION	During execution.
CAUSE	The (Rx/Tx) error frame counter exceeds the critical level (127).
SOLUTION	Make the relevant checks to assure the integrity of the CAN bus. If the error persists, contact the Service Department.

6017   'CAN controller's FIFO overrun'

DETECTION	During execution.
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'

DETECTION	During execution.
CAUSE	Overflow at the COMPCI's receiving FIFO. Possible loss of received messaged.
SOLUTION	Contact our technical service department.

6019   'CAN reading timeout'.

DETECTION	During execution.
CAUSE	Failed cyclic reading of digital and analog inputs, counters and keyboard handwheels. A node did not send the message in time.
SOLUTION	Check in diagnosis mode that the CNC recognizes all the modules and the CAN error counter; if there are errors, run the pertinent checks to ensure the integrity of the CAN bus. If necessary, increase the cycle times (parameters LOOPTIME and PLCFREQ). If the error persists, contact the Service Department.

6020   'CAN cycle overlap'

DETECTION	During execution.
CAUSE	Failed cyclic reading of digital and analog inputs, counters and keyboard handwheels. A node did not send the message in time.
SOLUTION	Check in diagnosis mode that the CNC recognizes all the modules and the CAN error counter; if there are errors, run the pertinent checks to ensure the integrity of the CAN bus. If necessary, increase the cycle times (parameters LOOPTIME and PLCFREQ). If the error persists, contact the Service Department.

6021   'Problems in CAN transmission'

DETECTION	During execution.
CAUSE	Failed transmission of digital and analog outputs, etc. Possible bus collapse.
SOLUTION	Check in diagnosis mode that the CNC recognizes all the modules and the CAN error counter; if there are errors, run the pertinent checks to ensure the integrity of the CAN bus. If necessary, increase the cycle times (parameters LOOPTIME and PLCFREQ). If the error persists, contact the Service Department.

6022   'Error when initializing the Axis counter'

DETECTION	During CNC startup.
CAUSE	The counter does not exist. The error window indicates which counter is causing the error.
SOLUTION	Check the machine parameters. If the error persists, contact the Service Department.

6023 **'Error when initializing the handwheel counter'**

DETECTION	During CNC startup.
CAUSE	The counter does not exist. The error window indicates which counter is causing the error.
SOLUTION	Check the machine parameters. If the error persists, contact the Service Department.

6024 **'Error when initializing the keyboard handwheel'**

DETECTION	During CNC startup.
CAUSE	The handwheel input does not exist. The error window indicates which handwheel is causing the error.
SOLUTION	Check the machine parameters. If the error persists, contact the Service Department.

6025 **'Checksum error at the COMPCI'**

DETECTION	During execution.
CAUSE	Error in the checksum of the data of the digital or analog outputs sent from the PC to the COMPCI through the common RAM memory. Error codes for the CANfagor bus.

Code.	Meaning.
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. The following tests are run on start-up: Flash, common RAM, external RAM, internal RAM and CAN controller.
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. The following tests are run on start-up: Flash, RAM and CAN controller.
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.

6028 **'Error when accessing the RAM of the SERCON'**

DETECTION	On system startup or during execution.
CAUSE	Failure when testing the common RAM of the SERCON executed on CNC start-up (R/W accesses in Byte/Word modes). The CNC runs this test on power-up and in every cycle of the Sercos, during phase 4, to check the integrity of the data RAM memory of the SERCON.
SOLUTION	Hardware problem on the Sercos piggy-back or in the connection to the PCI BUS Contact our technical service department.

6029 **'Position increment limit exceeded at the CNC'**

DETECTION	During execution.
CAUSE	Excessive position increment in 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'**

DETECTION	During execution.
CAUSE	Excessive position increment in 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 system startup.
CAUSE	Incompatible software version at CNC and COMPCI or remote modules.
SOLUTION	Update the software of the COMPCI and remote modules.

6032   'Error in the configuration of the CAN keyboard'

DETECTION   During execution.
CAUSE   The 165 does not respond or there are CAN communication problems.
SOLUTION   Check the version of the COMPCI and the connection through the CAN bus.

6033   The hardware configuration does not match that saved in the file

DETECTION   On system startup or during execution.
CAUSE   The hardware configuration saved in the disk is not the same as the one detected on start-up. Possible changes in the CAN configuration that have not been saved, undesired changes in the machine parameters, etc.
SOLUTION   In diagnosis mode, verify that the detected CAN configuration coincides with the real one; if not, save the configuration. Otherwise, check whether the parameters have been set wrong or there is an error in the detection of the nodes.

6034   'Reset in CAN node'

DETECTION   During execution.
CAUSE   A CAN node has been reset. The error window indicates the node number.
SOLUTION   Verify the node is properly supplied. If there are no problems at the bus, replace the node. Contact our technical service department.

6035   'Error when initializing local I/O'

DETECTION   During system startup.
CAUSE   More local digital outputs have been configured at the CNC than there actually are.
SOLUTION   Set a maximum of 8 local digital outputs.

6036   'There is no voltage supply for local I/O'

DETECTION   During execution.
CAUSE   There is no voltage supply at the local I/O (central units ICU and MCU).
SOLUTION   Supply local I/O with 24 V.

6037   'LOOPTIME parameter not valid for CAN configuration'

DETECTION   During system startup.
CAUSE   The value of parameter LOOPTIME is too small or is wrong.
SOLUTION   Verify that the value assigned to parameter LOOPTIME is a multiple of 0.5 ms and greater than 1.5 ms. Increase the value if the CAN configuration so advises.

6038   'Too many CAN elements in the system'

DETECTION   During system startup.
CAUSE   An element of the CAN bus exceeds the maximum permitted; number of analog inputs or outputs, digital inputs or outputs or counters.
SOLUTION   Turn the CNC off and remove the excessive elements from the bus.

6039   'Error when enabling the PT100 input'

DETECTION   During system startup.
CAUSE   The parameters of one of the PT100 inputs have been set wrong.
SOLUTION   Check the general machine parameters NPT100 and PT100.

6040   'Error when reading the local jog'

DETECTION   During the execution of the CNC.
CAUSE   Repeated error when reading the local jog module of the central unit ICU MCU.
SOLUTION   Check the hardware and the communication between the local jog and the central unit.

6041   'Detected keyboard configured in 15" mode'

DETECTION   After a CNC reset.
CAUSE   The CNC has detected an 8065 keyboard (qwerty or jog) configured to communicate with versions older than V04.10.
SOLUTION   Identify the 8065 keyboards (qwerty and jog) and configured them correctly. Contact our technical service department.

6042   'Over-current alarm'

DETECTION   The loop has detected an over-current alarm when reading the local feedback input.
CAUSE   Short-circuit at the local feedback of the CNC.
SOLUTION   Check the proper connection and signal levels of the local feedback of the CNC.

6043 **'Broken-cable alarm'**

DETECTION   The loop has detected a generic alarm indicating broken wire at the local feedback input.

CAUSE   Some signal of the local feedback of the CNC is missing (broken cable).

SOLUTION   Check the proper connection and signal levels of the local feedback of the CNC.

6044 **'Alarm for broken A signal cable'**

DETECTION   The loop has detected that the A signal cable of the local feedback is broken.

CAUSE   The local feedback of the CNC does not receive the A signal.

SOLUTION   Check the A signal of the local feedback of the CNC.

6045 **'Alarm for broken B signal cable'**

DETECTION   The loop has detected that the B signal cable of the local feedback is broken.

CAUSE   The local feedback of the CNC does not receive the B signal.

SOLUTION   Check the B signal of the local feedback of the CNC.

6046 **'Alarm for broken I0 signal cable'**

DETECTION   The loop has detected that the I0 signal cable of the local feedback is broken.

CAUSE   The local feedback of the CNC does not receive the I0 signal.

SOLUTION   Check the I0 signal of the local feedback of the CNC.

6047 **'Alarm for broken AL signal cable'**

DETECTION   The loop has detected that the AL signal cable of the local feedback is broken.

CAUSE   The local feedback of the CNC does not receive the AL signal.

SOLUTION   Check the AL signal of the local feedback of the CNC.

6048 **'External feedback alarm'**

DETECTION   The loop has detected an external alarm at the local feedback.

CAUSE   The feedback device connected to the local feedback reports an alarm.

SOLUTION   Check the feedback device connected to the local feedback of the CNC.

6049 **'Amplitude alarm'**

DETECTION   The loop has detected a Vpp amplitude alarm at the local feedback.

CAUSE   The amplitude of the sinusoidal signal connected to the local feedback exceeds the limits, too small or too large.

SOLUTION   Check the levels of the Vpp signal at the local feedback of the CNC.

6050 **'Short-circuit at the local I/O (over-current alarm)'**

DETECTION   During CNC operation.

CAUSE   Over-current alarm at the local I/O.

SOLUTION   Check the connection and the voltage supply for the local I/O.

6051 **'Short-circuit at the local probe (over-current alarm)'**

DETECTION   During the execution of the CNC.

CAUSE   Over-current alarm at the local probe.

SOLUTION   Check the connection and the voltage supply for the local probe.

6052 **'Short-circuit at the local analog output (report from LR)'**

DETECTION   The loop has detected an over-current alarm when writing the local analog output.

CAUSE   Short-circuit at the local analog output of the CNC.

SOLUTION   Check the connection of the analog output of the CNC.

6053 **'Error when initializing the local jog'**

DETECTION   During system startup.

CAUSE   The local jog panel could not be initialized correctly.

SOLUTION   Check the connection and the voltage supply for the local jog.

6054 **'Error when initializing local handwheels'**

DETECTION   During system startup.

CAUSE   The local handwheel board could not be initialized correctly.

SOLUTION   Check the connection and voltage supply of the handwheel board.

6055 **'Failure when reading local handwheels periodically'**

DETECTION   During the execution of the CNC.
CAUSE   Failure when reading local handwheels cyclically.
SOLUTION   Check the connection and the voltage supply for the local handwheels.

6056 **'Short-circuit at the jog handwheel (over-current alarm)'**

DETECTION   During the execution of the CNC.
CAUSE   Over-current alarm at the handwheel connected to the local jog.
SOLUTION   Check the connection and the voltage supply for the local handwheels.

6057 **'Handwheel feedback alarm'**

DETECTION   During the execution of the CNC.
CAUSE   Feedback alarm at the handwheel connected to the local feedback
SOLUTION   Check the connection and the voltage supply for the local handwheels.

6058 **'Some CANopen node is not in the operational phase'**

DETECTION   After a CNC reset.
CAUSE   When reinitializing the CANopen during a CNC reset, some node has not reached the operational phase.
SOLUTION   Check the connection and configuration of the CANopen nodes.

6059 **Error in the configuration of the analog input.**

DETECTION   During CNC power-up or after a reset.
CAUSE   During the initialization of the CANopen bus, it was not possible to correctly configure any of the parameterized current inputs.
SOLUTION   Check the parameterization of the analog current inputs.

6060 **Error in the configuration of the analog output.**

DETECTION   During CNC power-up or after a reset.
CAUSE   During the initialization of the CANopen bus, it was not possible to correctly configure any of the parameterized current outputs.
SOLUTION   Check the parameterization of the analog current outputs.

6061 **Non-Fagor CANopen nodes. Software option not allowed.**

DETECTION   During the initialization of the CANopen bus, while the CNC was starting-up or after a reset.
CAUSE   During the initialization of the CANopen bus, third party nodes had been detected without the corresponding software option.
SOLUTION   Disconnect any non-Fagor nodes from the CANopen bus or purchase the software option so that it may be used.

6062 **Emergency relay open due to a bus reinitialization error.**

DETECTION   Following a CNC reset or during machine parameter validation.
CAUSE   During machine parameter reset or validation, it was not possible to successfully complete the execution of the process due to an error arising after the reassembly of the CAN bus or the Sercos ring. As a result, the emergency relay has been left open.
VALUE 1   Sercos error during a CNC reset process.
VALUE 2   CAN error during a CNC reset process.
VALUE 3   Sercos and CAN error during a CNC reset process.
VALUE 4   Sercos error during a CNC validation process for the machine parameters.
VALUE 8   CAN error during a CNC validation process for the machine parameters.
VALUE 12   Sercos and CAN error during a CNC validation process for the machine parameters.
SOLUTION   Correct the CAN or Sercos issue preventing the emergency relay from closing.

7000-7999

7001   **'Two consecutive T's in cyclic magazine (M6 required)'**

DETECTION   During execution.
CAUSE   The tool magazine is cyclic and two consecutive tools have been programmed without M06.
SOLUTION   In a cyclic tool magazine, M06 must be programmed after each tool.

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 CNC assumes that the tool is a ground tool, but the machine parameters indicate that ground tools are not allowed.
SOLUTION   Configure the machine to accept ground tools or include the tool in the magazine table.

7003   **'T not defined in the table'**

DETECTION   During execution.
CAUSE   The programmed tool is not defined in tool table.
SOLUTION   All the tools must be defined in the tool table, even the ground tools.

7004   **'Tool rejected or worn out (expired)'**

DETECTION   During execution.
CAUSE   The programmed tool is expired (worn out) or has been rejected by the PLC.
SOLUTION   Repair the tool or use an equivalent tool.

7005   **'Tool rejected or expired and without replacement'**

DETECTION   During execution.
CAUSE   The programmed tool is expired or it has been rejected by the PLC and there are no more tools of the same family in the magazine.
SOLUTION   Repair the tool or enable a tool of the same family that can be used.

7006   **'D not allowed for this tool'**

DETECTION   During execution.
CAUSE   Wrong tool offset. The tool has fewer offsets than the one programmed.
SOLUTION   Program an existing offset or add new ones to the tool.

7007   **'In load mode: T0 not admitted'**

DETECTION   During execution.
CAUSE   Magazine in load mode. T0 cannot be programmed.
SOLUTION   Program the tool to be loaded into the magazine.

7008   **'In load mode: This tool is already loaded'**

DETECTION   During execution.
CAUSE   Magazine in load mode. The programmed tool 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   Magazine in load mode. An M6 cannot be programmed without a tool.
SOLUTION   Program the tool to be loaded and then the M06 function.

7010   **'In load mode: T with two M6'**

DETECTION   During execution.
CAUSE   Magazine in load mode. Two consecutive M6 have been programmed without a tool.
SOLUTION   Program the tool to be loaded and then the M06 function.

7011 **'In load mode: D not admitted alone'**

DETECTION   During execution.

CAUSE   Magazine in load mode. The offset cannot be programmed alone.

SOLUTION   It cannot be programmed like this.

7012 **'In load mode: The tool is either worn out (expired) or rejected'**

DETECTION   During execution.

CAUSE   Magazine in load mode. The tool to be loaded into the magazine is worn out (expired) or rejected.

SOLUTION   Program a valid tool.

7013 **'In load mode: Wrong position or there is no room in the magazine'**

DETECTION   During execution.

CAUSE   Magazine in load mode. 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   Magazine in unload mode. The tool is not in the magazine.

SOLUTION   Program a valid tool.

7015 **'In unload mode: D not allowed with T'**

DETECTION   During execution.

CAUSE   Magazine in unload mode. The offset cannot be programmed next to the tool.

SOLUTION   It cannot be programmed like this.

7016 **'In unload mode: D not admitted alone'**

DETECTION   During execution.

CAUSE   Magazine in unload mode. The offset cannot be programmed alone.

SOLUTION   It cannot be programmed like this.

7017 **'In unload mode: M6 without T'**

DETECTION   During execution.

CAUSE   Magazine in unload mode. An M6 cannot be programmed without a tool.

SOLUTION   Program the tool to be unloaded and then the M06 function.

7018 **'In unload mode: T with two M6'**

DETECTION   During execution.

CAUSE   Magazine in unload mode. Two consecutive M06 have been programmed for the same tool.

SOLUTION   Program the tool to be unloaded and then the M06 function.

7019 **'In setting mode: T0 not admitted'**

DETECTION   During execution.

CAUSE   Magazine in setting mode. T0 cannot be programmed.

SOLUTION   Program a tool other than 0.

7020 **'In Setting mode: D mandatory'**

DETECTION   During execution.

CAUSE   Magazine in setting mode. The offset has not been programmed.

SOLUTION   Program the offset.

7021 **'In Setting mode: D not admitted alone'**

DETECTION   During execution.

CAUSE   Magazine in setting mode. The offset cannot be programmed alone.

SOLUTION   It cannot be programmed like this.

7022 **'In Setting mode: M6 not admitted'**

DETECTION   During execution.

CAUSE   Magazine in setting mode. M06 cannot be programmed.

SOLUTION   It cannot be programmed like this.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

7023 **'In load mode: M6 mandatory'**

DETECTION	During execution.
CAUSE	Magazine in load mode. Two consecutive tools have been programmed without M06.
SOLUTION	Program M06 after each tool.

7024 **'In unload mode: T0 not admitted'**

DETECTION	During execution.
CAUSE	Magazine in unload mode. T0 cannot be programmed.
SOLUTION	Program the tool to be unloaded into the magazine.

7025 **'In unload mode: M6 mandatory'**

DETECTION	During execution.
CAUSE	Magazine in unload mode. Two consecutive tools have been programmed without M06.
SOLUTION	Program M06 after each tool.

7026 **'In unload mode: This tool is already unloaded'**

DETECTION	During execution.
CAUSE	Magazine in unload mode. The tool is not in the magazine.
SOLUTION	Program the tool to be unloaded into the magazine.

7027 **'M6 without T'**

DETECTION	During execution.
CAUSE	An M06 has been programmed without its corresponding tool. There is a machine parameter that indicates to the CNC what to do in this situation, show an error, a warning or do nothing.
SOLUTION	Program a tool and then the M06 function or change the machine parameter to change the behavior of the CNC.

7028 **'T0 not admitted'**

DETECTION	During execution.
CAUSE	T0 not allowed in a turret type magazine. In a turret type magazine, the tool change is done by rotating the turret; it is enough to program the tool.
SOLUTION	It cannot be programmed like this.

7029 **'The operation cannot be carried out: Error at the tool manager'**

DETECTION	During execution.
CAUSE	Magazine in emergency state. The PLC has activated an invalid mark of the maneuver, the PLC has activated the SETTMEM mark or there is an error in the maneuver.
SOLUTION	Correct the tool change maneuver in the PLC program. If the PLC has activated the SETTMEM mark, check why the PLC has set the emergency signal.

7030 **'M6 without T'**

DETECTION	During execution.
CAUSE	An M06 has been programmed without its corresponding tool. There is a machine parameter that indicates to the CNC what to do in this situation, show an error, a warning or do nothing.
SOLUTION	Program a tool and then the M6 function or change the machine parameter to change the behavior of the CNC.

7031 **'T with two M6'**

DETECTION	During execution.
CAUSE	Two consecutive M06 have been programmed for the same tool.
SOLUTION	Program one M06 for each tool.

7032 **Tool manager: Error in the tool change operation.**

DETECTION	During execution.
CAUSE	Error in the tool change sequence in the PLC program.
SOLUTION	Check the PLC program.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

7033   'The PLC generated an emergency in the tool magazine'

DETECTION	When the PLC is running.
CAUSE	The PLC activates the emergency signal at the tool manager.
SOLUTION	Remove the emergency and analyze the causes why the PLC has activated it.

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	Asynchronous or synchronous magazine with changer arm. The magazine is requesting the same tool that is in arm 2.
SOLUTION	Return the tool from the arm to the magazine and go on with the tool change as usual.

7040   'Operation error: First empty arm 1'

DETECTION	During execution.
CAUSE	Asynchronous or synchronous magazine with changer arm. The magazine is requesting the same tool that is in arm 1.
SOLUTION	Return the tool from the arm to the magazine and go on with the tool change as usual.

7041   'No tool is active'

DETECTION	During execution.
CAUSE	A tool offset 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 magazine position has been programmed, but the manager is not in load mode.
SOLUTION	The loading position can only be programmed 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 the active tool in another channel.
SOLUTION	Unload the tool from the other channel.

7044   'The number of cutting edges of the active tool cannot be modified'

DETECTION	During execution.
CAUSE	You have tried to change the number of edges of a tool, but that tool is active in some channel.
SOLUTION	Place the tool in the magazine or on the ground to change the number of edges.

7045   Not enough information for the operation, the magazine is not determined.

DETECTION	During execution.
CAUSE	The programmed tool is not in the magazine.
SOLUTION	Review the tool and magazine table.

7046   **The variable cannot be accessed.**

DETECTION   During execution.
CAUSE   The programmed tool is not accessible.
SOLUTION   Review the tool and magazine table.

7047   **Not enough memory for the total number of tools defined in the magazines.**

DETECTION   During execution.
CAUSE   The number of tools defined in the magazines exceeds the maximum allowed.
SOLUTION   Review the magazine and define a number of tools.

7048   **Error in the ROTATEMZ instruction**

DETECTION   During execution.
CAUSE   A tool change has been programmed in a turret and the #ROTATEMZ instruction has given an error.
SOLUTION   Check the magazine.

7049   **ROTATEMZ: the MZPOS register of the PLC cannot be 0.**

DETECTION   During execution.
CAUSE   The #ROTATEMZ instruction is forcing position 0 of the magazine.
SOLUTION   Check the program. The #ROTATEMZ instruction can only be executed in a turret magazine.

7050   **ROTATEMZ: Wrong position.**

DETECTION   During execution.
CAUSE   The position programmed in #ROTATEMZ is not valid.
SOLUTION   Check the program.

7051   **'Wrong edge number'**

DETECTION   During execution.
CAUSE   The tool has less edges than the one programmed.
SOLUTION   Program an existing edge or add new edges to the tool.

8000-8999

8000   'The simulation kernel is not active'

DETECTION   During system startup.

CAUSE   The simulation kernel has been started up but is not activated'

SOLUTION   Verify that the CNC has started up correctly. Turn the CNC off and back on. If the error persists, contact the Service Department.

8001   'The simulation kernel is not active'

DETECTION   During system startup.

CAUSE   The simulation kernel has been started up but is not activated'

SOLUTION   Verify that the CNC has started up correctly.

8002   'Error when registering a key'

DETECTION   During system startup.

CAUSE   The CNC has not been able to register one of the softkeys or one of the hotkeys (quick access keys)

SOLUTION   Verify that the CNC has started up correctly.

8003   'It is not possible to shut it down while it is in execution or interrupted'

DETECTION   During execution.

CAUSE   The CNC cannot be shut down if it is in execution, interrupted or has an axis occupied in motion control.

SOLUTION   Stop the programs that are in execution.

8004   'It is not possible to shut it down while it is in execution or interrupted'

DETECTION   During execution.

CAUSE   The CNC cannot be shut down if a channel is in execution, interrupted or has an axis occupied in motion control.

SOLUTION   Stop the programs that are in execution.

8005   'It is not possible to change modes while a channel of the group is in execution'

DETECTION   During execution.

CAUSE   It is not possible to change modes while a channel of the group is in execution.

SOLUTION   Interrupt the programs in execution of all the channels of the group.

8006   'Error while initiating the loading process'

DETECTION   During system startup.

CAUSE   Error when preparing to load the COMPCI or the remote modules.

SOLUTION   Verify that the COMPCI is recognized, check the value of parameter CANMODE and the installation of the communications driver.

8007   'Error while loading the program from the COMPCI'

DETECTION   During system startup.

CAUSE   Error when loading the COMPCI.

SOLUTION   Verify that the COMPCI has been properly recognized and installed. Contact our technical service department.

8008   'Error while saving the system configuration'

DETECTION   During system startup.

CAUSE   Error in the CAN configuration in the loading process.

SOLUTION   From Diagnosis mode, check the CAN configuration and compare it with the real one to determine whether there is a problem when detecting nodes or not.

8009   'Error while loading the software of the IO nodes'

DETECTION	During system startup.
CAUSE	Error when loading the I/O nodes.
SOLUTION	Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8010   'Error while loading the keyboard software'

DETECTION	During system startup.
CAUSE	Error when loading the 12"-keyboard node.
SOLUTION	Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8011   'Error while loading the software of the counters'

DETECTION	During system startup.
CAUSE	Error when loading the counter nodes.
SOLUTION	Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8012   'Error while loading the software of the counters'

DETECTION	During system startup.
CAUSE	Error when loading the counter nodes.
SOLUTION	Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8013   'Error while loading the software of compact keyboards'

DETECTION	During system startup.
CAUSE	Error when loading the 10"-keyboard node.
SOLUTION	Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8014   'Error while loading the RIOS software'

DETECTION	During system startup.
CAUSE	Error when loading the RIOS (I/Os CANopen) nodes.
SOLUTION	Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8015   'Shutdown not allowed: Axis in motion or in synchronization'

DETECTION	When trying to shut the CNC down.
CAUSE	The user has tried to shut the CNC down while an axis was moving or synchronized.
SOLUTION	Stop the movement of the axes or de-synchronize the axis.

8016   'Shutdown not allowed: Spindle running'

DETECTION	When trying to shut the CNC down.
CAUSE	The user has tried to shut the CNC down while a spindle is running.
SOLUTION	Stop the spindle.

8017   'Shutdown not allowed: Spindle synchronized'

DETECTION	When trying to shut the CNC down.
CAUSE	The user has tried to shut the CNC down while a spindle is synchronized.
SOLUTION	De-synchronize the spindle.

8018   'The plc cannot be generated while it is in execution or interrupted

DETECTION	When trying to compile the PLC.
CAUSE	'The plc cannot be compiled while the CNC is in execution or interrupted
SOLUTION	Stop the programs that are in execution.

8019   'The plc cannot be generated while any channel is in execution or interrupted'

DETECTION	When trying to compile the PLC.
CAUSE	'The plc cannot be compiled while a channel is in execution or interrupted
SOLUTION	Stop the programs that are in execution.

8020 **The plc cannot be generated: Axis in motion or synchronized'**

DETECTION	When trying to compile the PLC.
CAUSE	The PLC cannot be compiled while an axis is moving or synchronized.
SOLUTION	Stop the movement of the axes or de-synchronize the axis.

8021 **'The plc cannot be generated: Spindle running'**

DETECTION	When trying to compile the PLC.
CAUSE	The PLC cannot be compiled while a spindle is running.
SOLUTION	Stop the spindle.

8022 **'The plc cannot be generated: Spindle synchronized'**

DETECTION	When trying to compile the PLC.
CAUSE	The PLC cannot be compiled while a spindle is synchronized.
SOLUTION	De-synchronize the spindle.

8023 **'Disk memory below recommended minimum'**

DETECTION	During system startup.
CAUSE	The CNC has detected less than 50 MB of free disk memory.
SOLUTION	Check if there is a third-party application running and consuming system resources; if so, close it. Turn the CNC off and back on. If the error persists, contact the Service Department.

8024 **'CNC running in setup mode'**

DETECTION	On system startup, in setup mode.
CAUSE	The CNC has started up in setup mode and is working with unprotected disk.
SOLUTION	If you are not going to do the CNC setup, turn it off and use the DiskMonitor utility to switch to user mode and work with the disk protected. Start the CNC up again.

8025 **'A date change has been detected and it had to be restored'**

DETECTION	During execution.
CAUSE	The CNC has detected a time and date change and it has restored them to their expected values.
SOLUTION	Check whether the time and date are correct. If the error persists, contact the Service Department.

8026 **'OEM protection activated: The changes will be temporary'**

DETECTION	While editing programs, PLC program or machine parameters.
CAUSE	The CNC is in user mode, with the OEM protection activated, and therefore all changes will be temporary and will be lost when shutting the CNC down.
SOLUTION	Turn the CNC off and use the DiskMonitor utility to switch to setup mode. Start the CNC up again.

8027 **'Error when starting up the Kernel'**

DETECTION	During system startup.
CAUSE	The kernel has been started up but is not activated.
SOLUTION	Verify that the CNC has started up correctly. Turn the CNC off and back on. If the error persists, contact the Service Department.

8028 **'Compiler not available'**

DETECTION	During system startup.
CAUSE	The CNC has not found the folder where the PLC compiler has been installed.
SOLUTION	Contact our technical service department.

8029 **'PLC "includes" not available'**

DETECTION	During system startup.
CAUSE	The CNC has not found the folder where the includes needed to compile the PLC have been installed.
SOLUTION	Contact our technical service department.

8030 **'PLC libraries not available'**

DETECTION	During system startup.
CAUSE	The CNC has not found the folder where the libraries needed to compile the PLC have been installed.
SOLUTION	Contact our technical service department.

8031   **'The DRIVERS folder does not exist'**

DETECTION   During system startup.

CAUSE   The CNC has not found the folder that will contain the PLC once it is generated (C:\CNC8070\Drivers).

SOLUTION   In the utilities mode, use the "new directory" option of the softkey menu to create the folder.

8032   **'The USERS folder does not exist'**

DETECTION   During system startup.

CAUSE   The CNC has not found the user work folder (C:\CNC8070\Users).

SOLUTION   In the utilities mode, use the "new directory" option of the softkey menu to create the folder.

8034   **Incomplete setup: No backup copy has been found'**

DETECTION   On system startup, in setup mode.

CAUSE   The CNC has not found any backup copy of the setup.

SOLUTION   When shutting the CNC down, select the option "close and make a backup copy" so the CNC makes a backup copy of the setup.

8035   **'The following access passwords are yet to be defined'**

DETECTION   When starting up an open CNC, in setup mode.

CAUSE   The passwords indicated in the warning have not been defined.

SOLUTION   Having powered the CNC up, access the utilities mode and from the "Passwords" option of the softkey menu, enter the indicated passwords.

8036   **'Switch to user mode or setup mode to work with the CNC'**

DETECTION   On system startup, in setup mode.

CAUSE   It is only possible to operate the CNC in user mode and setup mode.

SOLUTION   Use the DiskMonitor utility to switch to user mode or setup mode and turn the CNC back on.

8037   **'Memory below recommended minimum. It must be restarted'**

DETECTION   During execution.

CAUSE   The CNC has detected that the available RAM memory is below 10% of the total RAM memory.

SOLUTION   Check if there is a third-party application running and consuming system resources; if so, close it. Turn the CNC off and back on. If the error persists, contact the Service Department.

8038   **'MDI in INSPECTION not allowed in the current status'**

DETECTION   During editing and execution in MDI/MDA mode.

CAUSE   It is not possible to get into MDI/MDA mode from the current status of tool inspection.

SOLUTION   Exit tool inspection mode to get into MDI/MDA mode.

8039   **'Error while loading the keyboard software'**

DETECTION   During CNC startup.

CAUSE   Error when loading the software of the keyboard node.

SOLUTION   Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8040   **'Disk memory under recommended minimum for the proper operation of the CNC. Free disk memory before starting the CNC up'**

DETECTION   During CNC startup.

CAUSE   The CNC has detected less than 50 MB of free disk memory.

SOLUTION   Free disk memory is required to start up the CNC.
- Check if there is a third-party application running and consuming system resources; if so, close it.
- Turn the CNC off and back on. If the error persists, contact the Service Department.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·**201**·

8041   **'Memory under the minimum recommended for execution'**

DETECTION   During execution.

CAUSE   The CNC has detected less than 50 MB of free disk memory.

SOLUTION   Free disk memory is required to start up the CNC.
- Check if there is a third-party application running and consuming system resources; if so, close it.
- Turn the CNC off and back on. If the error persists, contact the Service Department.

8042   **'Memory below recommended minimum'**

DETECTION   During execution.

CAUSE   The CNC has detected less than 50 MB of free disk memory.

SOLUTION   Free disk memory is required to start up the CNC.
- Check if there is a third-party application running and consuming system resources; if so, close it.
- Turn the CNC off and back on. If the error persists, contact the Service Department.

8043   **'Insufficient memory to upload the file'**

DETECTION   During editing.

CAUSE   Insufficient memory to upload the file.

SOLUTION   Free memory required to upload the file.
- Check whether the CNC is executing some program or not; if so, interrupt the execution and try uploading the program again.
- Check if there is a third-party application running and consuming system resources; if so, close it.
- Turn the CNC off and back on. If the error persists, contact the Service Department.

8044   **'Program too large to be edited in conversational mode. Switch to standard CNC mode to execute it or execute it'**

DETECTION   During editing or execution.

CAUSE   The conversational program has more than 10 MB.

SOLUTION   Switch to standard CNC mode to edit or execute the program'

8045   **'Temporary validation code expired. Restart the CNC'**

DETECTION   On CNC startup or during execution.

CAUSE   The CNC has detected that its temporary validation code has expired. The CNC will not let START until the user has restarted the CNC.

SOLUTION   Restart the CNC.

8046   **'The temporary validation code has been canceled. Restart the CNC'**

DETECTION   During execution.

CAUSE   The CNC has detected that the user has canceled the temporary validation code. The CNC will not let START until the user has restarted the CNC.

SOLUTION   Reset the CNC.

8047   **'CNC running in UNsafe mode'**

DETECTION   During CNC startup.

CAUSE   The CNC is running in setup mode.

SOLUTION   Turn the CNC off and use the DiskMonitor utility to switch to user mode.

8048   **'Incomplete setup'**

DETECTION   During CNC startup.

CAUSE   One of the following conditions is met in user mode:
- The protection passwords have not been set.
- Backup copy missing.

SOLUTION   Follow these steps to end the setup.

1. Turn the CNC off and use the DiskMonitor utility to switch to setup mode.
2. In utilities mode, go into the passwords window and enter the "general password".
3. Press [ALT][F4] to turn off CNC and select the option "Close and do a backup copy".
4. Use the DiskMonitor utility to switch to setup mode.

8050   **'Error while loading the RIOR software'**

DETECTION   During CNC startup.

CAUSE   Error when loading the software of the RIOR nodes.

SOLUTION   Verify the CAN configuration, system connection and boot versions of remote nodes. If the error persists, contact the Service Department.

8051   **'Error when restoring user session'**

DETECTION   During CNC startup.

CAUSE   One of the files of the user session is not accessible or it is corrupt. The CNC data related to coordinates, zero offsets, parts counter, kinematics, etc. may be lost.

SOLUTION   If the error persists after powering the CNC up several times, contact your supplier.

8052   **The sum of axes and spindles must not exceed the maximum value.**

DETECTION   During CNC startup.

CAUSE   The sum of axes and spindles exceeds the maximum value.

SOLUTION   The CNC has limited the maximum number of axes, spindles and the sum of both. The configuration exceeds the limit of axes plus spindles. Contact Fagor Automation.

8053   **The service period is about to expire. Contact the OEM to obtain an update code.**

DETECTION   On power-up of the CNC or when a day has passed since the last warning.

CAUSE   The CNC has the temporary user license active and there are seven days remaining before the license expires. The CNC will repeat this message at each start-up or when a day has passed since the last warning.

SOLUTION   As the OEM for the code to change or cancel the final license date and enter it in the CNC.

8054   **The term of service has expired. Contact the OEM to obtain an update code.**

DETECTION   On power-up of the CNC, after pressing the softkey [START] or when a day has passed since the last warning.

CAUSE   The temporary user license has expired. The CNC does not accept the [START] key.

SOLUTION   As the OEM for the code to change or cancel the final license date and enter it in the CNC.

8055   **The CNC has been blocked due to a breach of the terms of use. Contact the OEM.**

DETECTION   During CNC startup.

CAUSE   The CNC is blocked because an invalid date change has been detected.

SOLUTION   Contact your OEM.

8056   **Inconsistency in the recovery of the tool positions in the magazine.**

DETECTION   During CNC startup.

CAUSE   The CNC has not been able to retrieve the information of the tool positions in the magazine.

SOLUTION   Manually define the tools in the storage locations.

8057   **Inconsistency in the recovery of the tool position of the active tool in the magazine.**

DETECTION   During CNC startup.

CAUSE   The CNC has not been able to retrieve the information of the active tool in the magazine.

SOLUTION   Manually define which is the active tool.

8058   **Memory error. It is necessary to restart the CNC.**

DETECTION   During normal operation of the system.

CAUSE   The memory is below the minimum recommended for the correct operation of the CNC.

SOLUTION   Restart the CNC. The CNC may have problems to work properly.

8059   **Remaining disc life below 25%.**

DETECTION   During normal operation of the system.

CAUSE   The disc is nearing the end of its useful life.

SOLUTION   Contact Fagor Automation. The CNC may have problems to work properly.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

8060 **Remaining disc life below 10%. Close the CNC and contact Fagor Automation S. Coop. for technical assistance.**

DETECTION   During normal operation of the system.

CAUSE   The disc is nearing the end of its useful life.

SOLUTION   Contact Fagor Automation. The CNC may have problems to work properly.

8127 **Error while loading RIOR-ETHERCAT software.**

DETECTION   During CNC startup.

CAUSE   Error during the loading process of RIOR-ETHERCAT nodes.

SOLUTION   Check EtherCAT configuration, connections, boot version and application of remote nodes.

8128 **FRAM data could not be retrieved.**

DETECTION   During CNC startup.

CAUSE   Error during FRAM data recovery process. Start-up with RAM data.

SOLUTION   Restart the CNC.

8200 **'Lexer fault when recognizing integers'**

DETECTION   During editing and execution in MDI.

CAUSE   Error when converting a string of characters into a numerical value. 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   Error when converting a string of characters into a numerical value. 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 programmed without having programmed the opening character before.

SOLUTION   Check that the comments have both the opening and closing characters "(" and ")".

8204 **'$ or # missing'**

DETECTION   During editing and execution in MDI.

CAUSE   An instruction has been programmed without the # character or a flow controlling instruction without the "$" character.

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.

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 value 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   Too many decimal digits in a number.
SOLUTION   The maximum number of decimal digits 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.

8215   **'Invalid function format'**

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

8217   **'Floating limits exceeded'**

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

8218   **'Wrong character'**

DETECTION   During editing.
CAUSE   An invalid character has been detected in the block.
SOLUTION   Check the syntax of the block.

8219'   **Empty error string'**

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

8220   **'Empty warning string'**

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

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	An M function has been programmed more than once in the same block or there are two incompatible M functions in the same block.
SOLUTION	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.

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 be positive; 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.

8229 **'F or E out of range'**

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

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 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 parameter has been programmed more than once in the same block.

SOLUTION   Program the I parameter only once in the block.

8244 '"J" programmed twice'

DETECTION   During editing.

CAUSE   The J parameter has been programmed more than once in the same block.

SOLUTION   Program the J parameter only once in the block.

8245 '"K" programmed twice'

DETECTION   During editing.

CAUSE   The K parameter has been programmed more than once in the same block.

SOLUTION   Program the K parameter only once in the block.

8247 'Only R1 can be part of an expression'

DETECTION   During editing.

CAUSE   The radius has been programmed wrong.

SOLUTION   The radius can only be programmed with "R" or "R1".

8250 **'Axis variable without writing permission'**

DETECTION	During editing.
CAUSE	The variable has no writing permission.
SOLUTION	The variable is read-only.

8251 **'Global variable without writing permission'**

DETECTION	During editing.
CAUSE	The variable has no writing permission.
SOLUTION	The variable is read-only.

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.

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	There is a wrong character in the name of the main program or of the subroutine.
SOLUTION	The definition of the main program or subroutine can only be accompanied by a comment. The forbidden characters are: • Main program: \ / : * ? " < > | and blank space. • Local subroutine: / ? " < > |) and blank space.

8258 **'Expression expected after #TIME'**

DETECTION	During editing and execution in MDI.
CAUSE	The #TIME instruction has been programmed wrong.
SOLUTION	The #TIME instruction is programmed as #TIME [<time>] (the brackets are optional), where the <time> parameter may be an integer, an arithmetic parameter or a variable.

8265 **'Unknown or incomplete word'**

DETECTION	During editing.
CAUSE	The possible causes are: • The programmed function, instruction or expression is not valid. • M function has been programmed wrong for a particular spindle. • #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	The variable has no writing permission.
SOLUTION	The variable is read-only.

8268 **'Function argument out of range'**

DETECTION	During editing.
CAUSE	The argument programmed in the mathematical function is not valid.
SOLUTION	Program a correct argument in the mathematical function (LN, LOG, SQRT, EXP, DEXP, SIN, COS, TAN, ASIN, ACOS, ATAN).

8269 **'Function overflow'**

DETECTION	During editing.
CAUSE	The argument programmed in the mathematical function is not valid.
SOLUTION	Program a correct argument in the mathematical function (LN, LOG, SQRT, EXP, DEXP, SIN, COS, TAN, ASIN, ACOS, ATAN).

8270   'Partial loss in function'

DETECTION   During editing.

CAUSE   The argument programmed in the mathematical function is not valid.

SOLUTION   Program a correct argument in the mathematical function (LN, LOG, SQRT, EXP, DEXP, SIN, COS, TAN, ASIN, ACOS, ATAN).

8271   'Invalid argument in function'

DETECTION   During editing.

CAUSE   The argument programmed in the mathematical function is not valid.

SOLUTION   Program a correct argument in the mathematical function (LN, LOG, SQRT, EXP, DEXP, SIN, COS, TAN, ASIN, ACOS, ATAN).

8272   'Total loss in function'

DETECTION   During editing.

CAUSE   The argument programmed in the mathematical function is not valid.

SOLUTION   Program a correct argument in the mathematical function (LN, LOG, SQRT, EXP, DEXP, SIN, COS, TAN, ASIN, ACOS, ATAN).

8273   'Function underflow'

DETECTION   During editing.

CAUSE   The argument programmed in the mathematical function is not valid.

SOLUTION   Program a correct argument in the mathematical function (LN, LOG, SQRT, EXP, DEXP, SIN, COS, TAN, ASIN, ACOS, ATAN).

8274   'Unknown exception type'

DETECTION   During editing.

CAUSE   The argument programmed in the mathematical function is not valid.

SOLUTION   Program a correct argument in the mathematical function (LN, LOG, SQRT, EXP, DEXP, SIN, COS, TAN, ASIN, ACOS, ATAN).

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   Function G11, G12 or G13 has been programmed more than one in the same block. Function G10 or G14 has been programmed together with function G11, G12 or G13 in the same block.

SOLUTION   Functions G11, G12 and G13 may be combined in the same block, but each of them can only be programmed once. Functions G10 and 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.

SOLUTION   The positioning speed "Sn.POS" 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 "Sn.POS" only once.

8282   'Cycle parameter repeated'

DETECTION   During editing.

CAUSE   A 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	The variable has no writing permission.
SOLUTION	The variable is read-only.

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.

8287 **'The limits of the whole portion have been exceeded'**

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

8288 **'The limits of the fractional portion have been exceeded'**

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

8290 **'Variable of the tool manager without writing permission'**

DETECTION	During editing.
CAUSE	The variable has no writing permission.
SOLUTION	The variable is read-only.

8291 **'Variable of the machine parameters without writing permission'**

DETECTION	During editing.
CAUSE	The variable has no writing permission.
SOLUTION	The variable is read-only.

8295 **'Empty parameter list'**

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

8296 **'Repeated or incompatible parameter'**

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

8297 **'Parameter repeated'**

DETECTION	During editing.
CAUSE	The possible causes are: • The parameter CONTERROR has been programmed more than once in the #HSC instruction. • In the subroutine calling instruction #PCALL, #MCALL or G function with associated subroutine, some parameter has been written more than once. • 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 brackets.
SOLUTION	Check the syntax of the instruction in the programming manual.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF. 2209

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 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 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 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 is 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 programmed wrong.

SOLUTION   Function G4 may be programmed as G4 <time> or G4K<time>, where <time> is the dwell time.

8309   **'Too many axes have been programmed'**

DETECTION   During editing.

CAUSE   Too many axes 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   A required parameter is missing in the #POLY, #CS, #ACS or #PROBE instruction

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

8311   **'Wrong parameter value'**

DETECTION   During editing.

CAUSE   Invalid value in an argument of the #CS, #ACS, #HSC or #POLY instruction. Invalid value 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   Invalid value for the MODE command of the #CS or #ACS instruction.

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

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

8313   '#CS/#ACS: wrong coordinate system number'

DETECTION   During editing.

CAUSE   Invalid value for the system number in the #CS or #ACS instruction.

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   Wrong probing cycle number.

SOLUTION   Refer to the documentation on valid probing cycles.

8316   'Parameter not allowed'

DETECTION   During editing and execution in MDI.

CAUSE   Some parameter of the cycle or instruction is not valid.

SOLUTION   Refer to the programming manual to check the right parameters for each cycle and instruction.

8317   'Negative D function'

DETECTION   During editing.

CAUSE   Negative tool offset.

SOLUTION   The tool offset must be positive.

8318   'Only one active 1 axis allowed per block'

DETECTION   During editing.

CAUSE   The @1 axis has been programmed more than once in the same block.

SOLUTION   Program the @1 axis only once in the block.

8319   'Only one active 2 axis allowed per block'

DETECTION   During editing.

CAUSE   The @2 axis has been programmed more than once in the same block.

SOLUTION   Program the @2 axis only once in the block.

8320   'Only one active 3 axis allowed per block'

DETECTION   During editing.

CAUSE   The @3 axis has been programmed more than once in the same block.

SOLUTION   Program the @3 axis only once in the block.

8321   'Only one active 4 axis allowed per block'

DETECTION   During editing.

CAUSE   The @4 axis has been programmed more than once in the same block.

SOLUTION   Program the @4 axis only once in the block.

8322   'Only one active 5 axis allowed per block'

DETECTION   During editing.

CAUSE   The @5 axis has been programmed more than once in the same block.

SOLUTION   Program the @5 axis only once in the block.

8323   'Comparison expected'

DETECTION   During editing.

CAUSE   "=" has been programmed instead of "==".

SOLUTION   Program "==" instead of "=".

8324   'Tabulation not expected'

DETECTION   During editing.

CAUSE   There is a wrong tab.

SOLUTION   Eliminate the tab.

8325   'Line feed not expected'

DETECTION	During editing.
CAUSE	There is a wrong line feed.
SOLUTION	Eliminate the line feed.

8326   'Blank space unexpected'

DETECTION	During editing.
CAUSE	There is a wrong blank space.
SOLUTION	Eliminate the blank space.

8327   'Some parenthesis missing'

DETECTION	During editing and execution in MDI.
CAUSE	The number of opening parenthesis is not the same as 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.

8332   '#WARNING/#ERROR["Text <%d>"<,valor>]/[integer]'

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

8333   '#MSG["Text <%d>"<,value>]'

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

8334   '#FREE AX[axis, ...]'

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

8335   '#WAIT FOR[expression]'

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

8336   '#TANGFEED RMIN[radius]'

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

8337   '#CALL AX [axis, <position,> ...] <option>'

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

8338   '#SET AX [axis, ...] <option>'

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

8339   '#TOOL AX[axis +/-]'

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

8340   '#AXIS[axis, ...]'

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

8341   '#MPG[ResolutionValue,...] AxisName'

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

8342   '#CONTJOG[feedrateValue] axisName'

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

8343   '#INCJOG[[incrementValue, feedrateValue],..., [...]] axisName'

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

8344   '#SET OFFSET[lower, upper]axis'

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

8345   '#LINK[master, slave <,error>]...[...]'

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

8346   '#ROUNDPAR[param, …]: syntax error'

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

8347   '#ASPLINE MODE [<initial> <, final>]: syntax error'

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

8348   '#ASPLINE STARTTANG/ENDTANG axes: syntax error'

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

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

8349 **'#SLOPE[type <,jerk> <,accel> <,move>]: syntax error'**

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

8350 **'#TLC ON[n]': syntax error'**

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

8351 **'#KIN ID <[kinematic]>: syntax error'**

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

8352 **'#CS/ACS DEF <[n]> [MODE m, V1, V2, V3, j1, j2, j3, 0/1]: syntax error'**

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

8353 **'#CS/ACS ON/NEW <[n]> <[MODE m, V1, V2, V3, j1, j2, j3, 0/1]>: syntax error'**

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

8354 **'#CS/ACS DEF ACT[n]: syntax error'**

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

8356 **'#CAX <[spindle, name]>: syntax error'**

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

8357 **'#FACE [abscissa, ordinate <,longitudinal, axis, axis>]<[kinematic]>: syntax error'**

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

8358 **'#CYL [abscissa, ordinate, longitudinal radius] <[kinematics]>: syntax error'**

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

8359 **'#HSC ON <[CONTERROR/FAST value, CORNER value]>: syntax error'**

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

8360 **'#DGWZ[Axis1min, Axis1max, Axis2min, Axis2max <, Axis3min, Axis3max>]'**

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

8361 **'#CD ON <[horizon]>'**

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

8362   '#PATH["path"]'

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

8363   '#POLY[axis[a,b,c,d,e] ... SP EP R]: syntax error'

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

8364   '#PROBE num parameter_list'

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

8365   '#SCALE[value]'

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

8366   '#RPT[Ni, Nf <,repetitions>]'

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

8367   #RENAME AX[OldAxisName,NewAxisName][...]

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

8368   #SELECT PROBE[probe_number]

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

8369   #EXEC["path" <,channel>]

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

8370   #WAIT/#MEET [mark, channel, ...]

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

8371   #SIGNAL[mark, ...]

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

8372   #TIME <[>time<]>

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

8373   #FREE AX ALL

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

8374   #FLUSH

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

8375   #COMMENT BEGIN/END

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

8376   #SYNC POS

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

8377   #UNLINK

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

8378   '#SPLINE ON/OFF: syntax error'

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

8379   '#RTCP ON/OFF: syntax error'

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

8380   '#TLC OFF: syntax error'

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

8381   '#TOOL ORI: syntax error'

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

8382   '#CS/ACS OFF <ALL>: syntax error'

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

8383   '#ECS ON/OFF: syntax error'

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

8384   '#MCS <ON/OFF>: syntax error'

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

8385   '#INIT MACROTAB'

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

8386   '#ESBLK/DSBLK'

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

8387   '#CAX/FACE/CYL OFF: syntax error'

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

8388   '#ESTOP/DSTOP'

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

8389   '#EFHOLD/DFHOLD'

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

8390   '#RET'

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

8391   '#CYCLE BEGIN/END'

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

8392   '#VAR/ENDVAR/DELETE'

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

8393   '#PARK/UNPARK axis'

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

8394   '#CALL subroutine_name'

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

8395   '#PCALL subroutine_name <param>'

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

8396   '#MCALL subroutine_name <param>'

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

8397   '#MASTER spindle_name'

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

8398   '#FREE SP/#CALL SP/#SET SP[spindle_name, ...]'

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

8399   '#RENAME SP[previous_spindle_name, new_spindle_name][...][...]'

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

8400   '#FREE SP ALL'

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

8401   '#EXBLK[block <,channel>]'

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

8402   'Wrong tool magazine number'

DETECTION   During editing.
CAUSE   Wrong magazine number defined in the variable.
SOLUTION   The tool magazine number must be a value between 1 and 4. If the magazine number is not indicated, it assumes the first one.

8403   '#MOVE<ABS>[axis_coordinate<,feedrate><,PRESENT/NEXT/NULL/WAITINPOS>]'

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

8404   '#MOVE ADD[axis_coordinate <,feedrate> <,PRESENT/NEXT/NULL/WAITINPOS>]'

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

8405   '#MOVE INF[axis+/- <,feedrate> <,PRESENT/NEXT/NULL/WAITINPOS>]'

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

8406   '#<T>CAM ON[l, master, slave, n, m, o, p <,ONCE/CONT>]'

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

8407   '#<T>CAM OFF[slave]'

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

8408   '#<T>FOLLOW ON[master, slave <, Nm, Dn, VEL/POS>]'

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

8409   '#<T>FOLLOW OFF[slave]'

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

8410 **'#BSPLINE[axis]'**

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

8411 **'#BSPLINEND'**

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

8412 **'#LATHECY/MILLCY ON/OFF'**

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

8413 **'#<T>SYNC[master_spindle, slave_spindle <,Nx, Dx, Ox>] <[...] ...>'**

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

8414 **'#UNSYNC[slave_spindle <, slave_spindle...>]'**

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

8415 **'Numerator/denominator: program both or none'**

DETECTION	During editing.
CAUSE	Either parameter D or N is missing in the #FOLLOW ON and #SYNC instructions.
SOLUTION	Program both D and N parameters or none; neither of them can be programmed alone.

8416 **'Program the numerator before the denominator'**

DETECTION	During editing.
CAUSE	The D parameter has been programmed before the N parameter in the #FOLLOW ON and #SYNC instructions.
SOLUTION	Program the N parameter before the D parameter.

8418 **'#ANGAX OFF/SUSP <[expression1, expression2, expression3, ...]>'**

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

8419 **'#ANGAX ON [expression1, expression2, expression3, ...]'**

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

8420 **'#SERVO ON/OFF [axis/spindle]'**

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

8421 **'#MDOFF'**

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

8422 **'#CD OFF'**

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

8423   '#OSC axis1 axis2 ...'

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

8424   '#CLEAR<[mark, ...]>'

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

8425   '#HSC OFF'

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

8426   '#DEF "macro1" = "text1" "macro2" = "text2" ...'

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

8427   'Parameter FIRST/SECOND not allowed in this MODE'

DETECTION	During editing.
CAUSE	In the #CS and #ACS instructions, the FIRST and SECOND commands can only be programmed in MODE 6.
SOLUTION	Check the syntax of the instructions in the programming manual.

8428   '(ERROR <integer> <,"text">)

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8429   '(MSG "text")'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8430   "(DGWZ expresión1, expresión2, expresión3, expresión4 <,expresión5, expresión6>)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8431   '(ESBLK/DSBLK)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Instructions ESBLK/DSBLK are not alone in the block, or one of them has not been programmed.
SOLUTION	Refer to the 8055 manual for the correct syntax. Both instructions must be programmed alone and together in the block.

8432   '(ESTOP/DSTOP)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Instructions ESTOP/DSTOP are not alone in the block, or one of them has not been programmed.
SOLUTION	Refer to the 8055 manual for the correct syntax. Both instructions must be programmed alone and together in the block.

8433   '(EFHOLD/DFHOLD)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Instructions EFHOLD/DFHOLD are not alone in the block, or one of them has not been programmed.
SOLUTION	Refer to the 8055 manual for the correct syntax. Both instructions must be programmed alone and together in the block.

8434   '(GOTO N(expression))'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has been programmed wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8435   '(RPT N(expression),N(expression))'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has been programmed wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8436   '(IF condition action1 <ELSE action2>)'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has been programmed wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8437   '(SUB integer)'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has been programmed wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8438   '(RET)'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has not been programmed alone in the block or it has not been programmed.

SOLUTION   Refer to the 8055 manual for the correct syntax. The instruction must be programmed alone in the block.

8439   '(CALL expression)'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has been programmed wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8440   '(PCALL/MCALL/PROBE expression <,param>)'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has been programmed wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8441   '(MDOFF)'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has not been programmed alone in the block or it has not been programmed.

SOLUTION   Refer to the 8055 manual for the correct syntax. The instruction must be programmed alone in the block.

8442   '(EXEC P(expression) <,directory>)'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The instruction has been programmed wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8443   'Action not allowed in IF instruction'

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The actions programmed in the IF instruction are not valid; the actions can neither be IF nor SUB.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8444   G function programmed wrong.

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8445   'Wrong number of axes programmed'

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language. The number of axes programmed
                   for this G function is wrong.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8446   'The programmed G2/G3 block does not admit parameters I, J, K'

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language. 'The programmed G2/G3 block
                   does not admit parameters I, J, K to define the center of the interpolation.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8447   'The programmed function requires parameters I, J, K'

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language. Function G9 requires programming
                   the intermediate point (parameters I, J, K)
SOLUTION   Refer to the 8055 manual for the correct syntax.

8448   Parameters I,J,K programmed wrong.

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8449   Cartesian y Polar coordinates cannot be programmed in the same block.

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8450   The programmed G function does not admit helical programming.

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8451   The programmed G function does not admit programming a thread.

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8452   'Thread programmed wrong'

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language. Function G33/G34 has been
                   programmed wrong.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8453   Unknown or incomplete word after a correct block.

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language. An unknown term has been
                   programmed.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8454   'Wrong E parameter value'

DETECTION   During editing.
CAUSE   Error when programming in 8055 CNC language. Wrong E parameter value in
                   function G49.
SOLUTION   Refer to the 8055 manual for the correct syntax.

8455   'Wrong S parameter value'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Wrong S parameter value in G function.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8456   'Wrong L parameter value'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Wrong L parameter value in function G33/G34.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8457   'Wrong order (sequence) of axes'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The order (sequence) of the axes in the block is wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8458   'Function or variable does not exist in the mill model'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The programmed function only exists in the lathe model and the program is a milling program (pim file).
SOLUTION	Change the name of the file or use the proper software.

8459   'Function or variable does not exist in the lathe model'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The programmed function only exists in the mill model and the program is a lathe program (pit file).
SOLUTION	Change the name of the file or use the proper software.

8460   'PROBE cycle does not exist in the lathe model'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The PROBE cycle does not exist.
SOLUTION	Refer to the 8055 manual for available cycles.

8461   '(INIPAR)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has not been programmed alone in the block or it has not been programmed.
SOLUTION	Refer to the 8055 manual for the correct syntax. The instruction must be programmed alone in the block.

8462   '(DIGIT/TRACE expression <,param>)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8463   '(OPEN P(expression) <,directory><,A/D><,comment>)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8464   '(WRITE block)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8465   '(REPOS axis1, axis2, ...)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8466   '(PAGE expression)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8467   '(SYMBOL/ODW expression1, expression2, expression3)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8468   '(IB expression = INPUT"text" <,format>)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8469   '(DW expresión1 = expression2 <expression3 = expression4><...>)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8470   '(SK expresión1 = "text1" <expression2 = "text2"><...>)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8471   '(WKEY)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has not been programmed alone in the block or it has not been programmed.
SOLUTION	Refer to the 8055 manual for the correct syntax. The instruction must be programmed alone in the block.

8472   '(WBUF <"text", expression>)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has been programmed wrong.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8473   '(SYSTEM)'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The instruction has not been programmed alone in the block or it has not been programmed.
SOLUTION	Refer to the 8055 manual for the correct syntax. The instruction must be programmed alone in the block.

8474   'DIGIT cycle does not exist in the lathe model'

DETECTION	While editing or translating a program.
CAUSE	Error when programming in 8055 CNC language. The programmed function only exists in the mill model and the program is a lathe program (pit file).
SOLUTION	Change the name of the file or use the proper software.

8475   **'TRACE cycle does not exist in the lathe model'**

DETECTION   While editing or translating a program.

CAUSE   Error when programming in 8055 CNC language. The programmed function only exists in the mill model and the program is a lathe program (pit file).

SOLUTION   Change the name of the file or use the proper software.

8476   **'Function without translation'**

DETECTION   While translating a program.

CAUSE   The programmed function has no equivalent on the 8055.

SOLUTION   The CNC cannot translate the function. Refer to the 8055 manual for available functions.

8477   **'Lower case letters not allowed in 8055'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The 8055 language does not admit lower case letters.

SOLUTION   Program the reserved words in upper case letters.

8478   **G45 <axis1 angle1 axis2 angle2 ...>**

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8479   **'#TANGCTRL ON [<axis1 angle1, axis2 angle2, ...><, feedrate>]'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong.

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

8480   **'#TANGCTRL OFF/SUSP <[axis1, axis2, ...]'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong.

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

8481   **Empty block.**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. An empty block has been programmed.

SOLUTION   The 8055 language does not allow empty blocks.

8482   **'Data missing in the block'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The programmed block is not valid (empty block N10 type block, etc.)

SOLUTION   Refer to the 8055 manual for the correct syntax.

8483   **'Value out of range'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The value programmed for the block number or the number of repetitions is not valid.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8484   **'Wrong data order (sequence)'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The order (sequence) of the data in the block is wrong.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8485   **'#DGSPDL spindle_name'**

DETECTION   During execution.

CAUSE   The syntax of the instruction is wrong.

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

8486   'Wrong programming in milling'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Function G15 has been programmed wrong in milling.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8487   'Wrong programming in lathe'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Function G15 has been programmed wrong in lathe.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8488   'Number of repetitions not admitted'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. The number of repetitions can only be programmed in a motion block.
SOLUTION	Refer to the 8055 manual for the correct syntax.

8489   '$FOR i = initial, final, increment'

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

8490   '$GOTO Nexpression/$GOTO [expression]'

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

8491   '$IF condition'

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

8492   '$ELSEIF condition'

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

8493   '$SWITCH expression'

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

8494   '$CASE expression'

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

8495 $WHILE condition

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

8496   Wrong data or order.

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. An unknown term has been programmed in the block or the order of the programmed terms is not correct.
SOLUTION	Review the programming of the block.

8497   'Wrong K parameter value'

DETECTION	During editing.
CAUSE	Error when programming in 8055 CNC language. Wrong K parameter value for the programmed G function.
SOLUTION	Refer to the 8055 manual for the correct syntax.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

8498 **'There is one parameter too many'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. There is one parameter too many
                   in function G24/G27.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8499 **'Wrong parameters in function'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The programmed G function
                   contains invalid parameters

SOLUTION   Refer to the 8055 manual for the correct syntax.

8501 **The function or instruction must be alone in the block.**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. A term not related to the instruction
                   or function has been programmed in the block.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8502 **'The circular interpolation in Polar does not admit a radius'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. Circular interpolation in Polar
                   coordinates does not admit programming the radius; it only admits the Q parameter.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8503 **'The final coordinate of the movement has not yet been programmed'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. The target coordinate of the G8,
                   G9 or G33 move is missing.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8504 **'Wrong number of blocks (0-50)'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. 'Wrong number of blocks in
                   G41/G42.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8506 **'Program only parameter E'**

DETECTION   During editing.

CAUSE   Error when programming in 8055 CNC language. If the E parameter is programmed
                   in the G49 function, the function does not admit more parameters.

SOLUTION   Refer to the 8055 manual for the correct syntax.

8507 **'Wild card programmed twice'**

DETECTION   During editing.

CAUSE   The "?n" wild card has been programmed more than once in the same block.

SOLUTION   Program the wild card only once in the block.

8508 **This type of wild card cannot be used in an instruction.**

DETECTION   During editing.

CAUSE   The axis of a variable has been programmed with a wildcard, and the axis is in another
                   channel.

SOLUTION   A wildcard is not allowed as an axis of a variable when the axis is in another channel.

8509 **'Wrong wild card number'**

DETECTION   During editing.

CAUSE   The number programmed in the "?n" wild card is negative or larger than the number
                   of axes of the channel.

SOLUTION   The number programmed in the "?n" wild card must be a valid axis number.

8510 **#ABORT [expression]/[\"file_name\"]/Nexpression/OFF**

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

8511   #WARNINGSTOP["Text <%d>"<,value>]/[Integer].

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

8512   Blank space not allowed in the name of a subroutine.

DETECTION	During editing and execution.
CAUSE	The subroutine name does not support blank spaces.
SOLUTION	Address the subroutine naming issue.

8513   #DINDIST ON[0/1, ChannelNumber<,ChannelNumber,ChannelNumber>].

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

8514   #DINDIST OFF.

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

8515   The first #DINDIST parameter must be 0 or 1.

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

8516   #DINDIST_EXEC_CYCLE[ChannelNumber].

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

8517   #PATHND ON/OFF.

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

8518   #FEEDND ON/OFF.

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

8519   External variable without writing permission.

DETECTION	During editing and execution.
CAUSE	There is a read only variable or the CNC has momentarily locked the variable while performing an operation with it.
SOLUTION	If the variable is read only, it may not be written. If the CNC has locked the variable, wait for it to finish with its operations.

8520   #DELETE nameVariableUser <, nameVariableUser, ...>.

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

8521   Only one axis can be programmed with G174.

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

8522   #ROTATEMZ1-4 Pi/+i/-i.

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

8523   #POCKET2D BEGIN/END.

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

8524   #POCKET3D BEGIN/END.

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

8525   '#CAM SELECT[<cam,>"path\file"]'

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

8526   '#CAM DESELECT[cam]'

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

8527   '#REPOS <INT/INI> axisName/spindleName …'

DETECTION   During editing.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax of the instruction in the programming manual. After #REPOS program a sequence of axes and/or spindles of the channel.

8528   #VOLCOMP ON/OFF

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

8529   #RENAME AX OFF [axisName,...]

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

8530   #RENAME SP OFF [spindleName,...]

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

8531   #SWTOUT ON[TON = value,TOF = value,PON = value,POF = value]/OFF

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

8532   #HSC ON[FAST <, E value, CORNER valueAngle>]

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

8533   #PWMOUT ON<[FREQ/F value, DUTY/D value]>/OFF

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

8534   #VIRTAX OFF

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

8535   #VIRTAX <ON> <[coordinateAxis]>

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

8536   #CSROT<ON><[ROTATE]>/OFF

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

8537   #DEFROT[ERROR/WARNING/NONE, criterion, Q valueAngle]

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

8538   Criterion: LOW(F/S)/DPOS(F/S)/DNEG(F/S)/VPOS(F/S)/VNEG(F/S).

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

8539   #DEFGRAPH["path\file"]

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

8540   #RETDSBLK

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

8541   #COMP3D OFF

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

8542   #COMP3D <ON> <[PARAX/NORMAL]>

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

8543   Incompatible G and T functions.

DETECTION   During editing and execution.
CAUSE   Programmed T and G functions cannot be programmed together in the same block.
SOLUTION   The T function cannot be programmed in the same block as G801 to G813, G820, G821, G8331 to G833, G840 to G842, G850 or G851.

8544   The G820 function admits a maximum of three points.

DETECTION   During editing and execution.
CAUSE   More than three points have been programmed in the G820 function.
SOLUTION   Check the program.

8545   G802/803/832/833: The arc radius is programmed with I1.

DETECTION   During editing and execution.
CAUSE   The arc radius has been incorrectly programmed in function G802, G803, G832 or 833.
SOLUTION   Program the arc radius with "I1".

8546   Only G802/803/832/833 allow programming the arc radius with I1.

DETECTION   During editing and execution.
CAUSE   The programmed G function does not allow the radius to be programmed with "I1".
SOLUTION   Check the program.

8547 Geometrical entity not valid.

DETECTION During editing and execution.

CAUSE An invalid geometric entity "En" has been programmed.

SOLUTION The range of valid values for a geometric entity is 1 to 30.

8548 Invalid G function for the definition of a geometric entity.

DETECTION During editing and execution.

CAUSE An invalid G function has been programmed in the definition of a geometric entity.

SOLUTION Valid G-functions in the definition of a geometric entity are G801 to G803, G808 to G811, G813, G820, G821, G831 to G833, G841, G842 and G851.

8549 Too many elements in the definition of a geometric entity.

DETECTION During editing and execution.

CAUSE Too many elements have been indicated in the definition of the geometric entity.

SOLUTION The maximum number of elements to be used in the definition of a geometric entity is 6.

8550 #SELECT ORI[HEAD1-2/TABLE1-2, HEAD1-2/TABLE1-2]

DETECTION During editing and execution.

CAUSE The syntax of the instruction is wrong.

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

8551 #KINORG

DETECTION During editing and execution.

CAUSE The syntax of the instruction is wrong.

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

8552 Parameter D not allowed for block.

DETECTION During execution.

CAUSE Parameter D cannot be programmed for the block.

SOLUTION Correct the program.

8553 #DGWZ<CYL>[limInfEje1,limSupEje1,radio/diamInterno,radio/diamExterno]<Pi,Ci,...>

DETECTION During editing and execution.

CAUSE The syntax of the instruction is wrong.

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

8554 The internal radius or diameter must be smaller than the external diameter.

DETECTION During editing and execution.

CAUSE For a circular part defined using the instruction #DGWZ, the internal radius or internal is larger than the external diameter.

SOLUTION Program an internal radius or diameter that is larger than the external diameter.

8555 K expression expected after G04.

DETECTION During editing.

CAUSE Function G4 programmed wrong.

SOLUTION The function G4 may be programmed as G4 <time> or G4 K <time>, where <time> is the dwell.

8556 Invalid tangency solution.

DETECTION During editing and execution.

CAUSE An invalid tangency solution has been programmed in function G831, G832, G833.

SOLUTION The permissible range of values for the tangency solution is 1 to 8.

8557 Tangency solution not allowed for the block.

DETECTION During editing and execution.

CAUSE The G instruction or function programmed in the block does not allow programming of a tangency solution.

SOLUTION The tangency solution is only allowed in functions G831 to G833.

8558   'Incremental programming not allowed'

DETECTION	During editing and execution in MDI.
CAUSE	An incremental movement has been programmed (I command) in a function that does not allow it.
SOLUTION	Check the program.

8559   Only G820 allows several dimensions in the block to be programmed.

DETECTION	During editing and execution.
CAUSE	The G instruction or function programmed in the block does not allow the programming of comma separated dimensions.
SOLUTION	Check the program.

8560   #ISO <ON><[NAME=\"fileName\"]>

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

8561   #ISO OFF

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

8562   #HSC ON[SURFACE <, E value, CORNER valueAngle>]

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

8563   #GAPCTRL ON/OFF [<axisName posValue> <,GAP value>]

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

8564   #LEAPBEGIN[<DOWN> <,G90/G91> <,POS posValue> <,POSLIMIT value>]

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

8565   #LEAPEND

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

8566   #MATERIAL[fileName]

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

8567   NR cannot be programmed during the cycle.

DETECTION	During editing and execution.
CAUSE	The cycle does not allow for the programming of the NR command.
SOLUTION	Delete el NR command from the block. To repeat the block, use the instruction #RPT.

8568   Incorrect NR value (0-1).

DETECTION	During editing and execution.
CAUSE	The NR command does not allow values of 0 or 1.
SOLUTION	Program the NR command with a value equal to or greater than 2.

8569   #OPEN["file" <, A/D> <, KEEPLINE> <, FhandleFile>]

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

8570   #CLOSE<[FhandleFile]>

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

8571   #WRITE[<FhandleFile,> "Text <%d>"/wholeNo. <,value>]

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

8572   #CUTTING ON/OFF <[param]>

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

8573   #GAPPOS[<nameEje coordinateAxis> <nameAxis coordinateAxis> <nameAxis coordianteAxis>]

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

8574   #PARALAX MOV/VIS[numberTransforParallelAxes]

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

8575   #PREVIEW BEGIN/#PREVIEW END

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

8576   #PARALAX OFF<[numberTransforParallelAxes]>

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

8577   #KIN ID[kinematicsNumber <,TIP/TIP=value>]

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

8578   #RTCP   ON/OFF<[CLEAR,   HEAD=ST/DYN/OFF,   TABLE=ST/DYN/OFF, COROT=value/ROT/FIX]>

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

8579   #LOAD GAPTABLE[file].

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

8580   #LOAD PWRTABLE[fichero].

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

8581   #FOLLOW STEP[nombreEje incrementoEje].

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

8582   Programming not allowed with active rotation.

DETECTION   During execution.

CAUSE   An impermissible G-instruction or G-function is programmed when a coordinate system rotation is active.

SOLUTION   Deactivate coordinate rotation.

8583   Programming not allowed with active transform.

DETECTION   During execution.

CAUSE   A G-instruction or function is not allowed when any coordinate transformation is active; #CS/#ACS, #RTCP, #ANGAX, #PARALAX, #VIRTAX.

SOLUTION   Disable coordinate transformation.

8584   Programming not allowed with active compensation.

DETECTION   During execution.

CAUSE   An impermissible G-instruction or function has been programmed when radius compensation (G41/G42) is active.

SOLUTION   Deactivate radius compensation.

8585   Programming not allowed without active independent movement.

DETECTION   During execution.

CAUSE   An impermissible G-instruction or G-function has been programmed because the independent movement is not active.

SOLUTION   Activate independent movement.

8586   #FACE[nombreAbscisa, nombreOrdenada <,nombreLong, Eje4, Eje5, Q valorOrigenC>] <[númeroCinemática]>.

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8587   #CYL [nombreAbscisa, nombreOrdenada, nombreLong valorRadio <,Q valorOrigenC>] <[númeroCinemática]>.

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8701   'Either a resource or a number is missing after the symbol'

DETECTION   During editing.

CAUSE   A resource is missing either after DFU, DFD, ERA or after the symbol of DEF/PDEF.

SOLUTION   Program "DEF/PDEF symbol resource" or "DFU/DFD/ERA resource".

8702   'Resource before symbol'

DETECTION   During editing.

CAUSE   DEF or PDEF command. The resource has been programmed before the symbol.

SOLUTION   Program "DEF/PDEF symbol resource".

8703   'Number before symbol'

DETECTION   During editing.

CAUSE   DEF or PDEF command. The number has been programmed before the symbol.

SOLUTION   Program "DEF/PDEF symbol number".

8704   'A resource or number was expected instead of a symbol'

DETECTION   During editing.

CAUSE   DEF or PDEF command. Two symbols have been programmed.

SOLUTION   Program "DEF/PDEF symbol resource" or "DEF/PDEF symbol number".

8705   'Resources cannot be redefined'

DETECTION   During editing.

CAUSE   DEF or PDEF command. Two resources have been programmed.

SOLUTION   Program "DEF/PDEF symbol resource" or "DEF/PDEF symbol number".

8706   'Numbers cannot be redefined'

DETECTION   During editing.
CAUSE   DEF or PDEF command. Two numbers have been programmed.
SOLUTION   Program "DEF/PDEF symbol resource" or "DEF/PDEF symbol number".

8707   'A symbol followed by a resource or a number is missing'

DETECTION   During editing.
CAUSE   DEF or PDEF command. The symbol and the resource or number has not been programmed.
SOLUTION   Program "DEF/PDEF symbol resource" or "DEF/PDEF symbol number".

8708   'The periodic module period is missing'

DETECTION   During editing.
CAUSE   'The period of the PE periodic module has not been programmed.
SOLUTION   Program "PE period".

8709   'The first operand of the OR consultation is missing'

DETECTION   During editing.
CAUSE   The first consultation of the OR command is missing.
SOLUTION   Program "consultation OR consultation"

8710   'The second operand of the OR consultation is missing'

DETECTION   During editing.
CAUSE   The second consultation of the OR command is missing.
SOLUTION   Program "consultation OR consultation"

8711   'The first operand of the AND consultation is missing'

DETECTION   During editing.
CAUSE   The first consultation of the AND command is missing.
SOLUTION   Program "consultation AND consultation"

8712   'The second operand of the AND consultation is missing'

DETECTION   During editing.
CAUSE   The second consultation of the AND command is missing.
SOLUTION   Program "consultation AND consultation"

8713   'The first operand of the XOR consultation is missing'

DETECTION   During editing.
CAUSE   The first consultation of the XOR command is missing.
SOLUTION   Program "consultation XOR consultation"

8714   'The second operand of the XOR consultation is missing'

DETECTION   During editing.
CAUSE   The second consultation of the XOR command is missing.
SOLUTION   Program "consultation XOR consultation"

8715   'The operand of the NOT consultation is missing'

DETECTION   During editing.
CAUSE   The consultation of the NOT command is missing.
SOLUTION   Program "NOT consultation"

8716   'Closing parenthesis (bracket) missing'

DETECTION   During editing.
CAUSE   There is an opening parenthesis, but the closing parenthesis is missing.
SOLUTION   Program "(consultation)"

8717   'Opening parenthesis (bracket) missing'

DETECTION   During editing.
CAUSE   There is a closing parenthesis, but the opening parenthesis is missing.
SOLUTION   Program "(consultation)"

8718 **'Unmatched parenthesis (bracket)'**

DETECTION	During editing.
CAUSE	The number of opening parenthesis is not the same as the number of closing parenthesis.
SOLUTION	Check that each opening parenthesis has its corresponding closing parenthesis.

8719 **'DFU resource missing'**

DETECTION	During editing.
CAUSE	The resource is missing after the DFU command.
SOLUTION	Program "DFU resource".

8720 **'DFD resource missing'**

DETECTION	During editing.
CAUSE	The resource is missing after the DFD command.
SOLUTION	Program "DFD resource".

8721 **'Action instruction expected after "="'**

DETECTION	During editing.
CAUSE	Instruction missing after "=".
SOLUTION	Program "condition = instruction".

8722 **'The second operand of the CPS is missing'**

DETECTION	During editing.
CAUSE	The second resource of the CPS is missing.
SOLUTION	Program "CPS resource operation resource". The resources may be a register, a symbol or a number. The operation may be one of the commands GT, GE, EQ, NE, LE, LT.

8723 **'The operator of the CPS is missing'**

DETECTION	During editing.
CAUSE	The operation of the CPS command is missing.
SOLUTION	Program "CPS resource operation resource". The resources may be a register, a symbol or a number. The operation may be one of the commands GT, GE, EQ, NE, LE, LT.

8724 **'The operator and second operand of the CPS are missing'**

DETECTION	During editing.
CAUSE	The operator and second resource of the CPS command are missing.
SOLUTION	Program "CPS resource operation resource". The resources may be a register, a symbol or a number. The operation may be one of the commands GT, GE, EQ, NE, LE, LT.

8725 **'The first operand of the CPS is missing'**

DETECTION	During editing.
CAUSE	The first resource of the CPS is missing.
SOLUTION	Program "CPS resource operation resource". The resources may be a register, a symbol or a number. The operation may be one of the commands GT, GE, EQ, NE, LE, LT.

8726 **'The first operand, the operator and the second operand of the CPS are missing'**

DETECTION	During editing.
CAUSE	The operator and both resources of the CPS command are missing.
SOLUTION	Program "CPS resource operation resource". The resources may be a register, a symbol or a number. The operation may be one of the commands GT, GE, EQ, NE, LE, LT.

8727 **'Wrong comparison operator'**

DETECTION	During editing.
CAUSE	The operation programmed in the CPS command is not valid.
SOLUTION	Program "CPS resource operation resource". The resources may be a register, a symbol or a number. The operation may be one of the commands GT, GE, EQ, NE, LE, LT.

8728 **'The timer index is missing in TEN'**

DETECTION	During editing.
CAUSE	The timer is missing in the TEN command.
SOLUTION	Program "TEN timer".

8729 **'The timer index is missing in TRS'**

DETECTION	During editing.
CAUSE	The timer is missing in the TRS command.
SOLUTION	Program "TRS timer".

8730 **'Preset missing in TG'**

DETECTION	During editing.
CAUSE	The preset value is missing in the TG command.
SOLUTION	Program "TG timer preset". The preset value may be a register, a symbol or a number.

8731 **'The timer index and the preset are missing in TG'**

DETECTION	During editing.
CAUSE	The timer and the preset value are missing in the TG command.
SOLUTION	Program "TG timer preset". The preset value may be a register, a symbol or a number.

8732 **'The order of operands is inverted in TG'**

DETECTION	During editing.
CAUSE	The TG command has been programmed wrong.
SOLUTION	Program "TG timer preset". The preset value may be a register, a symbol or a number.

8733 **'Action instruction expected after NOT'**

DETECTION	During editing.
CAUSE	Instruction missing after NOT.
SOLUTION	Program "NOT instruction".

8734 **'Counter index missing in CUP'**

DETECTION	During editing.
CAUSE	The counter is missing in the CUP command.
SOLUTION	Program "CUP counter".

8735 **'Counter index missing in CDW'**

DETECTION	During editing.
CAUSE	The counter is missing in the CDW command.
SOLUTION	Program "CDW counter".

8736 **'Counter index missing in CEN'**

DETECTION	During editing.
CAUSE	The counter is missing in the CEN command.
SOLUTION	Program "CEN counter".

8737 **'Preset missing in CPR'**

DETECTION	During editing.
CAUSE	The preset value is missing in the CPR command.
SOLUTION	Program "CPR counter preset". The preset value may be a register, a symbol or a number.

8738 **'The counter index and the preset are missing in CPR'**

DETECTION	During editing.
CAUSE	The counter and the preset value are missing in the CPR command.
SOLUTION	Program "CPR counter preset". The preset value may be a register, a symbol or a number.

8739 **'The order of operands is inverted in CPR'**

DETECTION	During editing.
CAUSE	The CPR command has been programmed wrong.
SOLUTION	Program "CPR counter preset". The preset value may be a register, a symbol or a number.

CNCelite
8058   8060
8065   8070

8740 **'Operand missing in SET'**

DETECTION — During editing.
CAUSE — The resource is missing in the SET command.
SOLUTION — Program "SET resource".

8741 **'Operand missing in RES'**

DETECTION — During editing.
CAUSE — The resource is missing in the RES command.
SOLUTION — Program "RES resource".

8742 **'Operand missing in CPL'**

DETECTION — During editing.
CAUSE — The resource is missing in the CPL command.
SOLUTION — Program "CPL resource".

8743 **Subroutine name missing after CAL.**

DETECTION — While editing the PLC program.
CAUSE — Subroutine missing in the CAL command.
SOLUTION — Program "CAL subroutine".

8744 **Jump label missing after JMP.**

DETECTION — While editing the PLC program.
CAUSE — Label missing in JMP command.
SOLUTION — Program "JMP label".

8745 **Wrong MOV code.**

DETECTION — While editing the PLC program.
CAUSE — Source or destination code in the MOV command is not valid.
SOLUTION — The source and destination codes indicate the information format (binary or BCD). The codes may be 0 (binary) or 1 (BCD).

8746 **Parser failure when searching the number of bits to be transmitted in the MOV.**

DETECTION — During execution of the PLC program.
CAUSE — The number of bits in the MOV command is not programmed.
SOLUTION — The range of bits to transmit is 4, 8, 12, 16, 20, 24, 28 and 32.

8747 **The source and the destination of the MOV overlap.**

DETECTION — While editing the PLC program.
CAUSE — Some bits to move share same source and destination.
SOLUTION — The source and destination data must not overlap. Reduce the number of bits to move or select a different destination.

8748 **Source or destination missing in the MOV.**

DETECTION — While editing the PLC program.
CAUSE — Label missing in JMP command.
SOLUTION — Program "JMP label".

8749 **Source and destination missing in the MOV.**

DETECTION — While editing the PLC program.
CAUSE — Label missing in JMP command.
SOLUTION — Program "JMP label".

8750 **An argument is missing in the ADS operation.**

DETECTION — While editing the PLC program.
CAUSE — An argument is missing in the ADS operation.
SOLUTION — Program "ADS operand operand result".

8751 **Two arguments are missing in the ADS operation.**

DETECTION — While editing the PLC program.
CAUSE — Two arguments are missing in the ADS operation.
SOLUTION — Program "ADS operand operand result".

8752   All the arguments are missing in the ADS operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the ADS operation.
SOLUTION   Program "ADS operand operand result".

8753   An argument is missing in the SBS operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the SBS operation.
SOLUTION   Program "SBS operand operand result".

8754   Two arguments are missing in the SBS operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the SBS operation.
SOLUTION   Program "SBS operand operand result".

8755   All the arguments are missing in the SBS operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the SBS operation.
SOLUTION   Program "SBS operand operand result".

8756   An argument is missing in the MLS operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the MLS operation.
SOLUTION   Program "MLS operand operand result".

8757   Two arguments are missing in the MLS operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the MLS operation.
SOLUTION   Program "MLS operand operand result".

8758   All the arguments are missing in the MLS operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the MLS operation.
SOLUTION   Program "MLS operand operand result".

8759   An argument is missing in the DVS operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the DVS operation.
SOLUTION   Program "DVS operand operand result".

8760   Two arguments are missing in the DVS operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the DVS operation.
SOLUTION   Program "DVS operand operand result".

8761   All the arguments are missing in the DVS operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the DVS operation.
SOLUTION   Program "DVS operand operand result".

8762   An argument is missing in the MDS operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the MDS operation.
SOLUTION   Program "MDS operand operand result".

8763   Two arguments are missing in the MDS operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the MDS operation.
SOLUTION   Program "MDS operand operand result".

8764   All the arguments are missing in the MDS operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the MDS operation.
SOLUTION   Program "MDS operand operand result".

8765   Register missing in NGU.

DETECTION   While editing the PLC program.
CAUSE   Register missing in NGU.
SOLUTION   Program "NGU register".

8766   Register missing in NGS.

DETECTION   While editing the PLC program.
CAUSE   Register missing in NGU.
SOLUTION   Program "NGU register".

8767   An argument is missing in action instruction AND.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the AND operation.
SOLUTION   Program "AND operand operand result".

8768   Two arguments are missing in action instruction AND.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the AND operation.
SOLUTION   Program "AND operand operand result".

8769   All arguments are missing in action instruction AND.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the AND operation.
SOLUTION   Program "AND operand operand result".

8770   An argument is missing in action instruction OR.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the OR operation.
SOLUTION   Program "OR operand operand result".

8771   Two arguments are missing in action instruction OR.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the OR operation.
SOLUTION   Program "OR operand operand result".

8772   All arguments are missing in action instruction OR.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the OR operation.
SOLUTION   Program "OR operand operand result".

8773   An argument is missing in action instruction XOR.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the XOR operation.
SOLUTION   Program "XOR operand operand result".

8774   Two arguments are missing in action instruction XOR.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the XOR operation.
SOLUTION   Program "XOR operand operand result".

8775   All arguments are missing in action instruction XOR.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the XOR operation.
SOLUTION   Program "XOR operand operand result".

8776   An argument is missing in the RR1 operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the RR1 operation.
SOLUTION   Program "RR1 destination rotations origin".

8777   Two arguments are missing in the RR1 operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the RR1 operation.
SOLUTION   Program "RR1 destination rotations origin".

8778   All arguments are missing in the RR1 operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the RR1 operation.
SOLUTION   Program "RR1 destination rotations origin".

8779   An argument is missing in the RR2 operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the RR2 operation.
SOLUTION   Program "RR2 destination rotations origin".

8780   Two arguments are missing in the RR2 operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the RR2 operation.
SOLUTION   Program "RR2 destination rotations origin".

8781   All arguments are missing in the RR2 operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the RR2 operation.
SOLUTION   Program "RR2 destination rotations origin".

8782   An argument is missing in the RL1 operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the RL1 operation.
SOLUTION   Program "RL1 destination rotations origin".

8783   Two arguments are missing in the RL1 operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the RL1 operation.
SOLUTION   Program "RL1 destination rotations origin".

8784   All arguments are missing in the RL1 operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the RL1 operation.
SOLUTION   Program "RL1 destination rotations origin".

8785   An argument is missing in the RL2 operation.

DETECTION   While editing the PLC program.
CAUSE   An argument is missing in the RL2 operation.
SOLUTION   Program "RL2 destination rotations origin".

8786   Two arguments are missing in the RL2 operation.

DETECTION   While editing the PLC program.
CAUSE   Two arguments are missing in the RL2 operation.
SOLUTION   Program "RL2 destination rotations origin".

8787   All arguments are missing in the RL2 operation.

DETECTION   While editing the PLC program.
CAUSE   All the arguments are missing in the RL2 operation.
SOLUTION   Program "RL2 destination rotations origin".

8788   Last index smaller than the first one in ERA.

DETECTION   While editing the PLC program.

CAUSE   In the instruction ERA, the second argument is less than the first one.

SOLUTION   Program "ERA index resource". The index must be greater than the resource.

8789   Last index out of range in ERA.

DETECTION   While editing the PLC program.

CAUSE   The second index is not a valid resource.

SOLUTION   The valid range for each resource is the following.
- Inputs (I1-I1024).
- Outputs (O1-O1024).
- Local inputs (LI1-LI16).
- Local outputs (LO1-LO8).
- Marks (M1-M8192).
- Messages (MSG1-MSG1024).
- Errors (ERR1-ERR1024)
- Clocks (CLK).
- Registers (R1-R1024).
- Timer status (T1-T512).
- Counter status (C1-C256).

8790   Last index missing in ERA.

DETECTION   While editing the PLC program.

CAUSE   Index missing in the instruction ERA.

SOLUTION   Program "ERA index resource".

8791   First and last index missing in ERA.

DETECTION   While editing the PLC program.

CAUSE   Resource missing in the ERA instruction.

SOLUTION   Program "ERA index resource".

8792   Target missing in PAR.

DETECTION   While editing the PLC program.

CAUSE   Target missing in PAR.

SOLUTION   Program "PAR resource resource".

8793   Origin and target missing in PAR.

DETECTION   While editing the PLC program.

CAUSE   Origin and target missing in PAR.

SOLUTION   Program "PAR resource resource".

8794   Empty error string.

DETECTION   While editing or compiling the PLC program.

CAUSE   A missing text error is defined.

SOLUTION   Define the error text.

8795   Syntax error. Unexpected %.

DETECTION   While editing the PLC program.

CAUSE   The programmed instruction does not admit the indicated argument.

SOLUTION   Correct the syntax of the line.

8796   Empty warning string.

DETECTION   While editing or compiling the PLC program.

CAUSE   A missing text warning is defined.

SOLUTION   Define the warning text.

8797   The second argument must be an index in ERA.

DETECTION   While editing the PLC program.

CAUSE   The second argument must be a number.

SOLUTION   The second argument must be a number; there is no need to program the resource type.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

8798   The period must be greater than zero.

DETECTION	While editing or compiling the PLC program.
CAUSE	A period less than or equal to zero has been programmed in the "PE" directive.
SOLUTION	Set the "PE" directive to a value greater than zero.

8799   The symbol is missing.

DETECTION	While editing or compiling the PLC program.
CAUSE	In the "DEF/PDEF" directives, the resource symbol has not been programmed.
SOLUTION	Program the symbol.

8800   Too many characters in symbol.

DETECTION	While editing or compiling the PLC program.
CAUSE	In the "DEF/PDEF/CAL/SUB/EXTERN" directive, a symbol has been programmed whose length exceeds the maximum allowed.
SOLUTION	Review the programming of the block. The maximum length allowed for the symbol programmed in these directives is 46 characters.

8802   Axis missing in instruction.

DETECTION	While editing or compiling the PLC program.
CAUSE	The CAM or FOLLOW OFF command has been programmed without indicating the slave axis.
SOLUTION	Check the syntax in the installation manual.

8808   The operands of the OR expression are the same.

DETECTION	While editing or compiling the PLC program.
CAUSE	Two of the same operands of the "OR" expression.
SOLUTION	Check the syntax in the installation manual.

8809   The operands of the AND expression are the same.

DETECTION	While editing or compiling the PLC program.
CAUSE	Two of the same operands of the "AND" expression.
SOLUTION	Check the syntax in the installation manual.

8810   The operands of the XOR expression are the same.

DETECTION	While editing or compiling the PLC program.
CAUSE	Two of the same operands of the "XOR" expression.
SOLUTION	Check the syntax in the installation manual.

8811   The operands of the CPS expression are the same.

DETECTION	While editing or compiling the PLC program.
CAUSE	Two of the same operands of the "CPS" expression.
SOLUTION	Check the syntax in the installation manual.

8812   The resources of the MOV are the same.

DETECTION	While editing the PLC program.
CAUSE	The source and destination resources of the MOV instruction are the same.
SOLUTION	Program different source and destination resources.

8813   The resources of the MOV are the same.

DETECTION	While editing the PLC program.
CAUSE	The source and destination registers of the MOV instruction are the same.
SOLUTION	Program the different source and destination registers.

8814   A symbol cannot start with a digit.

DETECTION	While editing the PLC program.
CAUSE	The first character of the symbol is a number.
SOLUTION	The symbols must begin with a letter or with the character "/" followed by a letter.

8815   Input out of the permitted range.

DETECTION	While editing the PLC program.
CAUSE	The number of the digital input is not valid.
SOLUTION	The digital inputs must be within the range of I1 to I1024 and the locals from LI1 to LI16.

8816   Output out of the permitted range.

DETECTION   While editing the PLC program.

CAUSE   The number of the digital output is not valid.

SOLUTION   The digital outputs must be within the range O1 to O1024 and the locals from O1 to O8.

8817   Lexer error when recognizing inputs.

DETECTION   While editing or compiling the PLC program.

CAUSE   Error when converting a string of characters to inputs. Any programmed character is not correct.

SOLUTION   Check the syntax in the installation manual.

8818   Lexer error when recognizing outputs.

DETECTION   While editing or compiling the PLC program.

CAUSE   Error when converting a string of characters to outputs. Any programmed character is not correct.

SOLUTION   Check the syntax in the installation manual.

8819   Register bit out of range.

DETECTION   While editing the PLC program.

CAUSE   The programmed bit is not valid.

SOLUTION   The bits must be within the range B0 to B31.

8820   Message out of range.

DETECTION   While editing the PLC program.

CAUSE   The programmed message is not valid.

SOLUTION   The messages must be within the range MSG1 to MSG1024.

8821   Lexer error when recognizing messages.

DETECTION   While editing or compiling the PLC program.

CAUSE   Error when converting a string of characters to messages. Any programmed character is not correct.

SOLUTION   Check the syntax in the installation manual.

8822   Error out of range.

DETECTION   While editing the PLC program.

CAUSE   The programmed error is not valid.

SOLUTION   The errors must be within the range ERR1 to ERR1024.

8823   Lexer error when recognizing errors.

DETECTION   While editing or compiling the PLC program.

CAUSE   Error when converting a string of characters to errors. Any programmed character is not correct.

SOLUTION   Check the syntax in the installation manual.

8824   Screen out of range.

DETECTION   While editing the PLC program.

CAUSE   The programmed screen is not valid.

SOLUTION   Check the syntax in the installation manual.

8825   Lexer error when recognizing screens.

DETECTION   While editing or compiling the PLC program.

CAUSE   Error when converting a string of characters to screen. Any programmed character is not correct.

SOLUTION   Check the syntax in the installation manual.

8826   Lexer error when recognizing marks.

DETECTION   While editing or compiling the PLC program.

CAUSE   Error when converting a string of characters to marks. Any programmed character is not correct.

SOLUTION   Check the syntax in the installation manual.

8827   Lexer error when recognizing registers.

DETECTION   While editing or compiling the PLC program.
CAUSE   Error when converting a string of characters to registers. Any programmed character is not correct.
SOLUTION   Check the syntax in the installation manual.

8828   Lexer error when recognizing timers.

DETECTION   While editing or compiling the PLC program.
CAUSE   Error when converting a string of characters to timers. Any programmed character is not correct.
SOLUTION   Check the syntax in the installation manual.

8829   Lexer error when recognizing counters.

DETECTION   While editing or compiling the PLC program.
CAUSE   Error when converting a string of characters to counters. Any programmed character is not correct.
SOLUTION   Check the syntax in the installation manual.

8830   Lexer error when recognizing decimal numbers.

DETECTION   While editing or compiling the PLC program.
CAUSE   Error when converting a string of characters to decimal numbers. Any programmed character is not correct.
SOLUTION   Check the syntax in the installation manual.

8831   Mark out of the permitted range.

DETECTION   While editing the PLC program.
CAUSE   The programmed mark is not valid.
SOLUTION   The marks must not be within the range M1 to M8192.

8832   Register out of the permitted range.

DETECTION   While editing the PLC program.
CAUSE   The programmed record is not valid.
SOLUTION   The records must be within the range R1 to R1024.

8833   Timer out of the permitted range.

DETECTION   While editing the PLC program.
CAUSE   The programmed timer is not valid.
SOLUTION   The timer must be within the range T1 to T512.

8834   Counter out of the permitted range.

DETECTION   While editing the PLC program.
CAUSE   The programmed counter is not valid.
SOLUTION   The counter must be within the range C1 to C256.

8835   Clock out of range.

DETECTION   While editing the PLC program.
CAUSE   The programmed clock is not valid.
SOLUTION   Consult valid clocks in the installation manual.

8836   Lexer error when recognizing clocks.

DETECTION   While editing or compiling the PLC program.
CAUSE   Error when converting a string of characters to clocks. Any programmed character is not correct.
SOLUTION   Check the syntax in the installation manual.

8837   Trigger input out of the permitted range.

DETECTION   While editing the PLC program.
CAUSE   The input index is not valid.
SOLUTION   The trigger inputs must be within the range TG1 to TG4.

FAGOR AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·246·

8838   Lexer error when recognizing trigger inputs.

DETECTION   While editing or compiling the PLC program.

CAUSE   Error when converting a string of characters to trigger inputs. Any programmed character is not correct.

SOLUTION   Check the syntax in the installation manual.

8839   Trigger input index missing. It assumes TG1.

DETECTION   While editing the PLC program.

CAUSE   The trigger input index is not yet programmed.

SOLUTION   Program a trigger input within the range TG1 to TG4.

8840   PIC or PAG resource obsolete; substituted by MSG.

DETECTION   While editing the PLC program.

CAUSE   Both resources are now not valid.

SOLUTION   It is possible that both resources belong to another family of CNCs or their names have been changed. It is not possible to use both registers and the PLC for those replaced by MSG.

8841   Expected the starting instruction of a module or subroutine.

DETECTION   While compiling the PLC program.

CAUSE   Any module or subroutine is programmed and the starting instruction is missing.

SOLUTION   Program the instruction CY1, PRG, PE at the start of the module, or SUB at the start of each subroutine.

8842   Consultation expected instead of %.

DETECTION   While compiling the PLC program.

CAUSE   A consultation has not been programmed.

SOLUTION   Program a consultation.

8843   Action expected instead of %.

DETECTION   While compiling the PLC program.

CAUSE   An action has not been programmed.

SOLUTION   Program an action.

8844   Previous consultation missing.

DETECTION   While compiling the PLC program.

CAUSE   A consultation for executing the instruction has not been programmed.

SOLUTION   Check syntax.

8845   System mark defined twice.

DETECTION   While compiling the PLC program.

CAUSE   There are two signals (marks) with same name.

SOLUTION   Check the name of the marks.

8846   User symbol defined twice.

DETECTION   While compiling the PLC program.

CAUSE   There are two symbols with the same name.

SOLUTION   Check the name of the symbols.

8847   Too many user symbols.

DETECTION   While compiling the PLC program.

CAUSE   More than 100 PDEF symbols are defined.

SOLUTION   Program some PDEF symbols as DEF symbols. The PLC may be used to program 100 PDEF symbols and a limited number of DEF symbols.

8848   Too many errors.

DETECTION   While compiling the PLC program.

CAUSE   The PLC has detected too many errors and only shows the first ones.

SOLUTION   Correct the errors and go back to analyze the program.

8849   END or new instruction expected. A line feed may be missing.

DETECTION   While editing the PLC program.

CAUSE   END program missing at the end of a module or subroutine.

SOLUTION   Program END at the end of each module or subroutine. Program END alone in the block.

8850   Fatal error. It cannot go on.

DETECTION   While compiling the PLC program.

CAUSE   The PLC has detected an error and cannot compile or execute the program.

SOLUTION   Check the syntax of the program.

8851   A mark is wrong in this context.

DETECTION   While compiling the PLC program.

CAUSE   There is a mark programmed and the instruction needs another type of resource.

SOLUTION   Check the syntax in the installation manual.

8852   The symbolized resource is wrong in this context.

DETECTION   While compiling the PLC program.

CAUSE   A symbol is programmed and the instruction needs another type of resource in the symbol.

SOLUTION   Check the syntax in the installation manual.

8853   Undefined symbol.

DETECTION   While compiling the PLC program.

CAUSE   The programmed symbol does not exist.

SOLUTION   Check the program.

8854   A register is wrong in this context.

DETECTION   While compiling the PLC program.

CAUSE   There is a register programmed and the instruction needs another type of resource.

SOLUTION   Check the syntax in the installation manual.

8855   DEF expected instead of %.

DETECTION   While editing the PLC program.

CAUSE   The syntax of the instruction is wrong.

SOLUTION   Check the syntax in the installation manual.

8856   System register defined twice.

DETECTION   While compiling the PLC program.

CAUSE   There are two signals (registers) with same name.

SOLUTION   Check the name of the registers.

8857   Only one PRG module allowed.

DETECTION   While compiling the PLC program.

CAUSE   There are two PRG modules programmed.

SOLUTION   There can only be one PRG module in the project. Program the instructions of both modules into a single one.

8858   Only one CY1 module allowed.

DETECTION   While compiling the PLC program.

CAUSE   There are two CY1 modules programmed.

SOLUTION   There can only be one CY1 module in the project. Program the instructions of both modules into a single one.

8859   Too many periodic modules.

DETECTION   While compiling the PLC program.

CAUSE   There are too many periodic modules in the program.

SOLUTION   Group several periodic modules into a single one.

8860   Periodic module defined twice.

DETECTION   While compiling the PLC program.

CAUSE   There are two periodic modules with the same period.

SOLUTION   A project may have several periodic modules, but with different period. Check the period of the periodic modules. If two modules must have the same period, program them into a single module.

8861   Subroutine defined twice.

DETECTION   While compiling the PLC program.

CAUSE   There are two subroutines with the same name.

SOLUTION   Check the name of the subroutines.

8862   Too many subroutines.

DETECTION   While compiling the PLC program.

CAUSE   There are too many subroutines defined in the program.

SOLUTION   Use jumps to labels instead of subroutines, or group several subroutines into a single one.

8863   Undefined label.

DETECTION   While compiling the PLC program.

CAUSE   A jump to a label is defined but does not exist (=JMP label).

SOLUTION   Check the name of the labels and the jumps to the labels.

8864   Line feed missing after end of module.

DETECTION   While compiling the PLC program.

CAUSE   Any instruction is defined in the same line as an "END".

SOLUTION   Program a line break following the "END" instruction.

8865   Not enough memory for new symbols.

DETECTION   While compiling the PLC program.

CAUSE   The limit has been reached for symbols in the system.

SOLUTION   It is not possible to define more symbols.

8866   System marks cannot be MOVed.

DETECTION   While compiling the PLC program.

CAUSE   An attempt was made to move a resource to a PLC signal (mark).

SOLUTION   The consulting signals could not be edited. Modify the modifiable signals using action instructions.

8867   Obsolete system mark.

DETECTION   While compiling the PLC program.

CAUSE   The PLC signal (mark) is no longer valid.

SOLUTION   It is possible that the mark belongs to another family of CNCs or its name has been changed. It is not possible to use this mark.

8868   Obsolete system register.

DETECTION   While compiling the PLC program.

CAUSE   The PLC signal (register) is no longer valid.

SOLUTION   It is possible that the register belongs to another family of CNCs or its name has been changed. It is not possible to use this register.

8869   System mark pending to be implemented.

DETECTION   While compiling the PLC program.

CAUSE   The PLC signal (mark) is not valid.

SOLUTION   The mark belongs to a feature under development and has not yet been implemented. It is not possible to use this mark. Consult Fagor Automation for the availability of that mark or feature.

8870   Labeled defined, but not called upon.

DETECTION   While compiling the PLC program.

CAUSE   A label is defined in the program but there is no jump instruction to it (=JMP label).

SOLUTION   Check the jump instructions or delete the label if it is not used.

8871   Internal error: Line too long.

DETECTION	While compiling the PLC program.
CAUSE	The program line is too long.
SOLUTION	Divide the line into several instructions.

8872   PRG module missing.

DETECTION	While compiling the PLC program.
CAUSE	The PRG module has not been programmed.
SOLUTION	The PRG module must exist in the program.

8873   Lexer error when recognizing periodic modules.

DETECTION	While editing or compiling the PLC program.
CAUSE	Error when converting a string of characters into a periodic module. Any programmed character is not correct.
SOLUTION	Check the syntax in the installation manual.

8874   Undefined subroutine.

DETECTION	While compiling the PLC program.
CAUSE	A call to a subroutine is defined but does not exist (=CAL subroutine).
SOLUTION	Check the name of the subroutines and the calls to subroutines.

8875   Undefined register.

DETECTION	While editing or compiling the PLC program.
CAUSE	The programmed symbol does not exist.
SOLUTION	Program a valid symbol, where "Symbol" can be a system variable or a symbol defined with DEF/PDEF.

8876   The axis file could not be opened.

DETECTION	While compiling the PLC program.
CAUSE	Error opening the file "MTBODATA.tab".
SOLUTION	Check that the file exists and can be opened with a regular editor. If it is corrupted, please contact Fagor Automation.

8877   There is no physical axis equivalent to the logic axis.

DETECTION	While compiling the PLC program.
CAUSE	The program contains a reference to an axis that does not exist.
SOLUTION	Check the program.

8878   Subroutine defined, but not called upon.

DETECTION	While compiling the PLC program.
CAUSE	A subroutine is defined in the program but there is no call instruction to it (=CAL subroutine).
SOLUTION	Check the calls to subroutines or delete the subroutine if it is not used.

8879   Too many labels.

DETECTION	While compiling the PLC program.
CAUSE	There are too many labels defined in the program.
SOLUTION	Use subroutines instead of jumps to labels.

8880   Syntax error. Expected %.

DETECTION	While editing the PLC program.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the syntax in the installation manual.

8881   % instead of %.

DETECTION	While editing the PLC program.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the syntax in the installation manual.

8882   One parameter too many in instruction.

DETECTION	While editing the PLC program.
CAUSE	The syntax of the instruction is wrong.
SOLUTION	Check the syntax in the installation manual.

8883 CNCRD(variable,register,mark).

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8884 CNCWR(register,variable,mark).

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8885 CNCEX(block,mark <,channel>).

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8886 MOVE ABS/ADD(axis,position,speed,LinkType)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8887 MOVE INF(axis,direction,speed,LinkType)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8888 CAM ON(cam,master,slave,offsetM,offsetE,factorM,factorE,ONCE/CONT)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8889 FOLLOW/CAM OFF(slave)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8890 FOLLOW ON(master,slave <,Ni,Di><,VEL/POS/POSDYN/WAITLIM>)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8891 HOME(axis)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8892 TOUCHPROBE(axis,probe,window,lowerlim,Upperlim <,POS/NEG>)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8893 TOUCHPROBE(axis,probe <,window,lowerlim,Upperlim,POS/NEG>)

DETECTION   While editing the PLC program.
CAUSE   The syntax of the instruction is wrong.
SOLUTION   Check the syntax in the installation manual.

8894 'Previous instruction wrong'

DETECTION   When compiling the PLC program.
CAUSE   The analysis of the PLC program done by the compiler, before converting it to C, has detected a wrong instruction. The error window actually indicates the instruction after the wrong one.
SOLUTION   Correct, eliminate or comment the wrong instruction.

8895   'Wrong instruction'

DETECTION   When compiling the PLC program.

CAUSE   The analysis of the PLC program done by the compiler, before converting it to C, has detected a wrong instruction. The error window actually indicates the wrong instruction.

SOLUTION   Correct, eliminate or comment the wrong instruction.

8896   'Empty module only admitted in CY1'

DETECTION   When compiling the PLC program.

CAUSE   The PLC program contains an empty PRG, PE or SUB module

SOLUTION   There are two choices when there is an empty module; eliminate it or write a single instruction that is ";".

8897   System mark without write permission for the PLC.

DETECTION   While editing the PLC program.

CAUSE   The signal (mark) is not modifiable; it is a consulting signal.

SOLUTION   Use the signal in the consulting instructions.

8898   #CAM SELECT(<cam>,"path\file"")

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8899   #MSGVAR[indexVar]["Text <%d>"/wholeNo. <,value>]

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8900   #LEAP[<,G90/G91> <,POSvalue> <,POSLIMITvalue> <,GAPvalue> <,TIMEvalue> <,OVRvalue>] <AxisName axisLevel>

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8901   #LEAP[<,G90/G91> <,POSvalue> <,POSLIMITvalue> <,GAPvalue> <,TIMEvalue> <,OVRvalue>] <AxisName axisLevel>

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8902   #GAPCTRL ON/OFF[<INC> <axisName AxisPosition> <,GAPvaloue> <,Fvalue> <,F3DPvalue> <,OFFDISTvalue>]

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8903   #TECHTABLE<[N"nameMaterialFile",>"nameTable"<OFF]><Tvalue params>

DETECTION   During editing and execution.

CAUSE   The syntax of the instruction is wrong.

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

8904   CNCRD(idVar, canal, ejeLog, idxArray1, idxArray2, registro, marca <,registroError>)

DETECTION   While editing or compiling the PLC program.

CAUSE   The syntax of the command is not correct.

SOLUTION   Check the syntax in the installation manual.

8905   CNCWR(registro, idVar, canal, ejeLog, idxArray1, idxArray2, marca <,registroError>)

DETECTION   While editing or compiling the PLC program.

CAUSE   The syntax of the command is not correct.

SOLUTION   Check the syntax in the installation manual.

9000-9999

9014   Fatal error. Duplicate module or not allowed.

DETECTION	While compiling the PLC program.
CAUSE	The modules CY1 and PRG are repreated, or there are two periodic modules with the same period.
SOLUTION	The CY1 and PRG modules can be programmed once. A project may have several periodic modules, but with different period. If two modules must have the same period, program them into a single module.

9015   TG out of the permitted range.

DETECTION	While editing the PLC program.
CAUSE	The input index is not valid.
SOLUTION	Program a trigger input within the range TG1 to TG4.

9016   Resource out of the permitted range.

DETECTION	While compiling the PLC program.
CAUSE	Any programmed resource is out of the permitted range.
SOLUTION	The valid range for each resource is the following.

- Inputs (I1-I1024).
- Outputs (O1-O1024).
- Local inputs (LI1-LI16).
- Local outputs (LO1-LO8).
- Marks (M1-M8192).
- Messages (MSG1-MSG1024).
- Errors (ERR1-ERR1024)
- Clocks (CLK).
- Registers (R1-R1024).
- Register bits (B0-B31 R1-R1024).
- Timer status (T1-T512).
- Counter status (C1-C256).

9017   Number of bits out of the range allowed in MOV.

DETECTION	While compiling the PLC program.
CAUSE	The number of bits programmed in the MOV instruction is incorrect.
SOLUTION	The range of bits to transmit is 4, 8, 12, 16, 20, 24, 28 and 32.

9018   Too many DFU's in a single consultation.

DETECTION	While compiling the PLC program.
CAUSE	The executable instructions have too many DFU instructions.
SOLUTION	Analyze the DFU instructions in groups, saving the result in a mark.

9019   Too many DFD's in a single consultation.

DETECTION	While compiling the PLC program.
CAUSE	The executable instructions have too many DFU instructions.
SOLUTION	Analyze the DFD instructions in groups, saving the result in a mark.

9020   Too many actions.

DETECTION	While compiling the PLC program.
CAUSE	The executable instruction has too many actions.
SOLUTION	Duplicate the executable instruction and distribute the actions.

9026   End of module or subroutine 'END' missing.

DETECTION	While compiling the PLC program.
CAUSE	Missing directing instruction END at the end of a CY1, PRG or PE module.
SOLUTION	End all the modules with the END directing instruction.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

9027   Instruction not supported.

DETECTION   While editing or compiling the PLC program.
CAUSE   The programmed instruction does not exist.
SOLUTION   Check syntax.

9028   It must be a resource or a number.

DETECTION   While editing or compiling the PLC program.
CAUSE   The programmed instruction is wrong.
SOLUTION   Program a resource or a number.

9029   DEF incorrectly constructed.

DETECTION   While compiling the PLC program.
CAUSE   DEF incorrectly constructed.
SOLUTION   Check the program.

9030   Resource type unknown.

DETECTION   While compiling the PLC program.
CAUSE   Unknown resource type.
SOLUTION   Contact Fagor Automation.

9031   File end in comment.

DETECTION   While compiling the PLC program.
CAUSE   File end in comment.
SOLUTION   Contact Fagor Automation.

9032   The symbol file cannot be opened.

DETECTION   While compiling the PLC program.
CAUSE   The symbol file cannot be opened.
SOLUTION   Contact Fagor Automation.

9033   Not enough memory to create the list of symbols.

DETECTION   While compiling the PLC program.
CAUSE   The system has no memory available to create new resources.
SOLUTION   Contact Fagor Automation.

9034   The resource file with symbol cannot be opened.

DETECTION   While compiling the PLC program.
CAUSE   The system has no memory available to create new resources.
SOLUTION   Contact Fagor Automation.

9035   There is no physical axis corresponding to the logic axis.

DETECTION   While compiling the PLC program.
CAUSE   The program contains a reference to an axis that does not exist.
SOLUTION   Check the program.

9036   Unknown symbol.

DETECTION   While compiling the PLC program.
CAUSE   The program uses a symbol which is defined.
SOLUTION   Use the DEF or PDEF instruction to define the symbols.

9037   Obsolete system mark or register.

DETECTION   While compiling the PLC program.
CAUSE   The mark or register does not exist.
SOLUTION   Remove the mark or register from the PLC program.

9038   Obsolete system mark or register. Instead, use %.

DETECTION   While compiling the PLC program.
CAUSE   The mark or register does not exist.
SOLUTION   Use the mark or register suggested.

9039   System mark or register pending to be implemented.

DETECTION	While compiling the PLC program.
CAUSE	The mark or register has still not been implemented.
SOLUTION	The mark or register may not be used.

9040   Comment line too long.

DETECTION	While compiling the PLC program.
CAUSE	The comment is too long.
SOLUTION	Write a shorter comment or divide the comment into two.

9041   Obsolete system mark or register replaced by %.

DETECTION	While compiling the PLC program.
CAUSE	The mark or registry is not correct; the PLC has been replaced in the program compiled by the suggested resource.
SOLUTION	Replace the mark or registry with the suggested resource.

9042   '/' replaced by '_'.

DETECTION	While compiling the PLC program.
CAUSE	The character "/" is not correct; the PLC has replaced the character "_" in the compiled program.
SOLUTION	Replace the character "/" with "_" in the PLC program.

9043   '+' replaced by 'POS'.

DETECTION	While compiling the PLC program.
CAUSE	The character "+" is not correct; the PLC has replaced it with "POS" in the compiled program.
SOLUTION	Replace the character "+" with "POS" in the PLC program.

9044   '-' replaced by 'NEG'.

DETECTION	While compiling the PLC program.
CAUSE	The character "-" is not correct; the PLC has replaced it with "NEG" in the compiled program.
SOLUTION	Replace the character "-" with "NEG" in the PLC program.

9045   '-' replaced by '_'.

DETECTION	While compiling the PLC program.
CAUSE	The character "-" is not correct; the PLC has replaced it with the character "_" in the compiled program.
SOLUTION	Replace the character "-" with "_" in the PLC program.

9046   '.' replaced by '_'.

DETECTION	While compiling the PLC program.
CAUSE	The character "." is not correct; the PLC has replaced it with the character "_" in the compiled program.
SOLUTION	Replace the character "." with "_" in the PLC program.

9047   Not enough memory for new Binary resources.

DETECTION	While compiling the PLC program.
CAUSE	The system has no memory available to create new resources.
SOLUTION	Contact Fagor Automation.

9048   Not enough memory for new system Binary resources.

DETECTION	While compiling the PLC program.
CAUSE	The system has no memory available to create new resources.
SOLUTION	Contact Fagor Automation.

9049   Not enough memory for new register resources.

DETECTION	While compiling the PLC program.
CAUSE	The system has no memory available to create new resources.
SOLUTION	Contact Fagor Automation.

9050   Not enough memory for new system register resources.

DETECTION	While compiling the PLC program.
CAUSE	The system has no memory available to create new resources.
SOLUTION	Contact Fagor Automation.

9051   Not enough memory for new timer resources.

DETECTION	While compiling the PLC program.
CAUSE	The system has no memory available to create new resources.
SOLUTION	Contact Fagor Automation.

9052   Not enough memory for new counter resources.

DETECTION	While compiling the PLC program.
CAUSE	The system has no memory available to create new resources.
SOLUTION	Contact Fagor Automation.

9053   Not enough memory for new symbols.

DETECTION	While compiling the PLC program.
CAUSE	The system has no memory available to create new resources.
SOLUTION	Contact Fagor Automation.

9054   The COM port for MODBUSSVRRS cannot be opened.

DETECTION	While compiling the PLC program.
CAUSE	Connection to the serial port for the MODBUS server fails.
SOLUTION	Check the connection and parameterization of the serial port.

9301   'The REFFEED1 and REFFEED2 values of the master and slave axes must be the same'

DETECTION	During the validation of the machine parameters.
CAUSE	There is a gantry or tandem pair whose master and slave axes have different homing feedrates in a set (parameters REFFEED1 and REFFEED2).
SOLUTION	Check the machine parameters. The fast and slow home search speeds (parameters REFFEED1 and REFFEED2) must be the same for both axes.

9302   'Restart Windows to assume the new value'

DETECTION	During the validation of the machine parameters.
CAUSE	Some change in the machine parameters requires restarting Windows to be effective.
SOLUTION	Restart Windows.

9303   'The probe must be associated with a Fagor digital node input.

DETECTION	During the validation of the machine parameters.
CAUSE	Parameter PRBDI1 or PRBDI2 refers to an invalid digital input. Only the digital inputs that belong to Fagor modules or the logic inputs used only to simulate the probe are valid.
SOLUTION	Assign a digital input of a Fagor module or a logic input to parameters PRBDI1 and PRBDI2.

9304   'Gantry axes: Both SERCOS axes must have the same OPMODEP'

DETECTION	During the validation of the machine parameters.
CAUSE	There is a gantry o tandem pair whose master and slave axes have parameter OPMODEP set differently.
SOLUTION	Check the machine parameters. Both axes must have the OPMODEP parameter set the same way.

9305   'It is recommended to decrease the filter order'

DETECTION	During the validation of the machine parameters.
CAUSE	The order number of one of the low passing filters of the axis is too high for the defined frequency and it could cause overshooting.
SOLUTION	Check the machine parameters. Decrease the filter order or increase the frequency.

9306   'FAST filter incompatible with 3rd axis-filter'

DETECTION	During the validation of the machine parameters.
CAUSE	The CNC cannot activate the automatic filter of the HSC FAST because there are already three frequency filters active on the axis.
SOLUTION	Remove a frequency filter of the axis or remove the filter of the HSC FAST mode.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·256·

9307 **'With Motion Control software option, "CHTYPE" must be a PLC type channel'**

DETECTION	During the validation of the machine parameters.
CAUSE	It is an MC model and some channel is not defined as PLC channel.
SOLUTION	Check the machine parameters. At the MC model, all the channels must be PLC channels (parameter CHTYPE).

9308 **'Compensation positions out of the module range are not allowed'**

DETECTION	During the validation of the machine parameters.
CAUSE	The leadscrew or cross compensation table of the axis has positions that are defined beyond the limits of the module.
SOLUTION	Define all the positions within the module limits.

9309 **'If the number of magazine is 0, ground tools must be allowed'**

DETECTION	During the validation of the machine parameters.
CAUSE	The machine does not have any tool magazines and does not admit ground tools.
SOLUTION	In order to change the tool on a machine without tool magazines, the machine must admit ground tools (parameter GROUND).

9310 **'The spindle speed override cannot be changed while threading if feedforward < 90%'**

DETECTION	While validating the machine parameters or while executing a G33 thread.
CAUSE	The spindle override cannot be modified while thread cutting if the feed-forward is lower than 90% to avoid damaging the thread.
SOLUTION	Increase the feed-forward of the spindle.

9311 **'A TANDEM axis cannot be in a MUTIAXIS group'**

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	One of the axes belongs to a tandem pair and to a MULTIAXIS group.
SOLUTION	Correct the tandem and multi-axis table. An axis can only be in one of the two configurations.

9312 **'A GANTRY axis cannot be in a MUTIAXIS group'**

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	One of the axes belongs to a gantry pair and to a MULTIAXIS group.
SOLUTION	Correct the gantry and multi-axis table. An axis can only be in one of the two configurations.

9313 **'Wrong DRIVEID of the axis of the MULTIAXIS group in the system'**

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There is no axis in the system having the same DRIVEID as the one indicated when defining a MULTIAXIS group.
SOLUTION	Assign a DRIVEID insde the MULTIAXIS groups that is associated with the axes of the system.

9314 **'A SERCOS axis of the multi-axis group has a different DRIVEID'**

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	Not all the axes of a MULTIAXIS group are connected to the same drive (parameter DRIVEID).
SOLUTION	All the Sercos axes of a MULTIAXIS group must be associated with the same drive and, therefore, their parameter DRIVEID must be identical.

9315 **'There are more sets in the group than at the drive'**

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There can only be 8 gears and there are more parameter sets in a MULTIAXIS group than 8.
SOLUTION	Decrease the number of axes of the MULTIAXIS group or decrease the number of parameter sets of the axes of the group.

9316 **'The axes of the group do not have the same OPMODEP: all velocity feedback or all position feedback'**

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There is a multi-axis group whose axes do not have the same parameter OPMODEP.
SOLUTION	Check the machine parameters. All the axes of a multi-axis group must have the OPMODEP parameter set the same way.

9317   'Group of axes with different feedback: All SERCOS axes must be velocity Sercos'

DETECTION	During the validation of the machine parameters.
CAUSE	There is a multi-axis group whose axes do not have the same parameter COUNTERID and the axes are not velocity-Sercos.
SOLUTION	Check the machine parameters. All the axes of a multi-axis group must have identical parameter COUNTERID; if the parameter is not identical, the axes must be velocity-sercos.

9318   'Two axes in different multi-axis groups with the same COUNTERID in one of their sets'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There are axes in different multi-axis groups that share the same feedback input.
SOLUTION	Change parameter COUNTERID of the axes involved so they do not share the same feedback input.

9319   'Two axes in different multi-axis groups with the same ANAOUTID in one of their sets'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There are axes in different multi-axis groups that share the same analog output.
SOLUTION	Change parameter ANAOUTID of the axes involved so they do not share the same analog output.

9320   'The master and slave axes must have the same ABSFEEDBACK'

DETECTION	During the validation of the machine parameters.
CAUSE	There is a MULTIAXIS group whose axes do not have the same parameter ABSFEEDBACK.
SOLUTION	Check the machine parameters. All the axes of a MULTIAXIS group must have the ABSFEEDBACK parameter set the same way.

9321   'The axes of the multi-axis group must have the same DRIVETYPE'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There is a MULTIAXIS group whose axes do not have the same parameter DRIVETYPE.
SOLUTION	Check the machine parameters. All the axes of a MULTIAXIS group must have the DRIVETYPE parameter set the same way; all the axes must be analog or all of them Sercos.

9322   'There is no hardware for Mechatrolink axes'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The necessary hardware to work with Mechatrolink axes is missing.
SOLUTION	Contact Fagor Automation.

9323   'Tandem axes not admitted with Mechatrolink axes'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There is a Mechatrolink drive that belongs to a tandem pair.
SOLUTION	A Mechatrolink drive cannot be part of a tandem axis. Remove the Mechatrolink axis from the tandem pair or use Sercos drives.

9324   'MULTIAXIS groups not admitted with Mechatrolink axes'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	There is a Mechatrolink drive that belongs to a MULTIAXIS group.
SOLUTION	A Mechatrolink drive cannot be part of a MULTIAXIS group. Remove the Mechatrolink axis from the MULTIAXIS group or use Sercos drives.

9325   'Mechatrolink axes: The inverters must be spindles'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	In a Mechatrolink configuration, the inverters must always be spindles.
SOLUTION	Check the machine parameters.

9326   'Mechatrolink I: Looptime 2 ms'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The Mlink-I mode of Mechatrolink only admits a loop time of 2 ms.
SOLUTION	For the Mlink-I mode, assign a loop time of 2 ms (parameter LOOPTIME) or use the Mlink II (parametr MLINK).

9327 'Mechatrolink: Too many axes and spindles'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE The Mechatrolink configuration has too many axes and spindles (servos and inverters). The Mlink-I mode admits a maximum of 14 elements and Mlink-II 30. Using a time slot of 90 microseconds instead of 60 also reduces the maximum number of drives that the system can handle.

SOLUTION Reduce the number of drives or use the Mlink-II with a time slot of 60 microseconds.

9328 'MECHATROLINK II: Maximum 2 DRV variables per axis'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE The OEM parameters table contains more than two DRV variables assigned to the same axis.

SOLUTION Assign a maximum of two DRV variables to each axis.

9329 'MECHATROLINK II: DRV variables and parameters not admitted in the same axis'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE There are parameters and variables assigned to the same axis in the drive variables (DRV) table of the OEM parameters table.

SOLUTION Either the parameters or the variables may be defined for the same axis, not both.

9330 'Sercos axes not admitted with Mechatrolink axes'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE Sercos and Mechatrolink digital buses are not compatible with each other.

SOLUTION Check the machine parameters.
- With Sercos bus, set MLINK parameter to "No" for all the axes and DRIVETYPE cannot be set to "Mlink".
- With Mechatrolink bus, set MLINK parameter to other than "No" for all the axes and DRIVETYPE cannot be set to "Sercos".

9331 'No variables or DRV parameters are admitted with Mechatrolink I or Mechatrolink II at 17 bytes'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE Only sub-commands that resolve the variables and parameters in Mechtrolink II at 32 bytes may be sent.

SOLUTION Accessing the variables and parameters of the drive (DRV) requires configuring the bus as Mechatrolink II at 32 bytes (parameters MLINK and DATASIZE).

9332 'Remote COUNTERTYPE and COUNTERID 0 are incompatible with each other'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE On analog axes with remote feedback input (parameter COUNTERTYPE), parameter COUNTERID cannot be zero.

SOLUTION Check the machine parameters. Check the type of feedback assigned to the axis (parameter COUNTERTYPE). On analog axes with remote feedback input, parameter COUNTERID indicates the logic address of the drive and it cannot be zero.

9333 'The Sercos hardware is not compatible with Mechatrolink axes'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE The hardware of the central unit is for Sercos bus and the drives are Mechatrolink.

SOLUTION Use Sercos drives. Check the type of drive set in the machine parameters (parameter DRIVETYPE).

9334 'The Mechatrolink hardware is not compatible with Sercos axes'

DETECTION On CNC power-up or when validating the machine parameters.

CAUSE The hardware of the central unit is for Mechatrolink bus and the drives are Sercos.

SOLUTION Use Mechatrolink drives. Check the type of drive set in the machine parameters (parameter DRIVETYPE).

9335 The axis must have the same type of feedback in all the sets.

DETECTION During the validation of the machine parameters.

CAUSE In multi-axis, all parameter sets must have the same feedback type.

SOLUTION Change the type of collection in the different sets.

9336 'Incompatible machine parameters'

DETECTION | On CNC power-up or when validating the machine parameters.

CAUSE | There are two features incompatible with each other. The related machine parameters appear in the error message (for example, parameters HBLS and MODBUSSVRRS are incompatible because they both use the serial line).

SOLUTION | Cancel one of the two parameters mentioned in the error message.

9337 Position-Sercos does not work with LOOPTIME greater than 8 ms.

DETECTION | On CNC power-up or when validating the machine parameters.

CAUSE | The Sercos position drives have limited cycle time of 8 ms.

SOLUTION | Assign the machine parameter LOOPTIME to a value equal to or lower than 8 ms or use the speed drives (OPMODE = Speed).

9338 Insufficient memory. Reduce the value of the parameter.

DETECTION | During start-up or validation of machine parameters.

CAUSE | It has not been possible to allocate the memory for the number of points indicated by the NPOINTS parameter in Fagor > Diagnosis.

SOLUTION | Reduce the value of machine parameter NPOINTS of Fagor > Diagnosis.

9339 SWTDELAY value too high for current LOOPTIME.

DETECTION | During execution.

CAUSE | The value of SWTDELAY is too large for the defined cycle time.

SOLUTION | Reduce the SWTDELAY parameter value or increase the cycle time.

9340 Sercos counter: sets with same COUNTERID must have the same parameter(s):

DETECTION | On CNC power-up or when validating the machine parameters.

CAUSE | Several sets that having the same feedback input of an RCS-S module (parameter COUNTERID) do not have the same value in one of the following parameters: FEEDBACKTYPE, SSITYPE, SSI, I0TYPE, ABSOFF, EXTMULT, REFPULSE, I0CODDI1, I0CODDI2, NPULSES and FBACKAL'

SOLUTION | On sets with the same COUNTERID, define the indicated parameters with the same value.

9341 'Sercos counter: active set of several axes cannot have the same parameter ANAOUTID'

DETECTION | On CNC power-up or when validating the machine parameters.

CAUSE | The active set in several axes is using the same analog output of an RCS-S module (parameter COUNTERID).

SOLUTION | Two active sets cannot share the same analog output. Activate a different set on some axis or configure a different analog output in the set.

9342 'Kinematics offset out of the range allowed by the maximum value'

DETECTION | When validating the machine parameters or when writing a variable.

CAUSE | The defined kinematics offset is out of the range allowed by the machine parameter.

SOLUTION | Set the kinematics variable V.MPK.OFTDATAkin[nb] to a value between "- MAX OFFSET" and "+ "MAX OFFSET" (range defined by the machine parameter of the corresponding kinematics).

9343 DRV variable already exists.

DETECTION | During validation of machine parameters or definition of drive variables.

CAUSE | The DRV variable already exists in the OEM machine parameters.

SOLUTION | Check the definition of the variable.

9344 DRV axis and node variables cannot have the same MNEMONIC.

DETECTION | During the validation of the machine parameters.

CAUSE | Two different variables cannot have the same mnemonic.

SOLUTION | Define DRV variables with different mnemonic.

9345 MLINK: DRV variables having the same ID and TYPE cannot have a different MNEMONIC.

DETECTION | During the validation of the machine parameters.

CAUSE | In the OEM machine parameters, there are drive variables (DRV) with the same identifier (ID) and the same access type (TYPE), but different mnemonic.

SOLUTION | DRV variables with the same identifier must have the same mnemonic.

9346   MLINK: DRV variables having the same ID and a different MNEMONIC cannot have a different MODE.

DETECTION   During the validation of the machine parameters.

CAUSE   In the OEM machine parameters, there are DRVs with the same ID, different mnemonic and different access mode (MODE).

SOLUTION   DRV variables with the same identifier must have the same type of access (parameter TYPE) and the same access mode (parameter MODE).

9347   'Axis repeated in VOLCOMP'

DETECTION   On CNC power-up or when validating the machine parameters.

CAUSE   The same axis is defined twice in the VMOVAXIS or VCOMPAXIS volumetric compensation parameters.

SOLUTION   Correct the machine parameters.

9348   Adjustment of the rotary axis, (measured value=%d) outside the range allowed by the maximum value (V.K.MAXROTCOMP=%d) on the %s-axis.

DETECTION   During the calibration of kinematics.

CAUSE   The measured value is greater than the maximum permissible value (V.K.MAXROTCOMP).

SOLUTION   Change the value of V.K.MAXROTCOMP in subroutine KINCAL_BEGIN.NC or the offset value will b limited to that value.

9349   Active axis SET feedback not connected.

DETECTION   During system start-up or validation of machine parameters.

CAUSE   The feedback defined in the active set of the axis is not connected.

SOLUTION   Check connections.

9350   Active axis SET motor not connected.

DETECTION   During system start-up or validation of machine parameters.

CAUSE   The motor defined in the active set of the axis is not connected.

SOLUTION   Check connections.

9351   DEFAULTSET must match M41SET..M44SET for spindles.

DETECTION   During system start-up or validation of machine parameters.

CAUSE   Parameter DEFAULTSET has a wrong value.

SOLUTION   The value of DEFAULTSET must match one of the values defined in M41SET to M44SET.

9352   CAXSET must match M41SET..M44SET for spindles.

DETECTION   During system start-up or validation of machine parameters.

CAUSE   Parameter DEFAULTSET has a wrong value.

SOLUTION   The value of DEFAULTSET must match one of the values defined in M41SET to M44SET.

9353   SYNCSET must match M41SET..M44SET for spindles.

DETECTION   During system start-up or validation of machine parameters.

CAUSE   Parameter DEFAULTSET has a wrong value.

SOLUTION   The value of DEFAULTSET must match one of the values defined in M41SET to M44SET.

9355   The master and slave axes must have the same OPMODE.

DETECTION   During the validation of the machine parameters.

CAUSE   In a gantry axis, the master and slave axis do not have the same OPMODE parameter.

SOLUTION   The OPMODE parameter must be equal on both axes.

9356   Ballscrew compensation defined twice.

DETECTION   During the validation of the machine parameters.

CAUSE   Two equal spindle offsets (equal MOVAXIS and COMPAXIS parameters) are assigned to the same axis.

SOLUTION   Assign a single spindle offset to the axis.

9357   COMPAXIS axis for the selected compensation not valid.

DETECTION	During the validation of the machine parameters.
CAUSE	The axis is assigned an offset whose offset axis does not coincide with the axis. The offset has no effect on the movement of the axis.
SOLUTION	Assign COMPTABLEID an appropriate value.

9358   Undefined compensation.

DETECTION	During the validation of the machine parameters.
CAUSE	The COMPTABLEID parameter has a value that no longer exists. Offset has not been eliminated.
SOLUTION	Assign COMPTABLEID a value that exists.

9360   Not possible to validate parameters for an axis undergoing gear change.

DETECTION	During system start-up or validation of machine parameters.
CAUSE	An attempt was made to validate the machine parameter table during a range change.
SOLUTION	Wait for the end of the range change to validate the machine parameters.

9361   To change the number of channels, axes or spindles, the CNC must be restarted.

DETECTION	During system start-up or validation of machine parameters.
CAUSE	An attempt was made to validate the machine parameter table after changing the number of channels, axes or spindles.
SOLUTION	To validate the number of channels, axes or spindles it is necessary to restart the CNC.

9364   Various axes/spindles cannot share the same REFERENCEID at the same time.

DETECTION	During start-up or validation of machine parameters.
CAUSE	Two axes or spindles simultaneously share the same reference.
SOLUTION	Review the DEFAULTSET and REFERENCEID parameterisation of the different sets of the axes/spindles involved.

9369   DEFAULSET incompatible with axes/spindles that share the same REFERENCEID.

DETECTION	During start-up or validation of machine parameters.
CAUSE	Two axes or spindles simultaneously share the same reference.
SOLUTION	Review the DEFAULTSET and REFERENCEID parameterisation of the different sets of the axes/spindles involved.

9370   CURRBFID not allowed for analog axes.

DETECTION	During start-up or validation of machine parameters.
CAUSE	CURRFBID has been parameterised for an analogue axis/spindle.
SOLUTION	For analogue axis/spindle, leave this parameter blank.

9371   Unknown motor.

DETECTION	During start-up or validation of machine parameters.
CAUSE	The CNC does not recognise a motor parameterised as AUTODETECT; the motor is not available in Motors.xml.
SOLUTION	Define the engine by entering its ID in MOT_NAME.

9372   New motor detected.

DETECTION	During start-up or validation of machine parameters.
CAUSE	The motor associated with an axis has been replaced.
SOLUTION	Confirm that the engine is correct.

9373   Malfuntion in the Autophasing command

DETECTION	During execution.
CAUSE	Autophasing has been disabled in the process or returns an inconsistent result.
SOLUTION	Check system enablement and check the selected engine.

9375   ABSFEEDBACK and M19SPDLEREV setting incompatible, possible position loss.

DETECTION	During the validation of the machine parameters.
CAUSE	The values ABSFEEDBACK = YES and M19SPDLEREV = YES may lead to a loss of value if a change of SDPLEREV is applied.
SOLUTION	Parameterise M19SPDLEREV=FALSE and use the parameters LOOPCH, AXISCH, PP55 to change the sign.

9376 **All linear or all rotary axes and max. 2 rotary axes.**

DETECTION	During the validation of the machine parameters.
CAUSE	The MOVAXIS entered in the table are incorrect.
SOLUTION	Check the MOVAXIS parameters.

9379 **Error in the execution of the command to access the Stegmann sensor.**

DETECTION	During execution.
CAUSE	Error during the process.
SOLUTION	Check the parameters of the command, the connection and the status and type of sensor.

9380 **Different corrections for the same position are not allowed.**

DETECTION	During the validation of the machine parameters.
CAUSE	The spindle compensation table or an axis cross compensation table has different corrections defined for the same position.
SOLUTION	Review compensation tables.

9381 **No more than 1 HBH type flyer is allowed.**

DETECTION	During the validation of the machine parameters.
CAUSE	More than 1 HBH flyer has been parameterised.
SOLUTION	The CNC allows only one HBH handwheel.

9382 **Bus configuration, need CNC reset.**

DETECTION	During start-up.
CAUSE	The CNC requires a new reset to assume certain internal values related to the bus connections.
SOLUTION	Restart the CNC.

9500 **Overvoltage at the power Bus.**

DETECTION	During normal operation of the system.
CAUSE	The power bus voltage has exceeded the limit.
SOLUTION	Ballast resistor open or incorrectly dimensioned. Malfunctioning of the RPS.

9501 **Overcurrent.**

DETECTION	During normal operation of the system.
CAUSE	Excessive current flowing through the drive. Drive running improperly.
SOLUTION	Carry out a reset of the error, because the parameter settings may be wrong causing current overshooting. Check engine parameters and connections.

9502 **Short-circuit.**

DETECTION	During normal operation of the system.
CAUSE	Short-circuit in the drive.
SOLUTION	The possible solutions are:

- Carry out reset of the error. If the error persists, it is possible that the connection of the power cables may be wrong or may touch each other causing a short-circuit.
- The parameters may be wrong or the drive may be faulty. Contact Fagor Automation.

9503 **Drive communication error.**

DETECTION	During normal operation of the system.
CAUSE	Drive in error state.
SOLUTION	Eliminate the cause of the error and reset the drive.

9504 **Alarm in the feedback signals.**

DETECTION	During normal operation of the system.
CAUSE	The drive has detected an error in the Vpp signals of the encoder.
SOLUTION	Check the encoder connection.

9505 **Alarm for pulses between I0's.**

DETECTION	During normal operation of the system.
CAUSE	The drive has detected an error in the pulse correlation between two encoder I0 signals.
SOLUTION	Check the parameterisation and connection of the encoder.

9506   Check failed for CRC of MP.

DETECTION	During normal operation of the system.
CAUSE	Internal error of the drive.
SOLUTION	Contact Fagor Automation.

9507   Error during MP search.

DETECTION	During normal operation of the system.
CAUSE	Internal error of the drive.
SOLUTION	Contact Fagor Automation.

9508   Drive watchdog.

DETECTION	On startup or normal operation of the system.
CAUSE	The drive does not respond, possibly due to poor performance, bad contacts, etc.
SOLUTION	Contact Fagor Automation.

9509   Unexpected 24 Vdc fault.

DETECTION	On startup or normal operation of the system.
CAUSE	The drive has detected a drop in the control voltage.
SOLUTION	Check connection and 24 Vdc control power supply.

9510   Incompatible current loops on the same node.

DETECTION	During normal operation of the system.
CAUSE	A dual drive has incompatible control modes configured for each engine.
SOLUTION	For a double drive, parameterise the same type of control on both motors.

9511   Error when updating DSP version.

DETECTION	During system startup.
CAUSE	The CNC cannot update the DSP version in any drive or RPS.
SOLUTION	Check the connections. If the error persists, contact Fagor Automation.

9512   Error when updating FPGA version.

DETECTION	On startup or normal operation of the system.
CAUSE	The CNC cannot update the FPGA version in one of the drives or RPS.
SOLUTION	Check the connections. If the error persists, contact Fagor Automation.

9513   Error in the update of version CPLD.

DETECTION	On startup or normal operation of the system.
CAUSE	The CNC cannot update the CPDL version in any drive or RPS.
SOLUTION	Check the connections. If the error persists, contact Fagor Automation.

9514   Feedback: Broken wire, A signal.

DETECTION	On startup or normal operation of the system.
CAUSE	The feedback input of the drive does not receive signal A. The A and/or /A signal cable may be broken or disconnected.
SOLUTION	Check the level of the signals provided by the feedback device.

9515   Feedback: Broken wire, B signal.

DETECTION	On startup or normal operation of the system.
CAUSE	The feedback input of the drive does not receive signal B. The B and/or /B signal cable may be broken or disconnected.
SOLUTION	Check the level of the signals provided by the feedback device.

9516   Feedback: Broken wire, I0 signal.

DETECTION	On startup or normal operation of the system.
CAUSE	The feedback input of the drive does not receive the I0 signal. The I0 and/or /I0 signal wire may be broken or disconnected.
SOLUTION	Check the level of the signals provided by the feedback device.

9517   Feedback: Broken wire, AL signal.

DETECTION	On startup or normal operation of the system.
CAUSE	The feedback input of the drive does not receive the AL signal. The AL and/or /AL signal wire may be broken or disconnected.
SOLUTION	Check the level of the signals provided by the feedback device.

9518   Feedback: Over-current or short-circuit.

DETECTION   On startup or normal operation of the system.

CAUSE   Either a short-circuit has occurred or the feedback device is over-supplied.

SOLUTION   Check the connection and level of the signals provided by the feedback device.

9519   Feedback: Alarm signal received.

DETECTION   On startup or normal operation of the system.

CAUSE   The encoder has generated a signal alarm due to its poor performance.

SOLUTION   Check the encoder documentation to determine the source of (reason for) the alarm. It could also be a false positive due to poor connection.

9520   Feedback: Counting frequency too high.

DETECTION   On startup or normal operation of the system.

CAUSE   The axis speed exceeds the sampling capability of the hardware for this type of encoder signal.

SOLUTION   Decrease the maximum axis moving speed.

9521   Communication error at feedback. Wrong CRC.

DETECTION   On startup or normal operation of the system.

CAUSE   Rotary or linear encoder configuration has been lost. The CRC of the connected FAGOR absolute linear encoder has not been identified by the drive.

SOLUTION   The possible solutions are:
- Check the cable and connection of the feedback device.
- Check that the parameters of the feedback device have been set correctly.
- If everything is fine and the error still comes up, replace the encoder or the head of the linear encoder. Contact Fagor Automation.

9522   Communication error at feedback. Acknowledge not received.

DETECTION   On startup or normal operation of the system.

CAUSE   Error reading absolute position value; acknowledge not received.

SOLUTION   Check the cable and connection of the feedback device. The error could be due to a problem at the feedback device. If the error persists, contact Fagor Automation.

9523   ABSIND: Synchronism error.

DETECTION   On startup or normal operation of the system.

CAUSE   The drive has lost synchronization with the ABSIND module, which may have caused a wrong position reading. The error is likely due to a malfunction of the ABSIND module or a connection problem between the ABSIND module and the drive.

SOLUTION   Check that the connection and the cables are OK. If the error persists, contact Fagor Automation.

9524   ABSIND: Error when reading information.

DETECTION   On startup or normal operation of the system.

CAUSE   The drive has not been able to read the identification of the ABSIND module during system startup. The error is likely due to a malfunction of the ABSIND module or a connection problem between the ABSIND module and the drive.

SOLUTION   Check that the connection and the cables are OK. If the error persists, contact Fagor Automation.

9525   INDUCTOSYN: INC signal too short.

DETECTION   On startup or normal operation of the system.

CAUSE   Incremental signal level of the inductosyn/resolver feedback device too low.

SOLUTION   The possible solutions are:
- Check the gain and phase values of the ABSIND module with the adjustment software.
- Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

9526   INDUCTOSYN: INC signal too high.

DETECTION   On startup or normal operation of the system.

CAUSE   Incremental signal level of the inductosyn/resolver feedback device too high.

SOLUTION   The possible solutions are:

- Check the gain and phase values of the ABSIND module with the adjustment software.
- Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

9527   INDUCTOSYN: GRAY signal not present.

DETECTION   On startup or normal operation of the system.

CAUSE   GRAY signal level of the inductosyn/resolver feedback device missing.

SOLUTION   The possible solutions are:

- Check the gain and phase values of the ABSIND module with the adjustment software.
- Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

9528   INDUCTOSYN: Linear encoder unstable on Power-ON.

DETECTION   During system startup.

CAUSE   The position read when initializing the ABSIND module is unstable.

SOLUTION   The possible solutions are:

- Check that the axis is not moving while starting up or initializing the ABSIND module.
- Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
- Check the ground connections in the electrical cabinet (electrical noise on the signals).
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

9529   INDUCTOSYN: Wrong EEPROM checksum.

DETECTION   On startup or normal operation of the system.

CAUSE   The checksum reading during start up of the ABSIND module is not correct.

SOLUTION   The possible solutions are:

- Check the gain and phase values of the ABSIND module with the adjustment software.
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module associated with that feedback. If the error persists, contact Fagor Automation.

9530   INDUCTOSYN: Unknown error.

DETECTION   On startup or normal operation of the system.

CAUSE   The ABSIND module has generated an unknown error. The error is likely due to bad performance of the ABSIND module.

SOLUTION   It the error is repetitive and persistent, try replacing the ABSIND module associated with that feedback. If the error persists, contact Fagor Automation.

9531   INDUCTOSYN: Incoherent SSITYPE parameter with connected feedback device.

DETECTION   On startup or normal operation of the system.

CAUSE   The value of the SSITYPE parameter of the CNC does not match the selected value in the S50 dipswitch of the ABSIND module.

SOLUTION   Check the dipswitch S50 of the ABSIND module. The selected value must be coherent with CNC machine parameter SSITYPE and with the feedback type (inductosyn/resolver) connected to the ABSIND module.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·266·

9532   INDUCTOSYN: Voltage level 24 V DC/15 V DC too low.

DETECTION   On startup or normal operation of the system.
CAUSE   Power supply failure of the module.
SOLUTION   Check the power supply to the ABSIND module.

9533   INDUCTOSYN: Voltage level 24 V DC/15 V DC too high.

DETECTION   On startup or normal operation of the system.
CAUSE   Power supply failure of the module.
SOLUTION   Check the power supply to the ABSIND module.

9534   Fagor digital feedback. Error on the absolute track.

DETECTION   On startup or normal operation of the system.
CAUSE   The possible causes are:
              • The feedback cannot calculate the absolute position.
              • Feedback pulses lost.
              • The analog signal < 0.20 Vpp.
SOLUTION   The possible solutions are:
              • Check that the linear or rotary encoder is working at proper speed.
              • Check mounting tolerances.
              • Clean the glass or the steel tape of the linear encoder.
              • If none of those solutions solves the error, replace the encoder or the reader head
                of the linear encoder. Contact Fagor Automation.

9535   Fagor digital feedback. CPU error.

DETECTION   On startup or normal operation of the system.
CAUSE   Damaged rotary encoder or reader head of linear encoder.
SOLUTION   Replace the rotary encoder or the reader head of the linear encoder. Contact Fagor
              Automation.

9536   Fagor digital feedback. Error at the adjustment potentiometers.

DETECTION   On startup or normal operation of the system.
CAUSE   Damaged rotary encoder or reader head of linear encoder.
SOLUTION   Replace the rotary encoder or the reader head of the linear encoder. Contact Fagor
              Automation.

9537   Fagor digital feedback. Image capturing sensor (CCD) error.

DETECTION   On startup or normal operation of the system.
CAUSE   Damaged rotary encoder or reader head of the linear encoder; the CCD
              (Charge-Coupled Device) or the LED (Light-Emitting Diode).
SOLUTION   The possible solutions are:
              • Check mounting tolerances.
              • Clean the glass or the steel tape of the linear encoder.
              • If none of those solutions solves the error, replace the encoder or the reader head
                of the linear encoder. Contact Fagor Automation.

9538   Fagor digital feedback. Supply voltage out of range.

DETECTION   On startup or normal operation of the system.
CAUSE   The supply voltage must be between 5.3 V and 4.1 V.
SOLUTION   The possible solutions are:
              • Check that the power cable of the rotary or linear encoder is not damaged.
              • Check that the length of the cable is the right one and that the wire connection
                is OK.

9539   Fagor digital feedback. Parameter error at feedback.

DETECTION   On startup or normal operation of the system.
CAUSE   Rotary or linear encoder configuration has been lost.
SOLUTION   Replace the rotary encoder or the reader head of the linear encoder. Contact Fagor
              Automation.

9540   Error in power supply for the analog outputs.

DETECTION   On startup or normal operation of the system.
CAUSE   Error in the power supply to the analogue outputs of the CNC.
SOLUTION   Check analogue output connection. If the error persists, contact Fagor Automation.

9541   Power bus drop with torque.

DETECTION	On startup or normal operation of the system.
CAUSE	Possible drop in one of the three-phase lines or failure of a drive.
SOLUTION	Check the state of the lines, drives and the power control.

9542   Fagor digital feedback. Analog signal below 0.4 Vpp.

DETECTION	On startup or normal operation of the system.
CAUSE	Input signal amplitude under 0.4 Vpp (approximately).
SOLUTION	Check the cables and the connection of the feedback inputs. If the error persists, contact Fagor Automation.

9543   Fagor digital feedback: Encoder power outside desired values.

DETECTION	During normal operation of the system.
CAUSE	Internal error in the encoder.
SOLUTION	Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9544   Fagor digital feedback: Excessive axis speed.

DETECTION	During normal operation of the system.
CAUSE	Internal error in the encoder.
SOLUTION	Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9545   Fagor digital feedback: Internal temperature is out of range.

DETECTION	During normal operation of the system.
CAUSE	Encoder overtemperature.
SOLUTION	Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9546   Fagor digital feedback: Loss of SOL synchronization.

DETECTION	During normal operation of the system.
CAUSE	Encoder communication error.
SOLUTION	Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9547   I0 pulse not detected.

DETECTION	During normal operation of the system.
CAUSE	The drive has not received the I0 signal.
SOLUTION	Check the parameterisation and connection of the encoder.

9548   Phase missing in the sense card.

DETECTION	During normal operation of the system.
CAUSE	A network phase is not correctly reaching the sense card.
SOLUTION	Check network voltages and connect to the sense card.

9549   Voltage level below 80%.

DETECTION	During normal operation of the system.
CAUSE	Drop in mains voltage. For a short period, the mains voltage did not reach 80% of the expected value.
SOLUTION	Check network voltages and connect to the sense card.

9550   Voltage level below 70%.

DETECTION	During normal operation of the system.
CAUSE	Drop in mains voltage. For a short period, the mains voltage did not reach 70 % of the expected value.
SOLUTION	Check network voltages and connect to the sense card.

9551   Voltage level below 40%.

DETECTION	During normal operation of the system.
CAUSE	Drop in mains voltage. For a short period, the mains voltage did not reach 40 % of the expected value.
SOLUTION	Check network voltages and connect to the sense card.

9552   Voltage level near 0%.

DETECTION   During normal operation of the system.

CAUSE   Drop in mains voltage. For a short period, the mains voltage was close to 0 V.

SOLUTION   Check network voltages and connect to the sense card.

9553   Speed limit reached.

DETECTION   During execution.

CAUSE   A drive included on the list of dual-use products (identified with the text -MDU) has tried to exceed the 550 Hz to achieve the programmed speed.

SOLUTION   The drives included on the list of dual-use products (identified with the text -MDU) do not exceed 550 Hz. It is not possible to reach that speed.

9555   Unexpected drive shutdown.

DETECTION   During system startup.

CAUSE   Unexpected drive shutdown.

SOLUTION   Check the power source of the drives.

9557   Ballast resistor short circuit.

DETECTION   During normal operation of the system.

CAUSE   Current limit in the Ballast resistor.

SOLUTION   Check the connection and integrity of the Ballast resistor.

9558   Ballast resistor overload.

DETECTION   During normal operation of the system.

CAUSE   The duty cycle is too demanding for Ballast's endurance.

SOLUTION   Review Ballast resistance selection.

9610   Short-circuit.

DETECTION   On startup or normal operation of the system.

CAUSE   A short-circuit has been detected in the drive.

SOLUTION   The possible solutions are:
- Carry out reset of the error. If the error persists, it is possible that the connection of the power cables may be wrong or may touch each other causing a short-circuit.
- The parameters may be wrong or the drive may be faulty. Contact Fagor Automation.

9611   Drive communication error.

DETECTION   During normal operation of the system.

CAUSE   Drive in error state.

SOLUTION   Eliminate the cause of the error and reset the drive.

9612   Alarm in the feedback signals.

DETECTION   During normal operation of the system.

CAUSE   The drive has detected an error in the Vpp signals of the encoder.

SOLUTION   Check the encoder connection.

9613   Alarm for pulses between I0's.

DETECTION   During normal operation of the system.

CAUSE   The drive has detected an error in the pulse correlation between two encoder I0 signals.

SOLUTION   Check the parameterisation and connection of the encoder.

9614   Check failed for CRC of MP.

DETECTION   During normal operation of the system.

CAUSE   Internal error of the drive.

SOLUTION   Contact Fagor Automation.

9615   Error during MP search.

DETECTION   During normal operation of the system.

CAUSE   Internal error of the drive.

SOLUTION   Contact Fagor Automation.

9616   Drive watchdog.

DETECTION   On startup or normal operation of the system.
CAUSE   The drive does not respond, possibly due to poor performance, bad contacts, etc.
SOLUTION   Contact Fagor Automation.

9617   Unexpected 24 Vdc fault.

DETECTION   On startup or normal operation of the system.
CAUSE   The drive has detected a drop in the control voltage.
SOLUTION   Check connection and 24 Vdc control power supply.

9618   Incompatible current loops on the same node.

DETECTION   During normal operation of the system.
CAUSE   A dual drive has incompatible control modes configured for each engine.
SOLUTION   For a double drive, parameterise the same type of control on both motors.

9619   Error when updating DSP version.

DETECTION   During system startup.
CAUSE   The CNC cannot update the DSP version in any drive or RPS.
SOLUTION   Check the connections. If the error persists, contact Fagor Automation.

9620   Error when updating FPGA version.

DETECTION   On startup or normal operation of the system.
CAUSE   The CNC cannot update the FPGA version in one of the drives or RPS.
SOLUTION   Check the connections. If the error persists, contact Fagor Automation.

9621   Error in the update of version CPLD.

DETECTION   On startup or normal operation of the system.
CAUSE   The CNC cannot update the CPDL version in any drive or RPS.
SOLUTION   Check the connections. If the error persists, contact Fagor Automation.

9622   Feedback: Broken wire, A signal.

DETECTION   On startup or normal operation of the system.
CAUSE   Loop has detected broken wire on signal A of the drive count. The drive does not receive signal A from the feedback.
SOLUTION   Check the cable and the connection of the collection inlets. If the error persists, contact Fagor Automation.

9623   Feedback: Broken wire, B signal.

DETECTION   On startup or normal operation of the system.
CAUSE   Loop has detected broken wire on signal A of the drive count. The drive does not receive the B signal from the feedback.
SOLUTION   Check the cable and the connection of the collection inlets. If the error persists, contact Fagor Automation.

9624   Feedback: Broken wire, I0 signal.

DETECTION   On startup or normal operation of the system.
CAUSE   The loop has detected broken wire on the I0 signal of the drive count. The drive does not receive signal A from the feedback.
SOLUTION   Check the cable and the connection of the collection inlets. If the error persists, contact Fagor Automation.

9625   Feedback: Broken wire, AL signal.

DETECTION   On startup or normal operation of the system.
CAUSE   Loop has detected broken wire on the AL signal of the drive count. The drive does not receive signal A from the feedback.
SOLUTION   Check the cable and the connection of the collection inlets. If the error persists, contact Fagor Automation.

9626   Feedback: Over-current or short-circuit.

DETECTION   On startup or normal operation of the system.
CAUSE   Either a short-circuit has occurred or the feedback device is over-supplied.
SOLUTION   Check the connection and level of the signals provided by the feedback device.

9627   Feedback: Alarm signal received.

DETECTION   On startup or normal operation of the system.

CAUSE   The encoder has generated a signal alarm due to its poor performance.

SOLUTION   Check the encoder documentation to determine the source of (reason for) the alarm. It could also be a false positive due to poor connection.

9628   Feedback: Counting frequency too high.

DETECTION   On startup or normal operation of the system.

CAUSE   The axis speed exceeds the sampling capability of the hardware for this type of encoder signal.

SOLUTION   Decrease the maximum axis moving speed.

9629   Communication error at feedback. Wrong CRC.

DETECTION   On startup or normal operation of the system.

CAUSE   Rotary or linear encoder configuration has been lost. The CRC of the connected FAGOR absolute linear encoder has not been identified by the drive.

SOLUTION   The possible solutions are:
                 • Check the cable and connection of the feedback device.
                 • Check that the parameters of the feedback device have been set correctly.
                 • If everything is fine and the error still comes up, replace the encoder or the head of the linear encoder. Contact Fagor Automation.

9630   Communication error at feedback. Acknowledge not received.

DETECTION   On startup or normal operation of the system.

CAUSE   Error reading absolute position value; acknowledge not received.

SOLUTION   Check the cable and connection of the feedback device. The error could be due to a problem at the feedback device. If the error persists, contact Fagor Automation.

9631   ABSIND: Synchronism error.

DETECTION   On startup or normal operation of the system.

CAUSE   The drive has lost synchronization with the ABSIND module, which may have caused a wrong position reading. The error is likely due to a malfunction of the ABSIND module or a connection problem between the ABSIND module and the drive.

SOLUTION   Check that the connection and the cables are OK. If the error persists, contact Fagor Automation.

9632   ABSIND: Error when reading information.

DETECTION   On startup or normal operation of the system.

CAUSE   The drive has not been able to read the identification of the ABSIND module during system startup. The error is likely due to a malfunction of the ABSIND module or a connection problem between the ABSIND module and the drive.

SOLUTION   Check that the connection and the cables are OK. If the error persists, contact Fagor Automation.

9633   INDUCTOSYN: INC signal too low.

DETECTION   On startup or normal operation of the system.

CAUSE   Incremental signal level of the inductosyn/resolver feedback device too low.

SOLUTION   The possible solutions are:
                 • Check the gain and phase values of the ABSIND module with the adjustment software.
                 • Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
                 • If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

9634   INDUCTOSYN: INC signal too high.

DETECTION   On startup or normal operation of the system.

CAUSE   Incremental signal level of the inductosyn/resolver feedback device too high.

SOLUTION   The possible solutions are:

- Check the gain and phase values of the ABSIND module with the adjustment software.
- Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

9635   INDUCTOSYN: GRAY signal not found.

DETECTION   On startup or normal operation of the system.

CAUSE   GRAY signal level of the inductosyn/resolver feedback device missing.

SOLUTION   The possible solutions are:

- Check the gain and phase values of the ABSIND module with the adjustment software.
- Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

9636   INDUCTOSYN: Linear encoder unstable on Power-ON.

DETECTION   During system startup.

CAUSE   The position read when initializing the ABSIND module is unstable.

SOLUTION   The possible solutions are:

- Check that the axis is not moving while starting up or initializing the ABSIND module.
- Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device.
- Check the ground connections in the electrical cabinet (electrical noise on the signals).
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

9637   INDUCTOSYN: Wrong EEPROM checksum.

DETECTION   On startup or normal operation of the system.

CAUSE   The checksum reading during start up of the ABSIND module is not correct.

SOLUTION   The possible solutions are:

- Check the gain and phase values of the ABSIND module with the adjustment software.
- If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module associated with that feedback. If the error persists, contact Fagor Automation.

9638   INDUCTOSYN: Unknown error.

DETECTION   On startup or normal operation of the system.

CAUSE   The ABSIND module has generated an unknown error. The error is likely due to bad performance of the ABSIND module.

SOLUTION   It the error is repetitive and persistent, try replacing the ABSIND module associated with that feedback. If the error persists, contact Fagor Automation.

9639   INDUCTOSYN: Incoherent SSITYPE parameter with connected feedback device.

DETECTION   On startup or normal operation of the system.

CAUSE   The value of the SSITYPE parameter of the CNC does not match the selected value in the S50 dipswitch of the ABSIND module.

SOLUTION   Check the dipswitch S50 of the ABSIND module. The selected value must be coherent with CNC machine parameter SSITYPE and with the feedback type (inductosyn/resolver) connected to the ABSIND module.

9640   INDUCTOSYN: Voltage level 24 Vdc/15 Vdc too low.

DETECTION   On startup or normal operation of the system.
CAUSE   Power supply failure of the module.
SOLUTION   Check the power supply to the ABSIND module.

9641   INDUCTOSYN: Voltage level 24 Vdc/15 Vdc too high.

DETECTION   On startup or normal operation of the system.
CAUSE   Power supply failure of the module.
SOLUTION   Check the power supply to the ABSIND module.

9642   Fagor digital feedback. Error on the absolute track.

DETECTION   On startup or normal operation of the system.
CAUSE   The possible causes are:
- The feedback cannot calculate the absolute position.
- Feedback pulses lost.
- The analog signal < 0.20 Vpp.

SOLUTION   The possible solutions are:
- Check that the linear or rotary encoder is working at proper speed.
- Check mounting tolerances.
- Clean the glass or the steel tape of the linear encoder.
- If none of those solutions solves the error, replace the encoder or the reader head of the linear encoder. Contact Fagor Automation.

9643   Fagor digital feedback. CPU error.

DETECTION   On startup or normal operation of the system.
CAUSE   Damaged rotary encoder or reader head of linear encoder.
SOLUTION   Replace the rotary encoder or the reader head of the linear encoder. Contact Fagor Automation.

9644   Fagor digital feedback. Error at the adjustment potentiometers.

DETECTION   On startup or normal operation of the system.
CAUSE   Damaged rotary encoder or reader head of linear encoder.
SOLUTION   Replace the rotary encoder or the reader head of the linear encoder. Contact Fagor Automation.

9645   Fagor digital feedback. Image capturing sensor (CCD) error.

DETECTION   On startup or normal operation of the system.
CAUSE   Damaged rotary encoder or reader head of the linear encoder; the CCD (Charge-Coupled Device) or the LED (Light-Emitting Diode).
SOLUTION   The possible solutions are:
- Check mounting tolerances.
- Clean the glass or the steel tape of the linear encoder.
- If none of those solutions solves the error, replace the encoder or the reader head of the linear encoder. Contact Fagor Automation.

9646   Fagor digital feedback. Supply voltage out of range.

DETECTION   On startup or normal operation of the system.
CAUSE   The supply voltage must be between 5.3 V and 4.1 V.
SOLUTION   The possible solutions are:
- Check that the power cable of the rotary or linear encoder is not damaged.
- Check that the length of the cable is the right one and that the wire connection is OK.

9647   Fagor digital feedback. Parameter error at feedback.

DETECTION   On startup or normal operation of the system.
CAUSE   Rotary or linear encoder configuration has been lost.
SOLUTION   Replace the rotary encoder or the reader head of the linear encoder. Contact Fagor Automation.

9648   Error in power supply for the analog outputs.

DETECTION   On startup or normal operation of the system.
CAUSE   Error in the power supply to the analogue outputs of the CNC.
SOLUTION   Check analogue output connection. If the error persists, contact Fagor Automation.

9649   Power bus drop with torque.

DETECTION   On startup or normal operation of the system.

CAUSE   Possible drop in one of the three-phase lines or failure of a drive.

SOLUTION   Check the state of the lines, drives and the power control.

9650   Fagor digital feedback: Analog signal below 0.4 Vpp.

DETECTION   On startup or normal operation of the system.

CAUSE   Input signal amplitude under 0.4 Vpp (approximately).

SOLUTION   Check the cables and the connection of the feedback inputs. If the error persists, contact Fagor Automation.

9651   Fagor digital feedback: Encoder power outside desired values.

DETECTION   During normal operation of the system.

CAUSE   Internal error in the encoder.

SOLUTION   Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9652   Fagor digital feedback: Excessive axis speed.

DETECTION   During normal operation of the system.

CAUSE   Internal error in the encoder.

SOLUTION   Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9653   Fagor digital feedback: Internal temperature is out of range.

DETECTION   During normal operation of the system.

CAUSE   Encoder overtemperature.

SOLUTION   Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9654   Fagor digital feedback: Loss of SOL synchronization.

DETECTION   During normal operation of the system.

CAUSE   Encoder communication error.

SOLUTION   Check the connection and integrity of the encoder. If the error persists, contact Fagor Automation.

9655   I0 pulse not detected.

DETECTION   During normal operation of the system.

CAUSE   The drive has not received the I0 signal.

SOLUTION   Check the parameterisation and connection of the encoder.

9656   Phase missing in the sense card.

DETECTION   During normal operation of the system.

CAUSE   A network phase is not correctly reaching the sense card.

SOLUTION   Check network voltages and connect to the sense card.

9657   Voltage level below 80%.

DETECTION   During normal operation of the system.

CAUSE   Drop in mains voltage. For a short period, the mains voltage did not reach 80% of the expected value.

SOLUTION   Check network voltages and connect to the sense card.

9658   Voltage level below 70%.

DETECTION   During normal operation of the system.

CAUSE   Drop in mains voltage. For a short period, the mains voltage did not reach 70 % of the expected value.

SOLUTION   Check network voltages and connect to the sense card.

9659   Voltage level below 40%.

DETECTION   During normal operation of the system.

CAUSE   Drop in mains voltage. For a short period, the mains voltage did not reach 40 % of the expected value.

SOLUTION   Check network voltages and connect to the sense card.

9660   Voltage level near 0%.

DETECTION   During normal operation of the system.
CAUSE   Drop in mains voltage. For a short period, the mains voltage was close to 0 V.
SOLUTION   Check network voltages and connect to the sense card.

9700   Axis travel limit overrun.

DETECTION   During normal operation of the system.
CAUSE   Axis travel limit overrun.
SOLUTION   After restoring power to the servo drive system, the axis may be moved to the permitted zone.

9702   Temperature exceeds maximum allowed for drive.

DETECTION   On startup or normal operation of the system.
CAUSE   The temperature of the drive enclosure is too high.
SOLUTION   Reduce the ambient and cabinet temperature.

9703   Temperature has exceeded that permitted in the motor.

DETECTION   On startup or normal operation of the system.
CAUSE   The engine exceeds the maximum permissible temperature. The possible causes are:
                   • The motor temperature measuring cables (position sensor cable) or the temperature sensor are in poor condition.
                   • The application demands high current peaks.
SOLUTION   The possible solutions are:
                   • Stop the system for several minutes and decrease the effort demanded from the drive.
                   • Cool the motor.

9706   Actual speed exceeds range.

DETECTION   During normal operation of the system.
CAUSE   The axis or spindle has exceeded the maximum value defined in parameter G00FEED.
SOLUTION   Check parameter G00FEED.

9709   To much following error.

DETECTION   During normal operation of the system.
CAUSE   The axis has exceeded the allowed tracking error (parameter MAXFLWE). Possible improper axis adjustment, missing enablement, faulty motor or drive, feedback system and/or mechanics.
SOLUTION   Check parameters, settings, connections, condition of the drive, collection system, motor, etc.

9711   Positive limit switch reached.

DETECTION   During normal operation of the system.
CAUSE   The axis has reached the positive limit of travel.
SOLUTION   Review stroke limits.

9712   Negative limit switch reached.

DETECTION   During normal operation of the system.
CAUSE   The axis has reached the negative travel limit.
SOLUTION   Review stroke limits.

9715   Error in calculation of coded I0s.

DETECTION   During normal operation of the system.
CAUSE   The patched I0s are not coherent.
SOLUTION   Check parameter I0TYPE (change increasing to decreasing or vice versa).

9718   Synchronization fault between modules.

DETECTION   During normal operation of the system.
CAUSE   The drives do not synchronise with each other.
SOLUTION   Reduce the frequency of loop execution.

9719   DrcLoop warning.

DETECTION   During system start-up, parameter validation or change of being on an axis or spindle.

CAUSE   Error in the consistency check of cycle time parameters or error in the initialization of the parameters of an axis or spindle.

SOLUTION   Check parameters CNCTIME, SYSTEMTIME, POS_LOOPTIME.

9720   Drive error.

DETECTION   During normal operation of the system.

CAUSE   The drive has generated an undetermined error.

SOLUTION   Reset the errors. If the error persists, contact Fagor Automation.

9721   Motor overload.

DETECTION   On startup or normal operation of the system.

CAUSE I^2t motor protection. The duty cycle is too demanding for the engine.

SOLUTION   Adjust the duty cycle, reduce the torque applied to the motor or use a motor with higher torque.

9722   Motor overload.

DETECTION   On startup or normal operation of the system.

CAUSE I^2t motor protection. The duty cycle is too demanding for the engine.

SOLUTION   Adjust the duty cycle, reduce the torque applied to the motor or use a motor with higher torque.

9723   Torque saturation error.

DETECTION   On startup or normal operation of the system.

CAUSE   The possible causes are:
- The motor has locked up and it could not turn freely.
- With high motor torque, the turning speed has not exceeded GP8 for a time period longer than GP7.

SOLUTION   The possible solutions are:
- Free the motor and if the error comes up again for no reason, increase the values of GP7 and/or GP8. To cancel this error and prevent it from coming up again, set GP7=0.
- Verify that two phases have not been swapped and that the power cables make good contact.
- Check that the feedback cable is not defective and the pin-to-pin connection is correct (especially if the user has made the cable).
- Check that the value set for Np116 (with non-Fagor motors) is correct.
- The value of RV3 (or RP5 when using a resolver) is correct.

9724   Following error in the emergency ramp while braking.

DETECTION   On startup or normal operation of the system.

CAUSE   The spindle or spindle stop cannot follow the emergency ramp (parameters EMERGVELMAX and EMERGTIMEMAX).

SOLUTION   Adjust the braking demands of your machine and/or set the two above-mentioned parameters accordingly.

9725   Excessive difference for actual position.

DETECTION   During normal operation of the system.

CAUSE   The position difference between internal and external feedback is greater than the FBACKDIFF parameter.

SOLUTION   Review the parameterisation of the axis catchments.

9726   Drive overload.

DETECTION   On startup or normal operation of the system.

CAUSE I^2t protection of the drive. The duty cycle is too demanding for the drive.

SOLUTION   Adjust the duty cycle, reduce the torque applied to the motor or use a higher powerdrive.

9727   Drive overload.

DETECTION   On startup or normal operation of the system.

CAUSE I^2t protection of the drive. The duty cycle is too demanding for the drive.

SOLUTION   Adjust the duty cycle, reduce the torque applied to the motor or use a higher powerdrive.

9728   Overspeed.

DETECTION   During normal operation of the system.

CAUSE   Speed limit reached.

SOLUTION   Review parameterisation and/or adjustment of the system.

9729   OverRun.

DETECTION   During normal operation of the system.

CAUSE   The sign of the torque setpoint and speed feedback is wrong.

SOLUTION   Check parameters INVERT and LOOPCH.

9730   RPS error.

DETECTION   During normal operation of the system.

CAUSE   The RPS has generated an undetermined error.

SOLUTION   Reset the errors. If the error persists, contact Fagor Automation.

9731   Feedback alarm at handwheel.

DETECTION   On startup or normal operation of the system.

CAUSE   Feedback alarm in the drive feedback input handwheel.

SOLUTION   Check the connection and power supply of the handwheel.

9733   Double AT Miss communication fault.

DETECTION   On startup or normal operation of the system.

CAUSE   Initializing error of the Sercos III bus. The CNC has not received cyclical data from a drive.

SOLUTION   Check the Sercos III connection. If the error persists, contact Fagor Automation.

9734   AT communication fault.

DETECTION   On startup or normal operation of the system.

CAUSE   Initializing error of the Sercos III bus. The CNC has not received cyclical data from a drive.

SOLUTION   Check the Sercos III connection. If the error persists, contact Fagor Automation.

9735   MDT Miss communication fault.

DETECTION   On startup or normal operation of the system.

CAUSE   Initializing error of the Sercos III bus. One drive has not responded to the cyclical data sent by the NCC.

SOLUTION   Check the Sercos III connection. If the error persists, contact Fagor Automation.

9736   MDT communication fault.

DETECTION   On startup or normal operation of the system.

CAUSE   Initializing error of the Sercos III bus. One drive has not responded to the cyclical data sent by the NCC.

SOLUTION   Check the Sercos III connection. If the error persists, contact Fagor Automation.

9737   Synchronization fault.

DETECTION   On startup or normal operation of the system.

CAUSE   The CNC sends a synchronism message in every cycle (usually every 4 ms) that synchronizes the drives. This error appears when the drives can't synchronize or lose synchronism.

SOLUTION   Maybe the CNC has not sent this message or if it has, it wasn't in the precise moment. Check the transmission cable or check that there is no noise in the transmission.

9738   DataLink not found.

DETECTION   On startup or normal operation of the system.

CAUSE   Error in the Sercos III bus. The CNC cannot be connected to the bus.

SOLUTION   Check the Sercos III connection. Connect the Sercos III cable.

9739   Lost package.

DETECTION   On startup or normal operation of the system.

CAUSE   Error in the Sercos III bus. The cable is disconnected or a node is not operating correctly.

SOLUTION   Check the Sercos III connection and the status of the nodes.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·277·

9740   Wrong CRC.

DETECTION   On startup or normal operation of the system.

CAUSE   Error in the Sercos III bus. The cable is disconnected or there is interference in the bus.

SOLUTION   Check the Sercos III connection and the status of the nodes.

9741   Working Counter error.

DETECTION   On startup or normal operation of the system.

CAUSE   Error in the Sercos III bus. Wrong operation of any of the nodes.

SOLUTION   Check the Sercos III connection and the status of the nodes.

9742   RPS drive error.

DETECTION   During normal operation of the system.

CAUSE   The RPS has generated an undetermined error.

SOLUTION   Reset the errors. If the error persists, contact Fagor Automation.

9743   RPS drive overload.

DETECTION   During normal operation of the system.

CAUSE   I²t error in the RPS. The return duty cycle is too demanding for the power of the RPS.

SOLUTION   Gentler system return/braking. Use a more powerful RPS.

9744   RPS drive overload.

DETECTION   During normal operation of the system.

CAUSE   I²t error in the RPS. The return duty cycle is too demanding for the power of the RPS.

SOLUTION   Gentler system return/braking. Use a more powerful RPS.

20000-20999

20001   Waiting for AUXEND from the PLC.

DETECTION   During normal operation of the system.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   The PLC must respond when the appropriate conditions are met or press [RESET].

20002   Waiting for the %x spindle to be stopped.

DETECTION   During normal operation of the system.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The process must end.

20003   Waiting for program to be loaded.

DETECTION   During normal operation of the system.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The process must end.

20004   Waiting for REVOK signal from the %x spindle to begin G33/G34.

DETECTION   During normal operation of the system.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The process must end.

20005   Waiting for gear confirmation by the drive.

DETECTION   During normal operation of the system.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The process must end.

20006   Waiting for end of #PARK maneuver at the drive.

DETECTION   During execution.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The process must end.

20007   Waiting available channel for #EXEC/#EXBLK.

DETECTION   During execution.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The channel must be free when the execution is finished.

20008   Waiting for #WAIT FOR condition.

DETECTION   During execution.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] or wait for the condition to be met.

20009   Waiting for INPOS signal from the %x spindle.

DETECTION   During normal operation of the system.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The process must end.

20010   Waiting for the spindle synchronization process to end.

DETECTION   During execution.

CAUSE   The wait has exceeded a certain time due to good cause or blockage.

SOLUTION   Press [RESET] and analyse the causes of the delay. The process must end.

20011   Waiting for a response from the tool manager in the %c channel.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Wait until the tool is available, or press [RESET] and analyse the causes of the delay. The process must end.

20012   Waiting for the change to JOG mode of the %c channel to end.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The process must end.

20013   Waiting for gear confirmation by the PLC.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The PLC must confirm the new range.

20014   Waiting for the %x spindle home search to end.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The spindle to end the reference search.

20015   G33/G34/G95 with stopped spindle %x.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The spindle must be started to run G33/G34/G95.

20016   Waiting for the axis request to end.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The requested axis must reach the channel or be released by another axis.

20017   Waiting for the spindle request to end.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The requested spindle must reach the channel or be released by another.

20018   Waiting for the forcing of %x axis coordinate (G174) to end.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. Function G174 must be terminated.

20019   Waiting for XFERINH signal from the PLC.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The PLC must activate the _XFERINH signal.

20020   Waiting for axes to get in position.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The axes must be brought into position.

20021   Waiting for feedback initialization.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The initialisation of the catchments must be completed.

20022 #HSC FAST/SURFACE software option not allowed. Running in compatible mode.

DETECTION	During execution.
CAUSE	Programming of #HSC not allowed because software option is missing.
SOLUTION	Acquire the software option or program a permitted option.

20023 DMC: learning to adjust the target power.

DETECTION	During execution.
CAUSE	The DMC notifies that learning has started.
SOLUTION	No action required.

20024 Waiting for new parameters Set of axis/head %x.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The parameters of the new set should arrive.

20025 Waiting to finalize previous gear change for axis/spindle %x.

DETECTION	During normal operation of the system.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The set change must be completed.

20026 Waiting for cycle blocks.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The process must end.

20027 Waiting for spindle %x to enter open loop.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The process must end.

20028 Waiting for spindle %x to enter closed loop.

DETECTION	During execution.
CAUSE	The wait has exceeded a certain time due to good cause or blockage.
SOLUTION	Press [RESET] and analyse the causes of the delay. The process must end.

20029 Rotary axis movement not allowed with RTCP STATIC active.

DETECTION	During execution.
CAUSE	The CNC has attempted to move an axis affected by RTCP STATIC.
SOLUTION	Disable the RTCP STATIC function.

20111 The uploaded .xca file is an older version and not compatible with the current version of the CNC.

DETECTION	When uploading xca. files.
CAUSE	The xca file is v1.0 and should be v2.0.
SOLUTION	Convert the file to v2.0.

20112 here were errors generating the xca file from the pxca file. .xca file not generated.

DETECTION	When uploading xca. files.
CAUSE	Unable to generate xca file from pxca file.
SOLUTION	Check the Defgraph-err.log file for the causes of the error.

23000-23999

23001   Maximum number of compensation tables exceeded.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   There are more compensation tables than allowed.
SOLUTION   Reduce the number of tables.

23002   Error in the volumetric compensation: the memory of the compensation data could not be stored.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Error when allocating memory.
SOLUTION   If the error persists, contact Fagor Automation.

23003   Error in the volumetric compensation file: the line could not be analysed.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The volumetric compensation file is not in the correct format.
SOLUTION   Convert the volumetric compensation file to the correct format.

23004   Error in the volumetric compensation file: the first axis assigned is incorrect.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The first axis assigned to volumetric compensation is not correct.
SOLUTION   Correct the axis.

23005   Error in the volumetric compensation file: the second axis assigned is incorrect.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The second axis assigned to volumetric compensation is not correct.
SOLUTION   Correct the axis.

23006   Error in the volumetric compensation file: the third axis assigned is incorrect.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The third axis assigned to volumetric compensation is not correct.
SOLUTION   Correct the axis.

23007   Error in the volumetric compensation file: element out of expected range.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Variable or element of the volumetric compensation file out of range.
SOLUTION   Modify the volumetric compensation file.

23008   Error in the volumetric compensation file: error when loading data, volumetric points.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The points in the volumetric compensation file are not correct.
SOLUTION   Modify the points in the volumetric compensation file.

23009   Error in the volumetric compensation file: "SQUARENESS" parameters do not have contiguous points.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The points in the volumetric compensation file are not correct.
SOLUTION   Modify the points in the volumetric compensation file.

23010   Error in the volumetric compensation file: the coordinate is outside the lower limit of the compensation volume.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The coordinate is outside the lower limit of the compensation volume.
SOLUTION   Modify the coordinates to be within the compensation volume.

23011 **Error in the volumetric compensation file: the coordinate is outside the upper limit of the compensation volume.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The coordinate is outside the upper limit of the compensation volume.
SOLUTION   Modify the coordinates to be within the compensation volume.

23012 **Error in the volumetric compensation file: the minimum value is greater than the maximum value.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The minimum value defined in the offset file is greater than the maximum value.
SOLUTION   Modify the minimum or maximum value of the volumetric compensation file.

23013 **Error in the volumetric compensation file: the points in the file are in reverse order.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The items in the file are in reverse order.
SOLUTION   Modify the points in the volumetric compensation file.

23014 **Error in the volumetric compensation file: the coordinate is outside the interpolation edges.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The coordinate is outside the interpolation edges.
SOLUTION   Modify the coordinates or file of the volumetric compensation.

23015 **Error when analyzing the volumetric compensation definition file.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The volumetric compensation file does not contain the appropriate data.
SOLUTION   Modify the volumetric compensation file.

23016 **Error in the volumetric compensation file: the machine coordinate system "MACHINE" is not allowed.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The "MACHINE" coordinate system is not allowed.
SOLUTION   Modify the volumetric compensation file to be in the KINEMATIC coordinate system.

23017 **Error in the volumetric compensation file: the coordinate system is incorrect.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The coordinate system defined in the file is incorrect.
SOLUTION   Modify the volumetric compensation file.

23018 **Volumetric compensation file error: the value of the file parameter "SQUARENESS_IS_POSITIVE" is incorrect.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Parameter "SQUARENESS_IS_POSITIVE" is wrong.
SOLUTION   Modify the volumetric compensation file.

23019 **Error in the volumetric compensation file: the value of the file parameter "INTERVAL_TYPE" is incorrect.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Parameter "INTERVAL_TYPE" is wrong.
SOLUTION   Modify the volumetric compensation file.

23020 **Volumetric compensation file error: the value of the file parameter "VALUE_TYPE" is incorrect.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Parameter "VALUE_TYPE" is wrong.
SOLUTION   Modify the volumetric compensation file.

23021 **Error in the volumetric compensation file: the coordinate of the first axis is outside the lower limit of the volume.**

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The coordinate of the first axis is outside the lower limit of the volume.
SOLUTION   Modify the coordinates or file of the volumetric compensation.

23022 Error in the volumetric compensation file: the coordinate of the first axis is outside the upper limit of the volume.

DETECTION During system start-up or validation of machine parameters.

CAUSE The coordinate of the first axis is outside the upper limit of the volume.

SOLUTION Modify the coordinates or file of the volumetric compensation.

23023 Error in the volumetric compensation file: the coordinate of the second axis is outside the lower limit of the volume.

DETECTION During system start-up or validation of machine parameters.

CAUSE The coordinate of the second axis is outside the lower limit of the volume.

SOLUTION Modify the coordinates or file of the volumetric compensation.

23024 Error in the volumetric compensation file: the coordinate of the second axis is outside the upper limit of the volume.

DETECTION During system start-up or validation of machine parameters.

CAUSE The coordinate of the second axis is outside the upper limit of the volume.

SOLUTION Modify the coordinates or file of the volumetric compensation.

23025 Error in the volumetric compensation file: the coordinate of the third axis is outside the lower limit of the volume.

DETECTION During system start-up or validation of machine parameters.

CAUSE The coordinate of the third axis is outside the lower limit of the volume.

SOLUTION Modify the coordinates or file of the volumetric compensation.

23026 Error in the volumetric compensation file: the coordinate of the third axis is outside the upper limit of the volume.

DETECTION During system start-up or validation of machine parameters.

CAUSE The coordinate of the third axis is outside the upper limit of the volume.

SOLUTION Modify the coordinates or file of the volumetric compensation.

23027 Error in the volumetric compensation file: error in the units of the volumetric file.

DETECTION During system start-up or validation of machine parameters.

CAUSE The units of the file are not the same.

SOLUTION Modify the volumetric compensation file.

23028 Error in the volumetric compensation file: error in the units of the volumetric file.

DETECTION During system start-up or validation of machine parameters.

CAUSE The units of the file are invalid.

SOLUTION Modify the volumetric compensation file.

23029 Error in the volumetric compensation file: the value of the file parameter "HEADER:COUNT" is incorrect.

DETECTION During system start-up or validation of machine parameters.

CAUSE Parameter "HEADER:COUNT" is wrong.

SOLUTION Modify the volumetric compensation file.

23030 Error in the volumetric compensation file: error in the units of the volumetric file.

DETECTION During system start-up or validation of machine parameters.

CAUSE The units of the file are invalid.

SOLUTION Modify the volumetric compensation file.

23031 Error in the volumetric compensation file: point intervals are not constant.

DETECTION During system start-up or validation of machine parameters.

CAUSE The intervals of the points are not constant.

SOLUTION Modify the volumetric compensation file.

23032 Error in the volumetric compensation file: the points in the volumetric file are not in the expected order.

DETECTION During system start-up or validation of machine parameters.

CAUSE The points are not in the expected order.

SOLUTION Modify the volumetric compensation file.

23033   Error in the volumetric compensation file: the interval is negative.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The interval is negative.
SOLUTION   Modify the volumetric compensation file.

23034   Error in the volumetric compensation file: the value of the file parameter "AXIS_BEGIN" is incorrect.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Parameter "AXIS_BEGIN" is wrong.
SOLUTION   Modify the volumetric compensation file.

23035   Error in the volumetric compensation file: the value of the "AXIS_END" file parameter is incorrect.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Parameter "AXIS_END" is wrong.
SOLUTION   Modify the volumetric compensation file.

23036   Error in the volumetric compensation file: the value of the file parameter "AXIS_LENGTH" is incorrect.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Parameter "AXIS_LENGTH" is wrong.
SOLUTION   Modify the volumetric compensation file.

23037   Error in the volumetric compensation file: Minimum value exceeded.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The volumetric definition file does not contain the proper data.
SOLUTION   Set the maximum value of the file as the maximum value in the table.

23038   Error in the volumetric compensation file: Maximum value exceeded.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The volumetric definition file does not contain the proper data.
SOLUTION   Set the maximum value of the file as the maximum value in the table.

23039   Error in the volumetric compensation file: Wrong range value.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The volumetric definition file does not contain the proper data.
SOLUTION   Change the value of the range to the difference between the maximum and minimum value.

23040   Error in the volumetric compensation file: volumetric file cannot be opened.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The volumetric compensation definition file cannot be opened.
SOLUTION   Modify the volumetric compensation file.

23041   Error in the volumetric compensation file: not enough memory to load the volumetry.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The volumetric compensation definition file cannot be loaded due to lack of memory.
SOLUTION   Modify the volumetric compensation file.

23042   Error in the volumetric compensation file: compensation is deactivated.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   Compensation vector has not been calculated; compensation is deactivated.
SOLUTION   Activate volumetric compensation.

23043   Error in the volumetric compensation file: incomplete volumetric file.

DETECTION   During system start-up or validation of machine parameters.
CAUSE   The volumetric compensation definition file is not complete, one element is missing.
SOLUTION   Modify the volumetric compensation file.

23101   'Error when analyzing the volumetric compensation definition file'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The volumetric definition file does not contain the proper data.

Basic volumetric compensation:

- The file is not in csv format.
- The format of the file defined in VCOMPFILE is incorrect.
- The file defined in VCOMPFILE does not have all the points defined. The number of points is defined by the expression:

    $$NPOINTSAX1 \times NPOINTSAX2 \times NPOINTSAX3$$

Medium and large volumetric compensation:

- The format of the file defined in VCOMPFILE is incorrect.

SOLUTION	Select a proper file. The file must be in csv format and have all the points defined in the basic volumetric compensation.

23101   'Error when analyzing the volumetric compensation definition file'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The volumetric definition file does not contain the proper data.
SOLUTION	Select a proper file.

23102   'Volumetric compensation system error: Compensation off'

DETECTION	During execution.
CAUSE	The CNC has detected an internal error when calculating the volumetric compensation and has canceled the compensation values.
SOLUTION	Contact Fagor Automation.

23103   'The VOLCOMP mark cannot be activated without a validated table'

DETECTION	During execution.
CAUSE	The PLC has tried to activate a volumetric compensation that is not defined properly and therefore the machine cannot be compensated.
SOLUTION	Define a proper volumetric compensation in the machine parameters (VOLCOMP parameters).

23104   'The VOLCOMP mark cannot be activated while another table is active on common axes'

DETECTION	During execution.
CAUSE	The PLC has tried to activate a volumetric compensation while another one is active and one of the axes used is common to both compensations.
SOLUTION	There cannot be two active volumetric compensations having an axis in common. Before activating a compensation, cancel the active one using the corresponding PLC mark.

23151   Table number does not exist.

DETECTION	During execution.
CAUSE	The machine parameter table name or number (feedback, motor, tandem, etc.) indicated in the variable does not exist.
SOLUTION	The name or number indicated in the variable must be a valid table.

23152   Table name too long.

DETECTION	During execution.
CAUSE	The name of the machine parameter table (feedback, motor, tandem, etc.) indicated in the variable does not exist, or is too long.
SOLUTION	The name specified in the variable must be a valid table.

23153   The volumetric offset file is not valid.

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The volumetric offset file (parameter VCOMPFILE) is empty or not properly formatted.
SOLUTION	Add a valid file.

23154   Error when analyzing the volumetric compensation definition file.

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	Error when analyzing the volumetric compensation definition file.
SOLUTION	Add a valid file.

24000-24999

24001   EtherCAT: Datalink not found.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The EtherCAT cable is disconnected.
SOLUTION   Connect the EtherCAT cable.

24002   EtherCAT: Missing package.

DETECTION   During CNC operation.
CAUSE   The EtherCAT cable is disconnected or some slave is not working properly.
SOLUTION   Check the connection and the condition of the slaves.

24003   EtherCAT: Wrong CRC.

DETECTION   During CNC operation.
CAUSE   The EtherCAT cable is disconnected or there is interference in the bus.
SOLUTION   Check the connection and the condition of the slaves.

24004   EtherCAT: Working Counter error.

DETECTION   During CNC operation.
CAUSE   Wrong operation of some of the EtherCAT slaves.
SOLUTION   Check the connection and the condition of the slaves.

24010   EtherCAT: Configuration of ENI file does not exist.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The ENI file for configuring the EtherCAT network does not exist.
SOLUTION   Copy in Mtb/Data the ENI file for configuring the EtherCAT network. The file must not be named fagor_ethercatConfFile.xml.

24011   EtherCAT: Incorrect configuration of ENI file format.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The ENI file for configuring the EtherCAT network has an illegible format for the CNC. The configurator that generated the ENI file may not be compatible with the CNC. The file may be incomplete or corrupt.
SOLUTION   Check the file.

24020   EtherCAT: Routed file does not exist.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The file for accessing the EtherCAT bus resources from the CNC-PLC Fagor system does not exist.
SOLUTION   Generate a routed file (mapping) with the Fagor EtherCAT Mapper application and copy it into Mtb/Data. The file must be named resourceRouterConf.xml.

24021   EtherCAT: Incorrect routed file format.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The routed file (mapping) of the EtherCAT resources has an unreadable format for the CNC. The file may be incomplete or corrupt.
SOLUTION   Check the file.

24022   EtherCAT: Error during resource routing process.

DETECTION   On CNC power-up or when validating the machine parameters.
CAUSE   The possible causes are:
• The file format is wrong.
• The version of the application Fagor EtherCAT Mapping is incompatible with the version of the CNC.
• The version does not support any of the mapped resources.
SOLUTION   Contact Fagor Automation.

24023   EtherCAT: Non-Fagor EtherCAT nodes. Software option not allowed.

DETECTION   During CNC power-up or after a reset.

CAUSE   The initialisation of EtherCAT has detected some non-Fagor mode and the software option for connecting third-party nodes is not active.

SOLUTION   Remove the non-Fagor modes from the EtherCAT bus or contract the software option for third-party nodes

24255   EtherCAT: Unknown error.

DETECTION   During CNC operation.

CAUSE   Error of unknown cause in the EtherCAT bus.

SOLUTION   Contact Fagor Automation.

24499   EtherCAT: Node error. More information on diagnostics.

DETECTION   During CNC operation.

CAUSE   Node error.

SOLUTION   Refer to the diagnosis mode for more information.

24500   Bus failure has been fixed.

DETECTION   On the diagnostic screen, during execution.

CAUSE   Bus fault condition corrected.

SOLUTION   No action required.

24501   Defective UM supply voltage.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24502   UM supply voltage OK.

DETECTION   On the diagnostic screen, during execution.

CAUSE   Fault condition corrected.

SOLUTION   No action required.

24503   US supply voltage defective.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24504   Supply voltage US OK.

DETECTION   On the diagnostic screen, during execution.

CAUSE   Fault condition corrected.

SOLUTION   No action required.

24505   UL supply voltage defective.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24506   Supply voltage UL OK.

DETECTION   On the diagnostic screen, during execution.

CAUSE   Fault condition corrected.

SOLUTION   No action required.

24507   Restart the device after the watchdog.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24508   Generic local bus failure.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24509   No modules found.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24510   Incorrect module found for position %u.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24511   More devices than expected.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24512   Application in module %u not ready.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24513   Module %u device reset.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24514   Module %u communication timeout.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24515   Multiple communication errors in module %u.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24516   I/O data communication error in module %u.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24517 Management data communication error in module %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24518 Bus error in device %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24519 Configuration error in module %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24520 Hardware error in module %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24521 Firmware error in module %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24522 Peripheral error in module %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24523 Peripheral error in module %u has been fixed.

DETECTION On the diagnostic screen, during execution.

CAUSE Fault condition corrected.

SOLUTION No action required.

24524 Localbus module configuration does not match module %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24525 Start with the remaining system not allowed, %u expected modules, modules found %u.

DETECTION On the diagnostic screen, during execution.

CAUSE An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

FAGOR AUTOMATION

CNCelite
8058 8060
8065 8070

REF. 2209

·290·

24526   The application requested a reconfiguration of the local bus.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24527   Start with the remaining system, %u expected modules, modules found %u.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24550   Bus failure has been fixed.

DETECTION	On the diagnostic screen, during execution.
CAUSE	Fault condition corrected.
SOLUTION	No action required.

24551   A firmware error has occurred.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24552   Maximum number of exceeded modules.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24553   The number of modules is zero.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24554   Maximum number of exceeded I/O points.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24555   A module with an unsupported chip version was detected.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24556   Unsupported manufacturer's device module found.

DETECTION	On the diagnostic screen, during execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24557   Serious error in localbus module

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24558   A hardware or firmware error has occurred.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24559   A hardware or firmware error has occurred.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24560   A hardware or firmware error has occurred.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24561   A hardware or firmware error has occurred.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24562   A hardware or firmware error has occurred.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24563   The local bus could not be activated due to malfunctions.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24564   The local bus could not be activated due to malfunctions.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24565   The local bus could not be activated due to malfunctions.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24566 **The local bus could not be activated due to malfunctions.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24567 **Localbus module %u lost.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24568 **New localbus module %u added.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24569 **Local bus module %u incorrectly positioned.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24570 **Local bus module %u error.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24571 **The local bus module %u has an incorrect data processing width.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24572 **Error in retrieving local bus configuration.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24573 **Module %u: Channel short-circuit %u.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24574 **Module %u: Input overload group %u.**

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24575   Module %u: input/short-circuit overload.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24576   Module %u: input/short-circuit overload.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24577   Module %u: I/O supply failure.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24578   Module %u: I/O supply failure.

DETECTION   On the diagnostic screen, during execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24579   Module %u: Actuator supply not found.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24580   Module %u: 24 V supply defective.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24581   Module %u: 5 V supply defective.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24582   Module %u: Cold junction for Canal %u not valid.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24583   Module %u: supply voltage defective.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24584   Restart the device after the watchdog.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24585   Module %u: Device error.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24586   Module %u: Error in flash format.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24587   Module %u: Parameter table not valid.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24588   Module %u: Encoder error for Canal %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24589   Module %u: Encoder error for Canal %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24590   Module %u: Receive buffer full.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24591   Module %u: Send buffer full.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24592   Module %u: Device error.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24593   Module %u: Open circuit for Channel %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24594   Module %u: Input error for the incremental encoder Channel %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24595   Module %u: Overload channel %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24596   Module %u: Underground channel %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24597   Localbus module configuration does not match module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24600   Bus failure has been fixed.

DETECTION   During execution.

CAUSE   Bus fault condition corrected.

SOLUTION   No action required.

24601   Defective UM supply voltage.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24602   UM supply voltage OK.

DETECTION   During execution.

CAUSE   Fault condition corrected.

SOLUTION   No action required.

24603   US output supply voltage defective.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24604   Supply voltage US OK.

DETECTION   During execution.

CAUSE   Fault condition corrected.

SOLUTION   No action required.

24605 UL supply voltage defective.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24606 Supply voltage UL OK.

DETECTION   During execution.

CAUSE   Fault condition corrected.

SOLUTION   No action required.

24607 Restart the device after the watchdog.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24608 Generic local bus failure.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24609 No modules found.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24610 Incorrect module found for position %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24611 More devices than expected.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24612 Application in module %u not ready.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24613 Module %u device reset.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24614   Module %u communication timeout.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24615   Multiple communication errors in module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24616   I/O data communication error in module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24617   Management data communication error in module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24618   Bus error in device %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24619   Configuration error in module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24620   Hardware error in module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24621   Firmware error in module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24622   Peripheral error in module %u.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24623   Peripheral error in module %u has been fixed.

DETECTION	During execution.
CAUSE	Fault condition corrected.
SOLUTION	No action required.

24624   Localbus module configuration does not match module %u.

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24625   Start with the remaining system not allowed, %u expected modules, modules found %u.

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24626   The application requested the reconfiguration of the local bus.

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24627   Start with the remaining system, %u expected modules, modules found %u.

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24650   Request for change of status: From %u to:%u.

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24651   %u.

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24652   Incorrect calculation of bus cycle time (local timer too slow).

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24653   Sync Manager %d invalid address (%d).

DETECTION	During execution.
CAUSE	An EtherCAT node has sent an emergency indicating this error condition.
SOLUTION	It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24654   Sync Manager %d invalid size (%d).

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24655   Sync Manager %d invalid configuration (%d).

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24656   Invalid DC activation record.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24657   SyncType setting (0x1C32.1 or 0x1C33.1) not supported. Check the supported DC registers and SyncTypes(0x1C32.4 and 0x1C33.4).

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24658   Software exception, file: %s, line: %d.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24659   CoE emergency code: 0x%X, additional information: 0x%X.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24660   Overcurrent on digital outputs.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24661   Internal voltage error +15 V/-15 V.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24662   Overcurrent on analogue outputs.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·300·

24663   Memory test failed.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24664   PT100 input broken or not connected.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24665   Digital IOs board: No +24 V voltage.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24666   The device has restarted after a watchdog.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24667   Internal software emergency.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24668   Incorrect analogue board identifier.

DETECTION   During execution.

CAUSE   An EtherCAT node has sent an emergency indicating this error condition.

SOLUTION   It is recommended to consult the specific manual of the node, the diagnostic information or the additional value accompanying the error (hexadecimal code of the emergency).

24669   Firmware error in RIOR-E, BOOT mode.

DETECTION   During execution.

CAUSE   A RIOR-E node has been detected in BOOT mode.

SOLUTION   Reload the application firmware.

80000-80999

80001   internal error #00: error text is missing E%D P0=%D.

DETECTION   During execution.
CAUSE   Error in the execution of subroutine.
SOLUTION   Check the program.

80010   G%D A%D: Parameter A not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter A is not allowed.
SOLUTION   Check the program.

80011   G%D B%D: Parameter B is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter B is not allowed.
SOLUTION   Check the program.

80012   G%D C%D: Parameter C is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter C is not allowed.
SOLUTION   Check the program.

80013   G%D D%D: Parameter D is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter D is not allowed.
SOLUTION   Check the program.

80014   G%D E%D: Parameter E is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter E is not allowed.
SOLUTION   Check the program.

80015   G%D F%D: Parameter F is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter F is not allowed.
SOLUTION   Check the program.

80016   G%D G%D: Parameter G is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicted, parameter G is not allowed.
SOLUTION   Check the program.

80017   G%D H%D: Parameter H is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter H is not allowed.
SOLUTION   Check the program.

80018   G%D I%D: Parameter I is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter I is not allowed.
SOLUTION   Check the program.

80019   G%D J%D: Parameter J is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter J is not allowed.
SOLUTION   Check the program.

80020   G%D K%D: Parameter K is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter K is not allowed.
SOLUTION   Check the program.

80021   G%D L%D: Parameter L is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter L is not allowed.
SOLUTION   Check the program.

80022   G%D M%D: Parameter M is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter M is not allowed.
SOLUTION   Check the program.

80023   G%D N%D: Parameter N is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter N is not allowed.
SOLUTION   Check the program.

80024   G%D O%D: Parameter O is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter O is not allowed.
SOLUTION   Check the program.

80025   G%D P%D: Parameter P is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter P is not allowed.
SOLUTION   Check the program.

80026   G%D Q%D: Parameter Q is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter Q is not allowed.
SOLUTION   Check the program.

80027   G%D R%D: Parameter R is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter R is not allowed.
SOLUTION   Check the program.

80028   G%D S%D: Parameter S is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter S is not allowed.
SOLUTION   Check the program.

80029   G%D T%D: Parameter T is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter T is not allowed.
SOLUTION   Check the program.

80030   G%D U%D: Parameter U is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter U is not allowed.
SOLUTION   Check the program.

80031   G%D V%D: Parameter V is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter V is not allowed.
SOLUTION   Check the program.

80032   G%D W%D: Parameter W is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter W is not allowed.
SOLUTION   Check the program.

80033   G%D X%D: Parameter X is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter X is not allowed.
SOLUTION   Check the program.

80034   G%D Y%D: Parameter Y is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter Y is not allowed.
SOLUTION   Check the program.

80035   G%D Z%D: Parameter Z is not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, parameter Z is not allowed.
SOLUTION   Check the program.

80036   G%D one or more parameters A..Z not allowed.

DETECTION   During execution.
CAUSE   In the subroutine indicated, no parameter is allowed.
SOLUTION   Check the program.

85000-85999

85850   G8585 not allowed with temporary origin G92 active.

DETECTION   During execution.
CAUSE   The function G8585 has been programmed and there is an active coordinate preset (G92).
SOLUTION   Cancel the presetting of coordinates before executing the function G8585.

85851   G8585 does not allow programming of D and R or I and J at the same time.

DETECTION   During execution.
CAUSE   The radius and the outside diameter, or the radius and interior diameter have been programmed.
SOLUTION   Check the program. Program each dimension only once, with the radius or the diameter.

85852   G8585 with R&D, which is interior diameter > exterior diameter.

DETECTION   During execution.
CAUSE   The interior diameter is greater than the exterior.
SOLUTION   Program the exterior diameter greater than the interior.

85860   G8585 wrong configuration: set parameter MPA.LONGAXIS.%D=No.

DETECTION   During execution.
CAUSE   The axis indicated is defined as longitudinal.
SOLUTION   Define the LONGAXIS parameter of the axis with "No" value.

85861   G8585 wrong configuration: set parameter MPA.FACEAXIS.%D=No.

DETECTION   During execution.
CAUSE   The axis indicated is defined as face axis.
SOLUTION   Define the FACEAXIS parameter of the axis with "No" value.

85862   G8585 Wrong configuration: AXISNAME=%D does not exist.

DETECTION   During execution.
CAUSE   Subroutine G8585 has attempted to use a nonexistent axis.
SOLUTION   Check subroutine G8585 and the assignment of data to the variable V.S.G84585CFG.

85863   G8585 wrong configuration: incorrect value of the variable V.S.G84585CFG=%D.

DETECTION   During execution.
CAUSE   The value assigned to the variable is not valid.
SOLUTION   Check the assignment of data to the variable in subroutine G8585.

85870   Internal error #90 in G8585: KINID=%D, TYPE=%D.

DETECTION   During execution.
CAUSE   Error in the execution of subroutine G8585, in the kinematic indicated.
SOLUTION   Check subroutine G8585.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

RCS-S MODULE.

E034   Error identifying the CPU board.
Contact Fagor Automation.

E038   Code checksum error.
The loaded checksum of the program code is not correct. Load the software again. If the problem persists, the RAM, FLASH memories or the loaded code may be defective. Contact Fagor Automation.

E039   Error on the Sercos ® board .
Contact Fagor Automation.

E040   Failure of the clock of the SerCon® board.
Contact Fagor Automation.

E041   SerCon memory corrupted.
Contact Fagor Automation.

E044   Corrupted file system in flash.
Contact Fagor Automation.

E045   Error reading analog offsets.
Contact Fagor Automation.

E046   Error identifying the axis board.
Contact Fagor Automation.

E047   Unknown axis board identification.
Contact Fagor Automation.

E04   Error in FPGA code.
The FPGA code seems to be corrupted or is missing. Try reloading the software version. If the problem persists, the RAM, FLASH memories or the loaded code may be defective. Contact Fagor Automation.

E083   The FPGA could not be loaded.
Restart the counter. If the problem persists, contact Fagor Automation.

E084   Reset due to Watchdog.
The system has restarted because watchdog kicked in. If the problem is repetitive and/or persists, contact Fagor Automation.

E085   Error in power supply for the analog outputs.
Contact Fagor Automation.

E133   Broken wire, A signal of E21A feedback.
E233   Broken wire, A signal of E21B feedback.
E333   Broken wire, A signal of E21C feedback.
E433   Broken wire, A signal of E21D feedback.
For differential TTL feedback. The A and/or /A signal wire may be broken or disconnected.
Check the connection. Check the level of the signals provided by the feedback device.

E134   Broken wire, B signal of E21A feedback.
E234   Broken wire, B signal of E21B feedback.
E334   Broken wire, B signal of E21C feedback.
E434   Broken wire, B signal of E21D feedback.
For differential TTL feedback. The B and/or /B signal wire may be broken or disconnected.
Check the connection. Check the level of the signals provided by the feedback device.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

E135 Broken wire, I0 signal of E21A feedback.
E235 Broken wire, I0 signal of E21B feedback.
E335 Broken wire, I0 signal of E21C feedback.
E435 Broken wire, I0 signal of E21D feedback.
For differential TTL feedback. The I0 and/or /I0 signal wire may be broken or disconnected. Check the connection. Check the level of the signals provided by the feedback device.

E136 Broken wire, AL signal of E21A feedback.
E236 Broken wire, AL signal of E21B feedback.
E336 Broken wire, AL signal of E21C feedback.
E436 Broken wire, AL signal of E21D feedback.
For differential TTL feedback. The AL and/or /AL signal wire may be broken or disconnected. Check the connection. Check the level of the signals provided by the feedback device.

E137 Received alarm signal at E21A feedback.
E237 Received alarm signal at E21B feedback.
E337 Received alarm signal at E21C feedback.
E437 Received alarm signal at E21D feedback.
The encoder has generated a signal alarm due to its poor performance. Check the encoder documentation to determine the source of (reason for) the alarm. It could also be a false positive due to poor connection.

E138 Amplitude alarm at E21A feedback.
E238 Amplitude alarm at E21B feedback.
E338 Amplitude alarm at E21C feedback.
E438 Amplitude alarm at E21D feedback.
Too much damping or saturation of A and/or B signals. The wiring, the feedback device or the feedback of the Sercos counter may have some problem.Check the condition (shape) of the cable, of the connection or of the feedback device. If the problem persists, contact Fagor Automation.

E140 Feedback frequency too high at E21A feedback.
E240 Feedback frequency too high at E21B feedback.
E340 Feedback frequency too high at E21C feedback.
E440 Feedback frequency too high at E21D feedback.
The axis moving speed exceeds the sampling capability of the Sercos counter hardware for this type of encoder signal. Decrease the maximum axis moving speed.

E141 Wrong distance between I0, E21A feedback.
E241 Wrong distance between I0, E21B feedback.
E341 Wrong distance between I0, E21C feedback.
E441 Wrong distance between I0, E21D feedback.
Error indicating lack of repeatability of the I0 signal in each encoder revolution. Some pulses are being lost and/or the measurement provided by the feedback device is wrong.

Resettable error. Check that the connection and the wiring are correct. Check that the number of pulses allowed as error margin is not too low. Check that the value of CNC parameter REFPULSE is correct. Contact Fagor Automation. If the error persists, it may be because the feedback device is defective.

E142 Over-current or short-circuit at E21Afeedback.
E242 Over-current or short-circuit at E21B feedback.
E342 Over-current or short-circuit at E21C feedback.
E442 Over-current or short-circuit at E21D feedback.
Either a short-circuit has occurred or the feedback device is over-supplied.
Check cables and connections.

E143 Communication error at E21A feedback. Wrong CRC.
E243 Communication error at E21B feedback. Wrong CRC.
E343 Communication error at E21C feedback. Wrong CRC.
E443 Communication error at E21D feedback. Wrong CRC.
Error reading absolute position value; wrong CRC.

Check the cable and connection of the linear or rotary encoder. Check that the parameters of the feedback device have been set correctly. If the problem persists, contact Fagor Automation.

E144 Communication error at E21A feedback. Acknowledge not received.
E244 Communication error at E21B feedback. Acknowledge not received.
E344 Communication error at E21C feedback. Acknowledge not received.
E443 Communication error at E21D feedback. Acknowledge not received.
Error reading absolute position value; acknowledge not received.

Check the cable and connection of the feedback device. The error could be due to a problem at the feedback device. If the problem persists, contact Fagor Automation.

E145   ABSIND: Synchronism error at E21A feedback.
E245   ABSIND: Synchronism error at E21B feedback.
E345   ABSIND: Synchronism error at E21C feedback.
E445   ABSIND: Synchronism error at E21D feedback.

Synchronism lost between the Sercos counter and the ABSIND module which may have caused a wrong position reading. The error may be due to wrong performance of the ABSIND moduel or to a connection problem between the ABSIND module and the Sercos counter associated with that feedback.

If the problem persists, contact Fagor Automation.

E146   ABSIND: Error reading information at E21A feedback.
E246   ABSIND: Error reading information at E21B feedback.
E346   ABSIND: Error reading information at E21C feedback.
E446   ABSIND: Error reading information at E21D feedback.

It could not read the ABSIND module identification info during system startup. The error may be due to wrong performance of the ABSIND moduel or to a connection problem between the ABSIND module and the Sercos counter associated with that feedback.

If the problem persists, contact Fagor Automation.

E183   INDUCTOSYN: INC signal too low at E21A feedback.
E283   INDUCTOSYN: INC signal too low at E21B feedback.
E383   INDUCTOSYN: INC signal too low at E21C feedback.
E483   INDUCTOSYN: INC signal too low at E21D feedback.

Incremental signal level of the inductosyn/resolver feedback device too low. Check the gain and phase values of the ABSIND module with the adjustment software. Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device. If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

E184   INDUCTOSYN: INC signal too high at E21A feedback.
E284   INDUCTOSYN: INC signal too high at E21B feedback.
E384   INDUCTOSYN: INC signal too high at E21C feedback.
E484   INDUCTOSYN: INC signal too high at E21D feedback.

Incremental signal level of the inductosyn/resolver feedback device too high. Check the gain and phase values of the ABSIND module with the adjustment software. Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device. If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

E185   INDUCTOSYN: GRAY signal missing at E21A feedback.
E285   INDUCTOSYN: GRAY signal missing at E21B feedback.
E385   INDUCTOSYN: GRAY signal missing at E21C feedback.
E485   INDUCTOSYN: GRAY signal missing at E21D feedback.

GRAY signal level of the inductosyn/resolver feedback device missing. Check the gain and phase values of the ABSIND module with the adjustment software. Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device. If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

E186   INDUCTOSYN: Linear encoder unstable on Power-ON, E21A feedback.
E286   INDUCTOSYN: Linear encoder unstable on Power-ON, E21B feedback.
E386   INDUCTOSYN: Linear encoder unstable on Power-ON, E21C feedback.
E486   INDUCTOSYN: Linear encoder unstable on Power-ON, E21D feedback.

The position read when initializing the ABSIND module is unstable. Check that the axis is not moving while starting up or initializing. Check the cable and the connection between the ABSIND module and the inductosyn/resolver feedback device. Check the ground connections in the electrical cabinet (electrical noise on the signals). If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module and/or the cable associated with that feedback. If the error persists, contact Fagor Automation.

E187   INDUCTOSYN: Wrong EEPROM checksum, E21A feedback.
E287   INDUCTOSYN: Wrong EEPROM checksum, E21B feedback.
E387   INDUCTOSYN: Wrong EEPROM checksum, E21C feedback.
E487   INDUCTOSYN: Wrong EEPROM checksum, E21D feedback.

Check the gain and phase values of the ABSIND module with the adjustment software. If the error persists and cannot be changed with the previous actions, try replacing the ABSIND module associated with that feedback. If the error persists, contact Fagor Automation.

E188   INDUCTOSYN: Unknown error at E21A feedback.
E288   INDUCTOSYN: Unknown error at E21B feedback.
E388   INDUCTOSYN: Unknown error at E21C feedback.
E488   INDUCTOSYN: Unknown error at E21D feedback.
The ABSIND module has generated an unknown error. The error is likely due to bad performance. It the error is repetitive and persistent, try replacing the ABSIND module associated with that feedback. If the error persists, contact Fagor Automation S. Coop.

E189   INDUCTOSYN: Incoherent SSITYPE parameter with feedback device type connected to E21A feedback.
E289   INDUCTOSYN: Incoherent SSITYPE parameter with feedback device type connected to E21B feedback.
E389   INDUCTOSYN: Incoherent SSITYPE parameter with feedback device type connected to E21C feedback.
E489   INDUCTOSYN: Incoherent SSITYPE parameter with feedback device type connected to E21D feedback.
Check the dipswitch S50 of the ABSIND module. The selected value must be coherent with CNC machine parameter SSITYPE and with the feedback type (inductosyn/resolver) connected to the ABSIND module.

E192   wrong REFPULSE parameter at E21A feedback.
E292   wrong REFPULSE parameter at E21B feedback.
E392   wrong REFPULSE parameter at E21C feedback.
E492   wrong REFPULSE parameter at E21D feedback.
The value of machine parameter REFPULSE is inconsistent with the type of feedback connected.
Try changing its value. If the problem persists, contact Fagor Automation.

E193   Fagor digital feedback. Error on the absolute track E21A.
E293   Fagor digital feedback. Error on the absolute track E21B.
E393   Fagor digital feedback. Error on the absolute track E21C.
E493   Fagor digital feedback. Error on the absolute track E21D.
It cannot calculate the absolute position. Feedback pulses lost. The analog signal < 0.20 Vpp.
Check that the linear or rotary encoder is working at proper speed. Check mounting tolerances. Clean the glass or the steel tape of the linear encoder. If none of the previous solutions solves the error, replace the rotary encoder or reader head of the linear encoder.

E194   Fagor digital feedback. CPU error. E21A feedback.
E294   Fagor digital feedback. CPU error. E21B feedback.
E394   Fagor digital feedback. CPU error. E21C feedback.
E494   Fagor digital feedback. CPU error. E21D feedback.
Damaged rotary encoder or reader head of linear encoder. Replace the rotary encoder or the reader head of the linear encoder.

E195   Fagor digital feedback. Error at the adjustment potentiometers. E21A feedback.
E295   Fagor digital feedback. Error at the adjustment potentiometers. E21B feedback.
E395   Fagor digital feedback. Error at the adjustment potentiometers. E21C feedback.
E495   Fagor digital feedback. Error at the adjustment potentiometers. E21D feedback.
Damaged rotary encoder or reader head of linear encoder. Replace the rotary encoder or the reader head of the linear encoder.

E196   Fagor digital feedback. Image capturing sensor (CCD) error E21A feedback.
E296   Fagor digital feedback. Image capturing sensor (CCD) error E21B feedback.
E396   Fagor digital feedback. Image capturing sensor (CCD) error E21C feedback.
E496   Fagor digital feedback. Image capturing sensor (CCD) error E21D feedback.
Check mounting tolerances. Clean the glass or the steel tape of the linear encoder.
Damaged rotary encoder or reader head of the linear encoder; the CCD (Charge-Coupled Device) or the LED (Light-Emitting Diode). Replace the rotary encoder or the reader head of the linear encoder.

E197   Fagor digital feedback. Supply voltage out of range. E21A feedback.
E297   Fagor digital feedback. Supply voltage out of range. E21B feedback.
E397   Fagor digital feedback. Supply voltage out of range. E21C feedback.
E497   Fagor digital feedback. Supply voltage out of range. E21D feedback.
The supply voltage must be between 5.3 V and 4.1 V. Check that the power cable of the rotary or linear encoder is not damaged. Also check that the cable length is the right one and that the wiring connection is the right one.

E198   Fagor digital feedback. Parameter error.
E298   Fagor digital feedback. Parameter error.
E398   Fagor digital feedback. Parameter error.
E498   Fagor digital feedback. Parameter error.
Rotary or linear encoder configuration has been lost. Replace the rotary encoder or the reader head of the linear encoder.

E403   MST failure.
Communication problems through fiber optic ring. Check the ring connections and the identification of each module.

E404   MDT failure.

Communication problems through fiber optic ring. Check the ring connections and the identification of each module.

E405   Invalid phase(>4).

Communication problems through fiber optic ring. Check the ring connections and the identification of each module.

E406   Phase up-shift error.

Communication problems through fiber optic ring. Check the ring connections and the identification of each module.

E407   Phase down-shift error.

Communication problems through fiber optic ring. Check the ring connections and the identification of each module.

E410   Noise resets Sercon.

Noise gets in through the connection of the internal bus resetting the SerCon but not with the VeCon2.

E411   Error when receiving telegrams.

Communication problems through fiber optic ring. Check the ring connections and the identification of each module.

E412   Delayed synchronism message.

Communication problems through fiber optic ring. Check the ring connections and the identification of each module.

E413   Handshake error in Sercos counter.

The master element (CNC) sends a synchronism message in every cycle (usually 4ms) that synchronizes the drives. This error appears when they can't synchronize or lose synchronism.

Maybe the CNC has not sent this message or if it has, it wasn't in the precise moment. Check the transmission cable or check that there is no noise in the transmission.

TOOL AND TOOL MAGAZINE TABLE

'The tool cannot be loaded into position %1'

DETECTION	During the manual loading of a tool from a dialog box or from the magazine list. 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	While unloading a tool manually from a dialog box or from the magazine list. Loading the tool table or 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 with the data either does not correspond to the table to be loaded or has been modified 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	While 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 CNC cannot find the tool.
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 position of a tool 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	The CNC has tried 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 position of the magazine.

'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 the target folder.

'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.

'Error opening file %1'

DETECTION	When loading, saving or printing the tool table or magazine table.
CAUSE	The possible causes are:

- When loading a table. The data file does not exist, it does not have read permission or is being used by another application.
- When saving a table or printing a table to a file. 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.
- When printing a table to a printer, the printer either does not exist or is configured wrong.

SOLUTION	The possible solutions are:

- When loading a table. Choose existing files, indicate their path correctly, give them read permission or close the application that is using them.
- When saving a table or printing a table 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 a table to a printer, choose an existing printer that is configured correctly.

FAGOR
AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·312·

PROFILE EDITOR

'Unresolved profile'

CAUSE	The profile to be saved has not been resolved completely.
SOLUTION	Resolve the profile to be saved. The CNC only allows saving resolved profiles.

'Insufficient memory'

CAUSE	There isn't enough system memory to go on with the profile editor.
SOLUTION	Contact Fagor.

'Geometry error'

CAUSE	The selected profile has an invalid geometry. 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 wrong.
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	The editor cannot find an arc element that is coherent with the known data.
SOLUTION	Correct the data of the element.

'Element not tangent to the previous one'

CAUSE	The tangency of an element is not coherent with the previous element.
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 between which it has been defined.

'Error in the axis of the plane'

CAUSE	Some axis of the plane is wrong. Both axes of the plane are the same or one of the axes 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 AUTOMATION

CNCelite
8058 8060
8065 8070

Ref. 2209

·313·

User notes:

FAGOR

AUTOMATION

CNCelite
8058   8060
8065   8070

REF. 2209

·314·

User notes:

FAGOR AUTOMATION