CNC

8060
8065

Error solving manual

(Ref: 1402)

FAGOR

FAGOR AUTOMATION

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 a 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. An antivirus software is highly recommended if the CNC is connected directly to another PC, it is part of a computer network or floppy disks or other computer media is used to transmit data.

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.

All rights reserved. No part of this documentation may be transmitted, transcribed, stored in a backup device or translated into another language without Fagor Automation's consent. Unauthorized copying or distributing of this software is prohibited.

The information described in this manual may be changed due to technical modifications. Fagor Automation reserves the right to make any changes to the contents of this manual without prior notice.

All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.

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, thus no absolute match is guaranteed. Anyway, the contents of the manual is periodically checked making and including the necessary corrections in a future 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

0000-0999.....5
1000-1999.....17
2000-2999.....100
3000-3999.....107
4000-4999.....123
5000-5999.....132
6000-6999.....135
7000-7999.....144
8000-8999.....148
9000-9999.....176
23000-23999.....180
RCS-S module.....181
Tool and tool magazine table 186
Profile editor.....188

FAGOR

CNC 8060
CNC 8065

(REF: 1402)

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.

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.

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.

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 assigned to the channel of the oscilloscope is a simulation, asynchronous variable or is a string.

SOLUTION Refer to the operating manual for the list of variables that are admitted.

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 user has set a machine parameter that represents the name of an axis with a name that does not exist in parameter AXISNAME.

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.

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 velocity-Sercos'

DETECTION During the validation of the machine parameters.

CAUSE An axis of a tandem axis is not velocity-Sercos.

SOLUTION Both axes of a tandem axis must be velocity-Sercos.

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.

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 execution.

CAUSE The user has tried to validate a machine parameter table while a part-program is in execution or interrupted.

SOLUTION Wait for the program execution to finish or cancel the program execution in all the channels.

0106 'Parameters cannot be validated Spindle or axis moving'

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 \windows\system32\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.

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 '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 The CNC could not open a file for reading or writing. The file does not have the proper permissions, is not accessible or is corrupt.

SOLUTION Check that the file exists and that it has the proper permissions for the action to be carried out (read/write). If the file is corrupt, its data has been lost.

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.

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.

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 'LR 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'

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'

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 over-temperature'

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. This error shows the current temperature.

SOLUTION If the error persists, turn the CNC off and contact the Service Department.

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.

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 a nonexistent main axis'

DETECTION During execution.

CAUSE The programmed function needs one or two main axes and they are missing in the channel.

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.

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.

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.

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)'

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 'No access to the file'

DETECTION During execution.

CAUSE The CNC cannot access the program or subroutine.

SOLUTION Check that the files are valid and are not corrupted. When calling subroutines, check that the name and the path are correct. If the path 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).

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.

SOLUTION Check the name of the local subroutine in the calling block is the same as the name that appears in its definition. The local subroutines must be defined at the beginning of the program.

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.

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.

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.

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.

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

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

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).

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.

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 function #RTCP or #TLC, or a coordinate transformation (#CS/ACS) in mode 6 without having a kinematics active.
SOLUTION 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.

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'

DETECTION During execution.
CAUSE The block label defined in the $GOTO or #RPT instruction does not exist in the program.
SOLUTION Define the jump label in some point of the program.

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 positioning (orienting) speed 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.

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.

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 movement'

DETECTION During execution.

CAUSE A movement and an M function with associated subroutine and execution before the movement have been programmed in the same block. The subroutines are always executed at the end of the block; therefore, the M function will never be executed before the programmed movement.

SOLUTION Program the M function in another block or define the M function in the machine parameters to be executed after the 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 'It cannot be programmed if the spindle is not controlled in position'

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.

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

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 In a canned cycle, a parameter has the wrong value.

SOLUTION Check the program.

1446 'Starting block not allowed in a local subroutine'

DETECTION During execution.

CAUSE The starting block cannot be a block of a local subroutine.

SOLUTION Select another starting block.

1447 'Software option not allowed'

DETECTION During execution.

CAUSE The CNC does not have the software option required to execute the programmed command.

SOLUTION In diagnosis mode, it is possible to check the software options offered by the CNC.

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

DETECTION During execution.

CAUSE The possible causes are:

- The spindle type does not allow positioning the tool 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.

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.

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 'POS and T must be programmed in the same block'

DETECTION	During execution.
CAUSE	The tool T and the position POS has not been programmed in the same block.
SOLUTION	The tool T and the position it must be occupied in the magazine must be programmed in the same block.

1525 'The axis cannot be exchanged'

DETECTION	During execution.
CAUSE	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 or #WARNING, an invalid identifier has been programmed.
SOLUTION	The valid identifiers are %D or %d to display a number and %% to display the character "%".

1529 'Expecting list of identifiers or "]" after end-quotes'

DETECTION	During execution.
CAUSE	In the instruction #MSG, #ERROR or #WARNING , an invalid character has been programmed after the message to be displayed.
SOLUTION	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.

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.

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.

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.

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.

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.

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 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 The channel number must be between square brackets and followed by a ".".
SOLUTION Check the program. Check the syntax of the variable.

1662 'Unknown variable type'

DETECTION During execution.
CAUSE The programmed variable type does not exist.
SOLUTION Check the program. Check the syntax of the variable.

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.

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 While reading (V.)MPA.MULAXISNAMEn[grupo] variables.
CAUSE 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.

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, although exists, has no permission to be accessed from the program.
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.

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.

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.

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.

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 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.

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).

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.

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.

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 #PROBE probing cycle.
CAUSE	There is no probe at the spindle so the cycle can be executed.
SOLUTION	Execute the cycle with a tool at the spindle or with a probe, depending on the 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.

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 out'

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 allowed tolerance (parameter L or M).
SOLUTION	Change the tool and execute the cycle again.

1864 '#PROBE 1: Parameters U, V, W must be greater than X, Y, Z'

DETECTION 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.

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.

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.

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 tool radius compensation between arcs'

DETECTION During execution.

CAUSE The tool radius is too large for the programmed arc.

SOLUTION Programmed a smaller tool radius for machining the programmed arc, eliminate the programmed profile or program a larger arc.

2011 'Compensation movement on circular path (profile damaged)'

DETECTION During execution.

CAUSE The tool radius is too large for the programmed arc.

SOLUTION Programmed a smaller tool radius for machining the programmed arc, eliminate the programmed profile or program a larger arc.

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.

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.

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 'Programming instruction #ROUNDPAR with 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 'Programming instruction #ROUNDPAR with the wrong type of parameter'

DETECTION During execution.
CAUSE The programmed rounding type is not valid.
SOLUTION Check the format of the instruction in the programming manual.

2106 '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 'Undetermined initial tangent in spline'

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 'Undetermined final tangent in spline'

DETECTION During execution.
CAUSE Either the final point of the spline is not the same as the later 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 activating command 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 'Programming instruction #ASPLINE with wrong parameter'

DETECTION During execution.
CAUSE The type of tangency is wrong.
SOLUTION Check the correct format in the programming manual.

2116 'Error when activating the spline'

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.

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.

2133 'Master axis missing'

DETECTION During execution.
CAUSE The master axis of the coupling is not available in the channel.
SOLUTION Program the coupling with axes from the channel.

2134 'Slave axis missing'

DETECTION During execution.
CAUSE The slave axis of the coupling is not available in the channel.
SOLUTION Program the coupling with axes from the channel.

2135 'Zero maximum spindle speed programmed'

DETECTION During execution.
CAUSE In constant surface speed mode, the maximum spindle speed has not been programmed.
SOLUTION Program the maximum rpm of the spindle.

2136 'Zero maximum cutting speed programmed'

DETECTION During execution.
CAUSE In constant surface speed mode, the cutting speed has not been programmed.
SOLUTION Program the cutting speed.

2137 'FACEAXIS has not been defined'

DETECTION During execution.
CAUSE No face axis has been defined.
SOLUTION Set machine parameter FACEAXIS = TRUE for the face axis.

2138 'Threading axis missing'

DETECTION During execution.
CAUSE The threading axis is not in the channel.
SOLUTION 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 'Threading movement not programmed'

DETECTION During execution.
CAUSE No movement has been programmed for the threading axis.
SOLUTION Program the movement for the threading axis.

2140 'Wrong parameter in programming instruction #SLOPE'

DETECTION During execution.
CAUSE A parameter of the programming instruction #SLOPE is wrong.
SOLUTION Check the correct format in the programming manual.

2141 'Circular path not allowed (less than 2 axes)'

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 'Internal error in tangential control'

DETECTION During execution.
CAUSE Tangential control is not possible.
SOLUTION Eliminate the block that caused the error. Contact Fagor.

2150 'WARNING: #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.

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.

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.

2307 'The axis cannot be activated as Hirth, because it is part of the #VIRTAX transformation'

DETECTION During execution.

CAUSE The CNC has tried to activate a Hirth axis while it belonged to the virtual tool axis transformation, #VIRTAX.

SOLUTION First deactivate the virtual tool axis transformation #VIRTAX and then activate the axis as Hirth axis.

2308 'WARNING: 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 'Programming not allowed while virtual tool axis transformation, #VIRTAX, 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 'The axis of the virtual tool transformation, #VIRTAX, must be linear and must belong to the channel'

DETECTION During execution.

CAUSE The CNC has tried to activate virtual tool axis transformation (#VIRTAX) with a rotary axis or with an axis that does not belong to the channel.

SOLUTION Check the program.

2311 'The virtual axis of the tool cannot be a main axis or belong to 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.

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.

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.

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'

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 'The stop block has not been found in the block search'

DETECTION During execution.

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/G95'

DETECTION During execution.

CAUSE Functions G33 and G95 need a spindle to work. This spindle is by default the master spindle of the channel, but if PLC register SYNC indicates another spindle, 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 the SYNC register does not exist'

DETECTION During execution.

CAUSE A G33 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) is stopped or an M5 has been programmed in the same block as the G33.

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.

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 cancel it 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 The angular transformation is not necessary; cancel the transformation with the instruction #ANGAX OFF.

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 The angular transformation is not necessary. Cancel the angular transformation #ANGAX OFF.

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.

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 does not admit CLOOP'

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 in G33 must be homed'

DETECTION During execution.

CAUSE The spindle used in G33 threading has not been homed. If the CNC uses the master spindle of the channel, it always carries out a home search before the G33. If the PLC selects another spindle (SYNC mark), the CNC must home it.

SOLUTION Home the spindle.

3065 'The spindle speed programmed for G33 exceeds the threshold of the encoder'

DETECTION During execution.

CAUSE The speed programmed for the G33 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 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/G95: '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/G95'

DETECTION During execution.

CAUSE The spindle associated with function G33 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 are not admitted while a gear change process is active'

DETECTION During execution.

CAUSE The user has interrupted the program during a gear change. The CNC cannot change functions S, M or H.

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 S values for G33 exceed the maximum feedrate allowed'

DETECTION During execution.

CAUSE When starting a G33 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 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 with Motion Control'

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 is not admitted in this RETRACE process'

DETECTION During execution.

CAUSE In some programs with lots of very small motion blocks, the single-block mode cannot be used because it may saturate the resources of the system.

SOLUTION This retrace process does not allow the "single block" function. It is recommended to deactivate the "single block" function.

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 (OLOOP) for the master spindle, but it was working in closed loop for another previous synchronization and will stay in closed loop.

SOLUTION 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 an closed-loop synchronization (CLOOP) for the master spindle, but it was working in open loop for another previous synchronization and will change to closed loop.

SOLUTION In order for the spindle to stay in open loop, program also the second synchronization in open 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.

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.

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'

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.

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.

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 tries to control an axis 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 ON instruction has been programmed for a spindle that is in M19.

SOLUTION 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'

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'

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 out of limit'

DETECTION During execution.
CAUSE The axis has exceeded the admitted amount of following error set by machine parameters. 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 readings'

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'

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.

SOLUTION These axes or spindles can be parked neither from PLC nor from the CNC. Eliminate the signal for parking the axis or spindle from the PLC.

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 (Warning)'

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 'Some pulses got lost (not counted)'.

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/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, 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, 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.

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 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.

3736 '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.

SOLUTION Verify that the axis position is correct.

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.

SOLUTION Press reset to exit the jog mode. Functions G101 and G102 must be programmed for the master axis of the gantry axis.

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.

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.

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.

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.

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 181.

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.

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.

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.

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.

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 PLC digital inputs'

DETECTION	When reading the digital inputs of the PLC.
CAUSE	The digital I/O configuration table is wrong.
SOLUTION	Check the digital I/O configuration table. If the error persists, contact Fagor.

5014 'Error when writing the PLC digital outputs'

DETECTION	When writing the digital outputs of the PLC.
CAUSE	The table to set the digital I/O is wrong or the PLC has received a writing request before finishing the previous one.
SOLUTION	Check the table to set the digital I/O, runs the pertinent checks to ensure the integrity of the CAN bus or raise the cycle time in parameter PLCFREQ. If the error persists, contact Fagor.

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 When reading the local digital inputs (central units ICU and MCU).

CAUSE Reading is direct on central units ICU and MCU, so, this would be a formal error in the request to the driver.

SOLUTION Check the installation of the drivers of the CNC.

5024 'Error when writing local digital outputs'

DETECTION When writing the local digital outputs (central units ICU and MCU).

CAUSE Writing is direct on central units ICU and MCU, so, this would be a formal error in the request to the driver.

SOLUTION Check the installation of the drivers of the CNC.

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.

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.

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.

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.

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.

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 CAN configuration is not the same as the one saved in the file'

DETECTION	On system startup or during execution.
CAUSE	The CAN 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 or the CAN configuration has not detected all the nodes correctly.
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.

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.

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.

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 'Error in the PLC operation'

DETECTION During execution.
CAUSE Wrong sequence in the PLC program.
SOLUTION Check the PLC program.

7033 'The PLC generated an emergency in the tool magazine'

DETECTION	When the PLC is running.
CAUSE	The PLC 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.

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	When getting into MDI/MDA mode during inspection.
CAUSE	In the state where the program has been interrupted, it is not possible to execute blocks via MDI inside the inspection.

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.

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.

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.

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.

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[I, 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.

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.

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.

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.

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.

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.

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.

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.

8496 '$WHILE condition'

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

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.

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.

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.

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.

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.

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.

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".

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 ";".

9000-9999

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.

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 9, 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).

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.

23000-23999

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.

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.

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.

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

CNC 8060
CNC 8065

(REF: 1402)