CNC 8055 -M-

Error solution

Ref. 1310

FAGOR

FAGOR AUTOMATION

This product uses the following source code, subject to the terms of the GPL license. The applications *busybox* V0.60.2; *dosfstools* V2.9; *linux-ftpd* V0.17; *ppp* V2.4.0; *utelnet* V0.1.1. The library *grx* V2.4.4. The linux kernel V2.4.4. The linux boot *ppcboot* V1.1.3. If you would like to have a CD copy of this source code sent to you, send 10 Euros to Fagor Automation for shipping and handling.

INDEX

FAGOR

·M· Model

Ref. 1310

PROGRAMMING ERRORS

0001 'Empty line'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When trying to enter into a program or execute an empty block or containing the label (block number).
2. Within the «Irregular pocket canned cycle with islands (G66)», when parameter "S" (beginning of the profile) is greater than parameter "E" (end of profile).

SOLUTION The solution for each cause is:

1. The CNC cannot enter into the program or execute an empty line. To enter an empty line in the program, use the «;» symbol at the beginning of that block. The CNC will ignore the rest of the block.
2. The value of parameter "S" (block where the profile definition begins) must be lower than the value of parameter "E" (block where the profile definition ends).

0002 'Improper data'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When editing an axis coordinate after the cutting conditions (F, S, T or D) or the "M" functions.
2. When the marks of the block skip (conditional block /1, /2 or /3) are not at the beginning of the block.
3. When programming a block number greater than 99999999 while programming in ISO code.
4. When trying to define the coordinates of the machining starting point in the finishing operation (G68) of the "Irregular pocket canned cycle".
5. While programming in high-level, the value of the RPT instruction exceeds 9999.

SOLUTION The solution for each cause is:

1. Remember the programming order.
2. Remember the programming order.
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - "G" functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - "M" functions.
3. Correct the syntax of the block. Program the labels between 0 and 99999999.
4. No point can be programmed within the definition of the finishing cycle (G68) for the "Irregular pocket canned cycle". The CNC selects the point where it will start machining. The programming format is: G68 B...L...Q...I...R...K...V...

   And then the cutting conditions.
5. Correct the syntax of the block. Program a number of repetitions between 0 and 9999

0003 'Improper data order.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The machining conditions or the tool data have been programmed in the wrong order.

SOLUTION Remember that the programming order is:

… F...S...T...D.....

All the data need not be programmed.

0004 'No more information allowed in the block.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When editing a "G" function after an axis coordinate.
2. When trying to edit some data after a "G" function (or after its associated parameters) which must go alone in the block (or which only admits its own associated data).
3. When assigning a numeric value to a parameter that does not need it.

SOLUTION The solution for each cause is:

1. Remember the programming order.
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - "G" functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - "M" functions.
2. There are some "G" functions which carry associated data in the block. Maybe, this type of functions do not let program other type of information after their associated parameters. On the other hand, neither machining conditions, (F, S), tool data (T, D) nor "M" functions may be programmed.
3. There are some "G" functions having certain parameters associated to them which do not need to be defined with values.

0005 'Repeated information'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The same data has been entered twice in a block.

SOLUTION Correct the syntax of the block. The same data cannot be defined twice in a block.

0006 'Improper data format'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While defining the parameters of a machining canned cycle, a negative value has been assigned to a parameter which only admits positive values.

SOLUTION Verify the format of the canned cycle. In some canned cycles, there are parameters which only accept positive values.

0007 'Incompatible G functions.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When programming in the same block two "G" functions which are incompatible with each other.
2. When trying to define a canned cycle in a block containing a nonlinear movement (G02, G03, G08, G09, G33).

SOLUTION The solution for each cause is:

1. There are groups of "G" functions which cannot go together in the block because they involve actions incompatible with each other. For example:

   G01/G02: Linear and circular interpolation

   G41/G42: Left-hand or right-hand tool radius compensation.

   This type of functions must be programmed in different blocks.
2. A canned cycle must be defined in a block containing a linear movement. In other words, to define a cycle, a "G00" or a "G01" must be active. Nonlinear movements (G02, G03, G08 and G09) may be defined in the blocks following the profile definition.

0008 'Nonexistent G function'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A nonexistent "G" function has been programmed.

SOLUTION Check the syntax of the block and verify that a different "G" function is not being edited by mistake.

0009 'No more G functions allowed'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A "G" function has been programmed after the machining conditions or after the tool data.

SOLUTION Remember that the programming order is:

- Block skip (conditional block /1, /2 or /3).
- Label (N).
- "G" functions.
- Axis coordinates. (X, Y, Z…).
- Machining conditions (F, S, T, D).
- "M" functions.

0010 'No more M functions allowed'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE More than 7 "M" functions have been programmed in a block.

SOLUTION The CNC does not let program more than 7 "M" functions in a block. To execute any other functions, write them in a separate block. The "M" functions may go alone in a block.

0011 'This G or M function must be alone.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The block contains either a "G" or an "M" function that must go alone in the block.

SOLUTION Write it alone in the block.

0012 'Program F, S, T, D before the M functions.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A machining condition (F, S) or tool data (T, D) has been programmed after the "M" functions.

SOLUTION Remember that the programming order is:

… F...S...T...D...M...

Up to 7 "M" functions may be programmed .

All the data need not be programmed.

0013 'Program G30 D +/-359.9999'

No explanation required.

0014 'Do not program labels by parameters.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A label (block number) has been defined with a parameter.

SOLUTION Programming the block number is optional, but it cannot be defined with a parameter It can only be defined with a number between 0 and 99999999.

0015 'Number of repetitions not possible.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A repetition has been programmed wrong or the block does not admit repetitions.

SOLUTION High level instructions do not admit a number of repetitions at the end of the block. To do a repetition, assign to the block to be repeated a label (block number) and use the RPT instruction.

0016 'Program: G15 axis.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function "Longitudinal axis selection (G15)" the parameter for the axis has not been programmed.

SOLUTION Check the syntax of the block. The definition of the "G15" function requires the name of the new longitudinal axis.

0017 'Program: G16 axis-axis.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function "Main plane selection by two axes (G16)" one of the two parameters for the axes has not been programmed.

SOLUTION Check the syntax of the block. The definition of the "G16" function requires the name of the axes defining the new work plane.

0018 'Program: G22 K(1/2/3/4/5) S(0/1/2).'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function "Enable/Disable work zones (G22)" the type of enable or disable of the work zone has not been defined or it has been assigned the wrong value.

SOLUTION The parameter for enabling or disabling the work zones "S" must always be programmed and it may take the following values.

- S=0: The work zone is disabled.
- S=1: It is enabled as a no-entry zone.
- S=2: It is enabled as a no-exit zone.

0019 'Program zone K1, K2, K3, K4 or K5.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. A "G20", "G21" or "G22" function has been programmed without defining the work zone K1, K2, K3, K4 or K5
2. The programmed work zone is smaller than 0 or greater than 5.

SOLUTION The solution for each cause is:

1. The programming format for functions "G20", "G21" and "G22" is:

   G20 K...X...C±5.5 Definition of lower work zone limits.
   G21 K...X...C±5.5 Definition of upper work zone limits.
   G22 K...S... Enable/disable work zones.

   Where:

   K Is the work zone.
   X...C Are the axes where the limits are defined.
   S Is the type of work zone enable.

2. The "K" work zone may only have the values of K1, K2, K3, K4 or K5.

0020 'Program G36-G39 with R+5.5.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "G36" or "G39" function, the "R" parameter has not been programmed or it has been assigned a negative value.

SOLUTION To define "G36" or "G39", parameter "R" must also be defined and with a positive value).

G36 R= Rounding radius.
G39 R= Distance between the end of the programmed path and the point to be chamfered.

0021 'Program: G72 S5.5 or axis (axes).'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When programming a general scaling factor (G72) without the scaling factor to apply.
2. When programming a particular scaling factor (G72) to several axes, but the axes have been defined in the wrong order.

SOLUTION Remember that the programming format for this function is:

G72 S5.5" When applying a general scaling factor (to all axes).
G72 X…C5.5" When applying a particular scaling factor to one or several axes.

0022 'Program: G73 Q (angle) I J (center).'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "Pattern rotation (G73)" function have been programmed wrong. The causes may be:

1. The rotation angle has not been defined.
2. Only one of the rotation center coordinates has been defined.
3. The rotation center coordinates have been defined in the wrong order.

SOLUTION The programming format for this function is:

G73 Q (angle) [I J] (center)

The "Q" value must always be programmed.

The "I", "J" values are optional, but if programmed, both must be programmed.

0023 'Block incompatible when defining a profile.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the set of blocks defining a pocket profile, there is a block containing a "G" function that cannot be part of the profile definition.

SOLUTION The "G" functions available in the profile definition of a pocket (2D/3D) are:

G00: Beginning of the profile.
G01: Linear interpolation.
G02/G03: Clockwise/counterclockwise interpolation.
G06: Circle center in absolute coordinates.
G08: Arc tangent to previous path.
G09: Three point arc.
G36: Automatic radius blend.
G39: Chamfer.
G53: Programming with respect to home.
G70/G71: Inch/metric programming.
G90/G91: Programming in absolute/incremental coordinates.
G93: Polar origin preset.

And also, in the 3D pocket profile:

G16: Main plane selection by two axes.
G17: Main plane X-Y and longitudinal Z.
G18: Main plane Z-X and longitudinal Y.
G19: Main plane Y-Z and longitudinal X.

0024 'High level blocks not allowed when defining a profile.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Within the set of blocks defining a pocket profile, a high level block has been programmed.

SOLUTION The pocket profile must be defined in ISO code. High level instructions are not allowed (GOTO, MSG, RPT ...).

0025 'Program: G77 axes (2 to 6) or G77 S.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "axis slaving function (G77)" the parameters for the axes are missing or in "spindle synchronization (G77S) functions the "S" parameter is missing.

SOLUTION In the "axis slaving" function, program at least two axes and in the "spindle synchronization" function, always program the "S" parameter.

0026 'Program: G93 I J.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "Polar origin preset (G93)" function, some of the parameters for the new polar origin have not been programmed.

SOLUTION Remember that the programming format for this function is:

G93 I...J...

The "I", "J" values are optional, but if programmed, both must be programmed and they indicate the new polar origin.

0027 'G49 T X Y Z S, X Y Z A B C or X Y Z Q R S.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "Inclined plane definition (G49)" function, a parameter has been programmed twice.

SOLUTION Check the syntax of the block. The programming formats are:

T X Y Z S X Y Z A B C X Y Z Q R S

0028 'G2 or G3 not allowed when programming a canned cycle.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A canned cycle has been attempted to execute while the "G02", "G03" or "G33" functions were active.

SOLUTION To execute a canned cycle, "G00" or "G01" must be active. A "G02" or "G03" function may be programmed previously in the program history. Check that these functions are not active when the canned cycle is defined.

0029 'G60: [A] /X I K/(2) [P Q R S T U V].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "Multiple machining in a straight line (G60)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

SOLUTION In this type of machining, two of the following parameters must always be programmed:

- X Path length.
- I Step between machining operations.
- K Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0030 'G61-2: [A B] /X I K/(2) Y J D (2)/ [P Q R S T U V].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "Multiple machining in a parallelogram pattern (G61)" or "Multiple machining in a grid pattern (G62)" cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

SOLUTION This type of machining requires the programming of two parameters of each group (X, I, K) and (Y, J, D).

- X/Y Path length.
- I/J Step between machining operations.
- K/D Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0031 'G63: X Y /I K/(1) [C P][P Q R S T U V].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "Multiple machining in a circle (G63)" cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

SOLUTION This type of machining requires the programming of:

- X/Y Distance from the center to the first hole.

And one of the following data:

- I Angular step between machining operations.
- K Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0032 'G64: X Y B /I K/(1) [C P][P Q R S T U V].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "multiple machining in an arc (G64)" cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

SOLUTION This type of machining requires the programming of:

- X/Y Distance from the center to the first hole.
- B Total angular travel.

And one of the following data:

- I Angular step between machining operations.
- K Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0033 'G65: X Y /A I/(1) [C P].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "Multiple machining programmed by means of an arc chord (G65)" cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

SOLUTION This type of machining requires the programming of:

- X/Y Distance from the center to the first hole.

And one of the following data:

- A Angle of the matrix of the chord with the abscissa axis (in degrees).
- I Length of the chord.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0034 'G66: [D H][R I][C J][F K] S E [Q].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "Irregular pocket canned cycle with islands (G66)" have been programmed wrong. These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

SOLUTION This machining cycle requires the programming of :

- S First block of the description of the geometry of the profiles making up the pocket.
- E End block of the description of the geometry of the profiles making up the pocket.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. Also, the following parameters cannot be defined:

- H if D has not been defined.
- I if R has not been defined.
- J if C has not been defined.
- K if F has not been defined.

The (X...C) position where the machining takes place cannot be programmed either.

0035 'G67: [A] B [C] [I] [R] [K] [V] [Q].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the roughing (2D/3D pocket) or semi-finishing (3D pocket) operation have been programmed wrong in the "Irregular pocket canned cycle with islands". These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

SOLUTION This machining cycle requires the programming of :

Roughing operation (2D or 3D pockets)

- B Depth of pass.
- I Total pocket depth.
- R Coordinate of the reference plane.

Semi-finishing operation (3D pockets)

- B Depth of pass.
- I Total pocket depth (if no roughing operation has been defined).
- R Coordinate of the reference plane (if no roughing operation has been defined).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0036 'G68: [B] [L] [Q] [J] [I] [R] [K].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters for the finishing operation (2D/3D pocket) have been programmed wrong in the "Irregular pocket cycle with islands. These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

SOLUTION This machining cycle requires the programming of :

2D pockets

- B Cutting pass (if no roughing operation has been defined).
- I Total pocket depth (if no roughing operation has been defined).
- R Coordinate of the reference plane (if no roughing operation has been defined).

3D pockets

- B Depth of pass.
- I Total pocket depth (if no roughing or semi-finishing operation has been defined).
- R Coordinate of the reference plane (if no roughing or semi-finishing operation has been defined).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0037 'G69: I B [C D H J K L R].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the "Deep hole drilling cycle with variable peck (G69)". These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

SOLUTION This type of machining requires the programming of:

- I Machining depth.
- B Drilling peck (step).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0038 'G81-84-85-86-89: I [K].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters have been programmed wrong in the following cycles: drilling (G81), tapping (G84), reaming (G85) or boring (G86/G89). This could be because parameter "I : Machining depth" is missing in the canned cycle being edited.

SOLUTION This type of machining requires the programming of:

- I Machining depth.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0039 'G82: I K.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters have been programmed wrong in the "Drilling cycle with dwell (G82)". This could be because some parameter is missing.

SOLUTION Both parameters must be programmed in this cycle:

- I Machining depth.
- K Dwell at the bottom.

To program a drilling operation without dwell at the bottom, use function G81.

The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0040 'G83: I J.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters have been programmed wrong in the "Deep hole drilling with constant peck (G83)". This could be because some parameter is missing.

SOLUTION This type of machining requires the programming of:

- I Machining depth.
- J Number of pecks.

The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0041 'G87: I J K B [C] [D] [H] [L] [V].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters have been programmed wrong in the "Rectangular pocket canned cycle (G87)". These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

SOLUTION This type of machining requires the programming of:

- I Pocket depth.
- J Distance from the center to the edge of the pocket along the abscissa axis.
- K Distance from the center to the edge of the pocket along the ordinate axis.
- B Defines the cutting depth according to the longitudinal axis.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0042 'G88: I J B [C] [D] [H] [L] [V].'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters have been programmed wrong in the "Circular pocket canned cycle (G88)". These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

SOLUTION This type of machining requires the programming of:

- I Pocket depth.
- J Pocket radius.
- B Defines the cutting depth according to the longitudinal axis.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0043 'Incomplete Coordinates.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. During simulation or execution, when trying to make a movement defined with only one coordinate of the end point or without defining the arc radius while a "circular interpolation (G02/G03) is active.
2. During editing, when editing a circular movement (G02/G03) by defining only one coordinate of the end point or not defining the arc radius.

SOLUTION The solution for each cause is:

1. A "G02" or "G03" function may be programmed previously in the program history. In this case, to make a move, both coordinates of the end point and the arc radius must be defined. To make a linear movement, program "G01".
2. To make a circular movement (G02/G03), both coordinates of the end point and the arc radius must be programmed.

0044 'Incorrect Coordinates.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. An attempt has been made to execute a block syntactically incorrect (G1 X20K-15)
2. The "I" parameter is missing in the definition of a machining canned cycle (G81-G89) Machining depth.

SOLUTION The solution for each cause is:

1. Correct the syntax of the block.
2. This type of machining requires the programming of:

   I Machining depth.

   The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0045 'Polar coordinates not allowed.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE When "Programming with respect to home (G53)", the end point has been defined in polar or cylindrical coordinates or in Cartesian coordinates with an angle.

SOLUTION When programming with respect to home, only Cartesian coordinates may be programmed.

0046 'Axis does not exist.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When editing a block whose execution involves the movement of a nonexistent axis.
2. Sometimes, this error comes up while editing a block that is missing a parameter of the "G" function. This is because some parameters with an axis name have a special meaning inside certain "G" functions. For example: G69 I...B....

   In this case, parameter "B" has a special meaning after "I". If the "I" parameter is left out, the CNC assumes "B" as the position where the machining takes place on that axis. If that axis does not exist, it will issue this error message.

SOLUTION The solution for each cause is:

1. Check that the axis name being edited is correct.
2. Check the block syntax and make sure that all the mandatory parameters have been programmed.

0047 'Program axes.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE No axis has been programmed in a function requiring an axis.

SOLUTION Some instructions require the programming of axes (REPOS, G14, G20, G21...).

0048 'Incorrect order of axes.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The axis coordinates have not been programmed in the correct order or an axis has been programmed twice in the same block.

SOLUTION Remember that the correct programming order for the axes is:

X...Y...Z...U...V...W...A...B...C...

All axes need not be programmed:

0049 'Point incompatible with active plane.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When trying to do a circular interpolation, the end point is not in the active plane.
2. When trying to do a tangential exit in a path that is not in the active plane.

SOLUTION The solution for each cause is:

1. Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, circular interpolations can only be carried out on the main axes defining that plane. To define a circular interpolation in another plane, it must be defined beforehand.
2. Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.

0050 'Program positions on active plane.'

No explanation required.

0051 'Perpendicular axis included in active plane.'

No explanation required.

0052 'Center of circle programmed incorrectly.'

No explanation required.

0053 'Program pitch.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "Electronic threading cycle (G33)" the parameter for the thread pitch is missing.

SOLUTION Remember that the programming format for this function is:

G33 X...C...L...

Where: "L" is the thread pitch.

0054 'Pitch programmed incorrectly.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A helical interpolation has been programmed with the wrong or negative pitch.

SOLUTION Remember that the programming format is:

G02/G03 X...Y...I...J...Z...K...

Where: "K" is the helical pitch (always positive value).

0055 'Positioning axes or Hirth axes not allowed'

No explanation required.

0056 'The axis is already slaved.'

No explanation required.

0057 'Do not program a slaved axis.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When trying to move an axis alone while being slaved to another one.
2. When trying to slave an axis that is already slaved using the G77 function "Electronic axis slaving".

SOLUTION The solution for each cause is:

1. A slaved axis cannot be moved separately. To move a slaved axis, its master axis must be moved. Both axes will move at the same time.

   Example: If the Y axis is slaved to the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).

   To unslave the axes, program "G78".
2. An axis cannot be slaved to two different axes at the same time. To unslave the axes, program "G78".

0058 'Do not program a GANTRY axis.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. When trying to move an axis alone while being slaved to another one as a GANTRY axis
2. When defining an operation on a GANTRY axis. (Definition of work zone limits, planes, etc.).

SOLUTION The solution for each cause is:

1. A GANTRY axis cannot be moved separately. To move a GANTRY axis, its associated axis must be moved. Both axes will move at the same time.

   Example: If the Y axis is a GANTRY axis associated with the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).

   GANTRY axes are defined by machine parameter.
2. The axes defined as GANTRY cannot be used in the definition of operations or movements. These operations are defined with the main axis that the GANTRY axis is associated with.

0059 'Wrong position programmed for the Hirth axis.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A rotation of a HIRTH axis has been programmed with a decimal value.

SOLUTION HIRTH axes do not accept decimal angular values. They must be full degrees.

0060 'Invalid action.'

No explanation required.

0061 'ELSE not associated with IF.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. While editing in High level language, when editing the "ELSE" instruction without having previously programmed an "IF".
2. When programming in high level language, an "IF" is programmed without associating it with any action after the condition.

SOLUTION Remember that the programming formats for this instruction are:

(IF (condition) <action1>)

(IF (condition <action1> ELSE <action2>)

If the condition is true, it executes the <action1>, otherwise, it executes <action2>.

0062 'Program label N(0-99999999).'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a block number out of the 0-99999999 range has been programmed in the "RPT" or "GOTO" instruction.

SOLUTION Remember that the programming format of these instructions is:

(RPT N(block number), N(block number))

(GOTO N(block number))

The block number (label) must be between 0 and 99999999.

0063 'Program subroutine number 1 thru 9999.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a subroutine number out of the 0-9999 range has been programmed in the "SUB" instruction.

SOLUTION Remember that the programming format for this instruction is:

(SUB (integer))

The subroutine number must be between 0 and 9999.

0064 'Repeated subroutine.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE There has been an attempt to define a subroutine already existing in another program of the memory.

SOLUTION In the CNC memory, there could not be more than one subroutine with the same identifying number even if they are contained in different programs.

0065 'The main program cannot have a subroutine.'

DETECTION In execution or while executing programs transmitted via DNC.

CAUSE The possible causes are:

1. An attempt has been made to define a subroutine in the MDI execution mode.
2. A subroutine has been defined in the main program.

SOLUTION The solution for each cause is:

1. Subroutines cannot be defined from the "MDI execution" option of the menu.
2. Subroutines must be defined after the main program or in a separate program. They cannot be defined before or inside the main program.

0066 'Expecting a message.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level, the "MSG" or "ERROR" instruction has been edited but without the message to be displayed.

SOLUTION Remember that the programming format of these instructions is:

(MSG "message")

(ERROR integer, "error message")

Although it can also be programmed as follows:

(ERROR integer)

(ERROR "error message")

0067 'OPEN is missing.'

DETECTION	In execution or while executing programs transmitted via DNC.
CAUSE	While programming in high level, a "WRITE" instruction has been edited, but the OPEN instruction has not been written previously to tell it where that instruction has to be executed.
SOLUTION	The "OPEN" instruction must be edited before the "WRITE" instruction to "tell" the CNC where (in which program) it must execute the "WRITE" instruction.

0068 'Expecting a program number.'

No explanation required.

0069 'Program does not exist.'

DETECTION	In execution or while executing programs transmitted via DNC.
CAUSE	Inside the "Irregular pocket with islands cycle (G66)", it has been programmed that the profiles defining the irregular pocket are in another program (parameter "Q"), but that program does not exist.
SOLUTION	Parameter "Q" defines which program contains the definition of the profiles that, in turn, define the irregular pocket with islands. If this parameter is programmed, that program number must exist and it must contain the labels defined by parameters "S" and "E".

0070 'Program already exists.'

DETECTION	In execution or while executing programs transmitted via DNC.
CAUSE	This error comes up during execution when using the "OPEN" instruction (While programming in high level language) to create an already existing program.
SOLUTION	Change the program number or use parameters A/D in the "OPEN" instruction: (OPEN P.....,A/D,…) Where: A: Appends new blocks after the existing ones. D: Deletes the existing program and it opens it as a new one.

0071 'Expecting a parameter'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The possible causes are: 1. When defining the function "Modification of canned cycle parameters (G79)", the parameter to be modified has not been indicated. 2. While editing the machine parameter table, the wrong parameter number has been entered (maybe the "P" character is missing) or another action is being carried out (moving around in the table) before quitting the table editing mode.
SOLUTION	The solution for each cause is: 1. To define the "G79" function, the cycle parameter to be modified must be indicated as well as its new value. 2. Enter the parameter number to be edited or press [ESC] to quit this mode.

0072 'Parameter does not exist.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, the "ERROR" instruction has been edited, but the error number to be displayed has been defined either with a local parameter greater than 25 or with a global parameter greater than 299.
SOLUTION	The parameters used by the CNC are: Local: 0-25 Global: 100-299

0073 'Range of write-protected parameters.

No explanation required.

0074 'Variable not accessible from CNC.'

No explanation required.

0075 'Read-only variable.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An attempt has been made to assign a value to a read-only variable.
SOLUTION	Read-only variables cannot be assigned any values through programming. However, their values can be assigned to a parameter.

0076 'Write-only variable.'

No explanation required.

0077 'Analog output not available.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An attempt has been made to write to an analog output currently being used by the CNC.
SOLUTION	The selected analog output may be currently used by an axis or a spindle. Select another analog output between 1 and 8.

0078 'Program channel 0(CNC),1(PLC) or 2(DNC).'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, the "KEYSCR" instruction has been programmed, but the source of the keys is missing.
SOLUTION	When programming the "KEYSCR" instruction, the parameter for the source of the keys must always be programmed: (KEYSCR=0) : CNC keyboard (KEYSCR=1) : PLC (KEYSCR=2) : DNC The CNC only lets modifying the contents of this variable if it is "zero"

0079 'Program error number 0 thru 9999.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, the "ERROR" instruction has been programmed, but the error number to be displayed is missing.
SOLUTION	Remember that the programming format for this instruction is: (ERROR integer, "error message") Although it can also be programmed as follows: (ERROR integer) (ERROR "error message")

0080 'Operator missing.'

No explanation required.

0081 'Incorrect expression.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, an expression has been edited with the wrong format.
SOLUTION	Correct the syntax of the block.

0082 'Incorrect operation.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The possible causes are: 1. While programming in high level language, the assignment of a value to a parameter is incomplete. 2. While programming in high level language, the call to a subroutine is incomplete.
SOLUTION	Correct (complete) the format to assign a value to a parameter or a call to a subroutine.

0083 'Incomplete operation.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The various causes might be:

1. While programming in high level language, the "IF" instruction has been edited without the condition between brackets.
2. While programming in high level language, the "DIGIT" instruction has been edited without assigning a value to some parameter.

SOLUTION The solution for each cause is:

1. Remember that the programming formats for this instruction are:

   (IF (condition) <action1>)

   (IF (condition <action1> ELSE <action2>)

   If the condition is true, it executes the <action1>, otherwise, it executes <action2>.
2. Correct the syntax of the block. All the parameters defined within the "DIGIT" instruction must have a value assigned to them.

0084 'Expecting "=".'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the "=" symbol in the right place.

0085 'Expecting ")".'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the ")" symbol in the right place.

0086 'Expecting "(".'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the "(" symbol in the right place.

0087 'Expecting ",".'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The possible causes are:

1. While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.
2. While programming in high level language, an ISO-coded instruction has been programmed.
3. While programming in high level language, an operation has been assigned either to a local parameter greater than 25 or to a global parameter greater 299.

SOLUTION The solution for each cause is:

1. Enter the "," symbol in the right place.
2. A block cannot contain high level language instructions and ISO-coded instructions at the same time.
3. The parameters used by the CNC are:

   Local: 0-25.

   Global: 100-299.

   Other parameters out of this range cannot be used in operations.

0088 'Operation limit exceeded.'

No explanation required.

0089 'Logarithm of zero or negative number.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves the calculation of a negative number or a zero.

SOLUTION Only logarithms of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Verify that the parameter does not reach the operation with that value (0).

0090 'Square root of a negative number.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An operation has been programmed which involves the calculation of the square root of a negative number.
SOLUTION	Only the square root of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Verify that the parameter does not reach the operation with that value (0).

0091 'Division by zero.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An operation has been programmed whose execution involves dividing by zero.
SOLUTION	It is only possible to divide by numbers other than zero. When working with parameters, that parameter may have already acquired a negative value or zero. Verify that the parameter does not reach the operation with that value (0).

0092 'Base zero with positive exponent.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An operation has been programmed which involves elevating zero to a negative exponent (or zero).
SOLUTION	Zero can only be elevated to positive exponents greater than zero. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0093 'Negative base with decimal exponent.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An operation has been programmed which involves elevating a negative number to a decimal exponent.
SOLUTION	Negative numbers can only be elevated to integer exponents. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0094 'ASIN/ACOS range exceeded.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An operation has been programmed which involves calculating the arcsine or arccosine of a number out of the ±1 range.
SOLUTION	Only the arc sine (ASIN) or the arc cosine (ACOS) of numbers between ±1 can be calculated. When working with parameters, that parameter may have already acquired a value out of the mentioned values. Verify that the parameter does not reach the operation with that value (0).

0095 'Program row number.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While editing a customizing program, a window has been programmed with the "ODW" instruction, but the vertical position of the window on the screen is missing.
SOLUTION	The vertical position of the window on the screen is defined by rows (0-25).

0096 'Program column number.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While editing a customizing program, a window has been programmed with the "ODW" instruction, but the horizontal position of the window on the screen is missing.
SOLUTION	The horizontal position of the window on the screen is defined by columns (0-79).

0097 'Program another softkey.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While editing a customizing program, the programming format for the "SK" instruction has not been respected.
SOLUTION	Correct the syntax of the block. The programming format is: (SK1=(text 1), SK2=(text 2)...) If the "," character is entered after a text, the CNC expects the name of another softkey.

0098 'Program softkeys 1 thru 7.'

DETECTION	While executing in the user channel.
CAUSE	In the block syntax, a softkey has been programmed out of the 1 to 7 range.
SOLUTION	Only softkeys within the 1 to 7 range can be programmed.

0099 'Program another window.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While editing a customizing program, the programming format for the "DW" instruction has not been respected.
SOLUTION	Correct the syntax of the block. The programming format is: (DW1=(assignment), DW2=(assignment)...) If the "," character is entered after an assignment, the CNC expects the name of another window.

0100 'Program windows 0 thru 25.'

DETECTION	While executing in the user channel.
CAUSE	In the block syntax, a window has been programmed out of the 0 to 25 range.
SOLUTION	Only windows within the 0 to 25 range can be programmed.

0101 'Program rows 0 thru 20.'

DETECTION	While executing in the user channel.
CAUSE	In the block syntax, a row has been programmed out of the 0 to 20 range.
SOLUTION	Only rows within the 0 to 20 range can be programmed.

0102 'Program columns 0 thru 79.'

DETECTION	While executing in the user channel.
CAUSE	In the block syntax, a column has been programmed out of the 0 to 79 range.
SOLUTION	Only columns within the 0 to 79 range can be programmed.

0103 'Program pages 0 thru 255.'

DETECTION	While executing in the user channel.
CAUSE	In the block syntax, a page has been programmed out of the 0 to 255 range.
SOLUTION	Only pages within the 0 to 255 range can be programmed.

0104 'Program INPUT.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, an "IB" instruction has been edited without associating an "INPUT" to it.
SOLUTION	Remember that the programming formats for this instruction are: (IB (expression) = INPUT "text", format) (IB (expression) = INPUT "text")

0105 'Program inputs 0 thru 25.'

DETECTION	While executing in the user channel.
CAUSE	In the block syntax, an input has been programmed out of the 0 to 25 range.
SOLUTION	Only inputs within the 0 to 25 range can be programmed.

0106 'Program numerical format.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, an "IB" instruction has been edited with non-numeric format.
SOLUTION	Remember that the programming format for this instruction is: (IB (expression) = INPUT "text", format) Where "format" must be a signed number with 6 entire digits and 5 decimals at the most. If the "," character is entered after the text, the CNC expects the format.

0107 'Do not program formats greater than 6.5 .'

DETECTION While executing in the user channel.

CAUSE While programming in high level language, an "IB" instruction has been edited in a format with more than 6 entire digits or more than 5 decimals.

SOLUTION Remember that the programming format for this instruction is:

(IB (expression) = INPUT "text", format)

Where "format" must be a signed number with 6 entire digits and 5 decimals at the most.

0108 'This command can only be executed in the user channel.'

DETECTION During execution.

CAUSE An attempt has been made to execute a block containing information that can only be executed through the user channel.

SOLUTION There are specific expressions for customizing programs that can only be executed inside the user program.

0109 'C. user: do not program geometric help, compensation or cycles.'

DETECTION While executing in the user channel.

CAUSE An attempt has been made to execute a block containing geometric aide, tool radius/length compensation or machining canned cycles.

SOLUTION Inside a customizing program the following cannot be programmed:

Neither geometric assistance nor movements.

Neither tool radius nor length compensation.

Canned cycles.

0110 'Local parameters not allowed.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Some functions can only be programmed with global parameters.

SOLUTION Global parameters are the ones included in the 100-299 range.

0111 'Block cannot be executed while running another program'

DETECTION While executing in MDI mode

CAUSE An attempt has been made to execute a customizing instruction from MDI mode while the user channel program is running.

SOLUTION Customizing instructions can only be executed through the user channel.

0112 'WBUF can only be executed in user channel while editing'

DETECTION During normal execution or execution through the user channel.

CAUSE An attempt has been made to execute the "WBUF" instruction.

SOLUTION The "WBUF" instruction cannot be executed. It can only be used in the editing stage through the user input.

0113 'Table limits exceeded.'

DETECTION While editing tables.

CAUSE The possible causes are:

1. In the tool offset table, an attempt has been made to define a tool offset with a greater number than allowed by the manufacturer.
2. In the parameter tables, an attempt has been made to define a nonexistent parameter.

SOLUTION The tool offset number must be smaller than the one allowed by the manufacturer.

0114 'Offset: D3 R L I K.'

DETECTION While editing tables.

CAUSE In the tool offset table, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order.

0115 'Tool: T4 D3 F3 N5 R5(.2).'

DETECTION While editing tables.

CAUSE In the tool table, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order.

0116 'Origin: G54-59 G159N(1-20) axes (1-7).'

DETECTION	While editing tables.
CAUSE	In the Zero offset table, the zero offset to be defined (G54-G59) or G159N(1-20) has not be selected.
SOLUTION	Enter the table parameters in the right order. To fill out the zero offset table, first select the offset to be defined (G54-G59) or G159N(1-20) and then the zero offset position for each axis.

0117 'M function: M4 S4 bits (8).'

DETECTION	While editing tables.
CAUSE	In the "M" function table, the parameter editing order has not been respected.
SOLUTION	Edit table following the format: M1234 (associated subroutine) (customizing bits)

0118 'G51 [A] E'

DETECTION	In execution or while executing programs transmitted via DNC.
CAUSE	In the "Look-Ahead (G51)" function, the parameter for the maximum contouring error is missing.
SOLUTION	This type of machining requires the programming of: E: Maximum contouring error. The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0119 'Leadscrew: Coordinate-error.'

DETECTION	While editing tables.
CAUSE	In the leadscrew compensation tables, the parameter editing order has not been respected.
SOLUTION	Enter the table parameters in the right order. P123 (position of the axis to be compensated) (leadscrew error at that point)

0120 'Incorrect axis.'

DETECTION	While editing tables.
CAUSE	In the leadscrew compensation tables, an attempt has been made to edit a different axis from the one corresponding to that table.
SOLUTION	Each axis has its own table for leadscrew compensation. The table for each axis can only contain the positions for that axis.

0121 'Program P3 = value.'

DETECTION	While editing tables.
CAUSE	In the machine parameter table, the editing format has not been respected.
SOLUTION	Enter the table parameters in the right order. P123 = (parameter value)

0122 'Tool magazine: Magazine:

DETECTION	While editing tables.
CAUSE	In the tool magazine table, the editing format has not been respected or some data is missing.
SOLUTION	Enter the table parameters in the right order.

0123 'Tool T0 does not exist.'

DETECTION	While editing tables.
CAUSE	In the tool table, an attempt has been made to edit a tool as T0.
SOLUTION	No tool can be edited as T0. The first tool must be T1.

0124 'Offset D0 does not exist.'

DETECTION	While editing tables.
CAUSE	In the tool table, an attempt has been made to edit a tool offset as D0.
SOLUTION	No tool offset can be edited as D0. The first tool offset must be D1.

0125 'Do not modify the active tool or the next one.'

DETECTION	During execution.
CAUSE	In the tool magazine table, an attempt has been made to change the active tool or the next one.
SOLUTION	During execution, neither the active tool nor the next one may be changed.

0126 'Tool not defined.'

DETECTION	While editing tables.
CAUSE	In the tool magazine table, an attempt has been made to assign to the magazine position a tool that is not defined in the tool table.
SOLUTION	Define the tool in the tool table.

0127 'Magazine is not RANDOM.'

DETECTION	While editing tables.
CAUSE	There is no RANDOM magazine and, in the tool magazine table, the tool number does not match the tool magazine position.
SOLUTION	When the tool magazine is not RANDOM, the tool number must be the same as the magazine position (pocket number).

0128 'The position of a special tool is set.'

DETECTION	While editing tables.
CAUSE	In the tool magazine table, an attempt has been made to place a tool in a magazine position reserved for a special tool.
SOLUTION	When a special tool occupies more than one position in the magazine, it has a reserved position in the magazine. No other tool can be placed in this position.

0129 'Next tool only possible in machining centers.'

DETECTION	During execution.
CAUSE	A tool change has been programmed with M06, but the machine is not a machining center. (it is not expecting the next tool).
SOLUTION	When the machining is not a machining center, the tool change is done automatically when programming the tool number "T".

0130 'Write 0/1.'

DETECTION	While editing machine parameters
CAUSE	An attempt has been made to assign the wrong value to a parameter.
SOLUTION	The parameter only admits values of 0 or 1.

0131 'Write +/-.'

DETECTION	While editing machine parameters
CAUSE	An attempt has been made to assign the wrong value to a parameter.
SOLUTION	The parameter only admits values of + or -.

0132 'Write YES/NO.'

DETECTION	While editing machine parameters
CAUSE	An attempt has been made to assign the wrong value to a parameter.
SOLUTION	The parameter only admits values of YES or NO.

0133 'Write ON/OFF.'

DETECTION	While editing machine parameters
CAUSE	An attempt has been made to assign the wrong value to a parameter.
SOLUTION	The parameter only admits values of ON or OFF.

0134 'Values 0 thru 2.'
0135 'Values 0 thru 3.'
0136 'Values 0 thru 4.'
0137 'Values 0 thru 9.'
0138 'Values 0 thru 29.'
0139 'Values 0 thru 100.'
0140 'Values 0 thru 255.'
0141 'Values 0 thru 9999.'
0142 'Values 0 thru 32767.'
0143 'Values within +/-32767.'
0144 'Values 0 thru 65535.'

DETECTION	While editing machine parameters
CAUSE	The possible causes are: 1. An attempt has been made to assign the wrong value to a parameter. 2. During execution, when inside the program a call has been made to a subroutine (MCALL, PCALL) with a value greater than allowed.

0145 'Format +/- 5.5.'

DETECTION	While editing machine parameters
CAUSE	An attempt has been made to assign the wrong value to a parameter.
SOLUTION	The parameter only admits values with the format:

0146 'Word does not exist.'

No explanation required.

0147 'Numerical format exceeded.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	A data or parameter has been assigned a value greater than the established format.
SOLUTION	Correct the syntax of the block. Most of the time, the numeric format will be 5.4 (5 integers and 4 decimals).

0148 'Text too long.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, the "ERROR" or "MSG" instruction has been assigned a text with more than 59 characters.
SOLUTION	Correct the syntax of the block. The "ERROR" and "MSG" instructions cannot be assigned texts longer than 59 characters.

0149 'Incorrect message.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, the text associated with the "ERROR" or "MSG" instruction has been edited wrong.
SOLUTION	Correct the syntax of the block. The programming format is: (MSG "message") (ERROR number, "message") The message must be between " ".

0150 'Incorrect number of bits.'

DETECTION	While editing tables.
CAUSE	The possible causes are: 1. In the "M" function table, in the section on customizing bits: The number does not have 8 bits. The number does not consist of 0's and 1's. 2. In the machine parameter table, an attempt has been made to assign the wrong value of bit to a parameter.
SOLUTION	The solution for each cause is: 1. The customizing bits must consist of 8 digits of 0's and 1's. 2. The parameter only admits 8-bit or 16-bit numbers.

0151 'Negative numbers not allowed.'

No explanation required.

0152 'Incorrect parametric programming.'

DETECTION	During execution.
CAUSE	The parameter has a value that is incompatible with the function it has been assigned to.
SOLUTION	This parameter may have taken the wrong value, in the program history. Correct the program so this parameter does not reach the function with that value.

0153 'Decimal format not allowed.'

No explanation required.

0154 'Insufficient memory.'

DETECTION	During execution.
CAUSE	The CNC does not have enough memory to internally calculate the paths.
SOLUTION	Sometimes, this error is taken care of by changing the machining conditions.

0155 'Help not available.'

No explanation required.

0156 'Don't program G33 ,G95 or M19 S with no spindle encoder'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	A "G33", "G95" or "M19 S" has been programmed without having an encoder on the spindle.
SOLUTION	If the spindle does not have an encoder, functions "M19 S", "G33" or "G95" cannot be programmed. Spindle machine parameter "NPULSES (P13)" indicates the number of encoder pulses per turn.

0157 'G79 not allowed when there is no active canned cycle.'

DETECTION	During execution.
CAUSE	An attempt has been made to execute the "Modification of canned cycle parameters (G79)" function without any canned cycle being active.
SOLUTION	The "G79" function modifies the values of a canned cycle; therefore, there must be an active canned cycle and the "G79" must be programmed in the influence zone of that canned cycle.

0158 'Tool T must be programmed with G67 and G68.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	In the "Irregular pocket canned cycle with islands (G66)" the tool has not been defined for roughing "G67" (2D/3D pockets) for semi-finishing "G67" (3D pocket) or finishing "G68" (2D/3D pocket).
SOLUTION	The irregular pocket canned cycle with islands requires the programming of the roughing tool "G67" (2D/3D pockets), the semi-finishing tool "G67" (3D pocket) and the finishing tool "G68" (2D/3D pocket).

0159 'Inch programming limit exceeded.'

DETECTION	During execution.
CAUSE	An attempt has been made to execute in inches a program edited in millimeters.
SOLUTION	Enter function G70 (inch programming) or G71 (mm programming) at the beginning of the program.

0160 'G79 not allowed when executing the canned cycle.'

No explanation required.

0161 'G66 must be programmed before G67 and G68.'

DETECTION	During execution.
CAUSE	A roughing operation "G67" (2D/3D pockets), a semi-finishing operation "G67" (3D pocket) or a finishing operation "G68" (2D/3D pocket) has been programmed without having previous programmed the call to an "Irregular pocket canned cycle with islands (G66)".
SOLUTION	When working with irregular pockets, before programming the aforementioned cycles, the call to the "Irregular canned cycle with islands (G66)" must be programmed.

0162 'No negative radius allowed with absolute coordinates'

DETECTION	During execution.
CAUSE	While operating with absolute polar coordinates, a movement with a negative radius has been programmed.
SOLUTION	Negative radius cannot be programmed when using absolute polar coordinates.

0163 'The programmed axis is not longitudinal.'

DETECTION	During execution.
CAUSE	An attempt has been made to modify the coordinates of the point where the canned cycle is to be executed using the "Modification of the canned cycle parameters (G79)"function.
SOLUTION	With "G79", the parameters defining a canned cycle may be modified, except the coordinates of the point where it will be executed. To change those coordinates, program only the new coordinates.

0164 'Wrong password.'

DETECTION	While assigning protections.
CAUSE	[ENTER] has been pressed before selecting the type of code to be assigned a password.
SOLUTION	Use the softkeys to select the type of code to which a password is to be assigned.

0165 'Password: use uppercase/lowercase letters or digits.'

DETECTION	While assigning protections.
CAUSE	A bad character has been entered in the password.
SOLUTION	The password can only consist of letters (upper and lower case) or digits.

0166 'Only one HIRTH axis per block is allowed.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	A movement has been programmed which involves the movement of two HIRTH axes simultaneously.
SOLUTION	The CNC does not admit movements involving more than one HIRTH axis at a time. HIRTH axes must move one at a time.

0167 'Rot. axis position.: absolute values (G90) between 0-359.9999.'

DETECTION	During execution.
CAUSE	A movement of a positioning-only rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.
SOLUTION	Positioning-only rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

= 0168 'Rotary axis: absolute values (G90) within ±359.9999.'

DETECTION	During execution.
CAUSE	A movement of a rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.
SOLUTION	Rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0169 'Modal subroutines cannot be programmed.'

DETECTION	While executing in MDI mode
CAUSE	An attempt has been made to call upon a modal subroutine (MCALL).
SOLUTION	MCALL modal subroutines cannot be executed from the menu option "MDI execution".

0170 'Program symbols 0 thru 255 in positions 0-639, 0-335. '

No explanation required.

0171 'The window must be previously defined.'

DETECTION	During normal execution or execution through the user channel.
CAUSE	An attempt has been made to write in a window (DW) that has not been previously defined (ODW).
SOLUTION	It is not possible to write in a window that has not been previously defined. Check that the window to write in (DW) has been previously defined.

0172 'The program is not accessible'

DETECTION	During execution.
CAUSE	An attempt has been made to execute a program that cannot be executed.
SOLUTION	The program may be protected against execution. To know whether a program may be executed, check for the "X" character on the attributes column. If this character is missing, the program cannot be executed.

0173 'It is not possible to program angle + angle.'

No explanation required.

0174 'Circular (helical) interpolation not possible.'

DETECTION	During execution.
CAUSE	An attempt has been made to execute a helical interpolation while the "LOOK-AHEAD (G51)" function was active.
SOLUTION	Helical interpolations are not possible while the "LOOK-AHEAD (G51)" function is active.

0175 'Analog inputs: ANAI(1-8) = +/-5 Volts.'

DETECTION	During execution.
CAUSE	An analog input has taken a value out of the ±5V range.
SOLUTION	Analog inputs may only take values within the ±5V range.

0176 'Analog outputs: ANAO(1-8) = +/-10 Volts.'

DETECTION	During execution.
CAUSE	An analog output has been assigned a value out of the ±10V range.
SOLUTION	Analog outputs may only take values within the ±10V range.

0177 'A gantry axis cannot be part of the active plane.'

No explanation required.

0178 'G96 only possible with analog spindle.'

DETECTION	During execution.
CAUSE	The "G96" function has been programmed but either the spindle speed is not controlled or the spindle does not have an encoder.
SOLUTION	To operate with the "G96" function, the spindle speed must be controlled (SPDLTYPE(P0)=0) and the spindle must have an encoder (NPULSES(P13) other than zero).

0179 'Do not program more than 4 axes simultaneously.'

No explanation required.

0180 'Program DNC1/2/E, HD or CARD A (optional).'

DETECTION While editing or executing.

CAUSE While programming in high level language, in the "OPEN" and "EXEC" instructions, an attempt has been made to program a parameter other than DNC1/2E, HD or CARD A, or the DNC parameter has been assigned a value other than 1, 2 or E.

SOLUTION Check the syntax of the block.

0181 'Program A (append) or D (delete).'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "OPEN" instruction the A/D parameter is missing.

SOLUTION Check the syntax of the block. The programming format is:

(OPEN P.....,A/D,…)

Where:

A Appends new blocks after the existing ones.

D Deletes the existing program and it opens it as a new one.

0182 'Option not available.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A "G" function has been defined which is not a software option.

0183 'Cycle does not exist.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "DIGIT" instruction, a digitizing cycle has been defined which is not available.

SOLUTION The "DIGIT" instruction only admits two types of digitizing:

(DIGIT 1,…) : Grid pattern digitizing cycle.

(DIGIT 2,…) : Arc pattern digitizing cycle.

0184 'T with subroutine: program only T and D.'

No explanation required.

0185 'Tool offset does not exist'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Within the block syntax, a tool offset has been called upon which is greater than the ones allowed by the manufacturer.

SOLUTION Program a new smaller tool offset.

0188 'Function not possible from PLC.'

DETECTION During execution.

CAUSE From the PLC channel and using the "CNCEX" instruction, an attempt has been made to execute a function that is incompatible with the PLC channel execution.

SOLUTION The installation manual (chapter 11.1.2) offers a list of the functions and instructions that may be executed through the PLC channel.

0189 'The live tool does not exist.'

No explanation required.

0190 'Programming not allowed while in tracing mode.'

DETECTION During execution.

CAUSE Among the blocks defining the "Tracing and digitizing canned cycles (TRACE)", there is block that contains a "G" function which does not belong in the profile definition.

SOLUTION The "G" functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09	G36
G39	G53	G70	G71	G90	G91	G93	

0191 'Do not program tracing axes.'

DETECTION During execution.

CAUSE An attempt has been made to move an axis that has been defined as a tracing axis using the "G23" function.

SOLUTION The tracing axes are controlled by the CNC. To deactivate the tracing axes, use the "G25" function..

0192 'Incorrect active plane and longitudinal axis.'

DETECTION	During execution.
CAUSE	While programming in high level language, an attempt has been made to execute a probing cycle using the "PROBE" instruction, but the longitudinal axis is included in the active plane.
SOLUTION	The "PROBE" probing canned cycles are executed on the X, Y, Z axes, the active plane being formed by two of them. The other axis must be perpendicular and it must be selected as the longitudinal axis.

0193 'G23 has not been programmed.'

DETECTION	During execution.
CAUSE	Digitizing "G24" has been activated or a tracing contour "G27" has been programmed, but without previously activating the tracing function "G23".
SOLUTION	To digitize or operate with a contour, the tracing function must be activated previously.

0194 'Repositioning not allowed.'

DETECTION	During execution.
CAUSE	The axes cannot be repositioned using the "REPOS" instruction because the subroutine has not been activated with one of the interruption inputs.
SOLUTION	Before executing the "REPOS" instruction, one of the interruption inputs must be activated.

0195 'Axes X, Y or Z slaved or synchronized.'

DETECTION	During execution.
CAUSE	While programming in high level language, an attempt has been made to execute a probing cycle using the "PROBE" instruction, but one of the X, Y or Z axis is slaved or synchronized.
SOLUTION	To execute the "PROBE"¨ instruction, the X, Y, Z axes must not be slaved or synchronized. To unslave the axes, program "G78".

0196 'Axes X, Y and Z must exist.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, an attempt has been made to edit the "PROBE" instruction, but one of the X, Y or Z axis is missing.
SOLUTION	To operate with the "PROBE" instruction, the X, Y, Z axes must be defined.

0198 'Deflection out of range.'

DETECTION	During execution.
CAUSE	In the tracing cycle "G23", a nominal probe deflection has been defined which is greater than the value set by machine parameter.
SOLUTION	Program a smaller nominal probe deflection.

0199 'Presetting rotary axes: values between 0 and -359.9999.'

DETECTION	While presetting coordinates.
CAUSE	An attempt has been made preset the coordinates of a rotary axis with a value out of the 0 to 359.9999 range.
SOLUTION	The preset value of rotary axes must be within the 0 to 359.9999 range.

0200 'Program: G52 axis +/-5.5'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	When programming the "Movement against a hard stop (G52)", either the axis to be moved has not been programmed or several axes have been programmed.
SOLUTION	When programming "G52", the axis to be moved must be indicated. Only one axis may be programmed at a time.

0201 'Program only one positioning axis in G01.'

No explanation required.

0202 'Program G27 only when tracing a profile.'

DETECTION During execution.

CAUSE A tracing contour (G27) has been defined, but the tracing function is neither bi-dimensional nor three-dimensional.

SOLUTION The "Definition of a tracing contour (G27)" function must only be defined when tracing or digitizing in two or three dimensions.

0203 'G23-G27 not allowed during INSPECTION.'

No explanation required.

0204 'Incorrect tracing method.'

DETECTION During execution.

CAUSE While executing a manual tracing "G23", an attempt has been made to jog a "follower" axis with the jog keys or the electronic handwheels.

SOLUTION When executing a manual tracing, the axes selected as followers are moved by hand. The rest may be jogged with the jog keys or the electronic handwheels.

0205 'Incorrect digitizing method.'

DETECTION During execution.

CAUSE Point-to-point digitizing has been defined, but the CNC is not in jog mode (it is in either in simulation or execution mode, instead).

SOLUTION To execute point-to-point digitizing, the CNC must be in jog mode.

0206 'Values 0 thru 6.'

DETECTION While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values between 0 and 6.

0207 'Complete Table.'

DETECTION While editing tables.

CAUSE In the tables for "M" functions or tool offsets, an attempt has been made to define more data than those allowed by the manufacturer by means of machine parameters. When loading a table via DNC, the CNC does not delete the previous table, it replaces the existing values and it copies the new data in the free positions of the table.

SOLUTION The maximum number of data that can be defined is limited by the machine parameters:

Maximum number of "M" functions	NMISCFUN(P29).
Maximum number of	NTOOL(P23).
Maximum number of tool offset	NTOFFSET(P27).
Maximum number of magazine positions	NPOCKET(P25).

To load a new table via DNC, the previous table should be deleted.

0208 'Program A from 0 to 255'

DETECTION During execution.

CAUSE In the "LOOK-AHEAD (G51)" function, parameter "A" (% of acceleration to be applied) has been programmed with a value greater than 255.

SOLUTION Parameter "A" is optional, but when programmed, it must have a value between 0 and 255.

0209 'Program nesting not allowed.'

DETECTION During execution.

CAUSE From a running program, an attempt has been made to execute another program with the "EXEC" instruction which in turn also has an "EXEC" instruction.

SOLUTION Another program cannot be called upon from a program being executed using the "EXEC" instruction.

0210 'No compensation is permitted.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to activate or cancel tool radius compensation (G41, G42, G40) in a block containing a nonlinear movement.

SOLUTION Tool radius compensation must be activated/deactivated in linear movements (G00, G01).

0211 'Do not program a zero offset without cancelling the previous one.'

DETECTION During execution.

CAUSE An attempt has been made to define an inclined plane using the "Definition of the inclined plane (G49)" function while another one was already defined.

SOLUTION To define a new inclined plane, the one previously defined must be canceled first. To cancel an inclined plane, program "G49" without parameters.

0212 'Programming not permitted while G48-G49 are active.'

DETECTION During execution.

CAUSE While programming in high level language, an attempt has been made to execute a probing cycle with the "PROBE" instruction while function "G48" or "G49" was active.

SOLUTION "PROBE" digitizing cycles are carried out on the X, Y and Z axes. Therefore, in order to be able to execute them, function "G48" or "G49" must not be active.

0213 'A second spindle is required for G28, G29, G77 or G78.'

DETECTION While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to select the work spindle with "G28/G29" or synchronize spindles with "G77/G78", but the machine only has one work spindle.

SOLUTION If the machine only has one work spindle, the "G28, G29, G77 and G78" functions cannot be programmed.

0214 'Invalid G function when selecting a profile'

DETECTION While restoring a profile.

CAUSE Within the group of blocks selected to restore the profile, there is a block containing a "G" code that does not belong in the profile definition.

SOLUTION The "G" functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0215 'Invalid G function after first point of profile'

DETECTION While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a block containing a "G" function that does not belong in the profile definition.

SOLUTION The "G" functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0216 'Nonparametric assignment after first point of profile'

DETECTION While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, a nonparametric assignment has been programmed in high level language (a local or global parameter).

SOLUTION The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0217 'Invalid programming after first point of profile'

DETECTION While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a high level block that is not an assignment.

SOLUTION The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0218 'The axis cannot be programmed after first point of profile'

DETECTION	While restoring a profile.
CAUSE	Within the selected blocks for restoring the profile, and after the starting point of a profile, a position has been defined on an axis that does not belong to the active plane. A surface coordinate may have been defined after the starting point of the profile.
SOLUTION	The surface coordinate of the profiles is only defined in the starting block of the first profile, the one corresponding to the starting point of the outside profile.

0219 'First point programmed wrong when selecting profile'

DETECTION	While selecting a profile.
CAUSE	The starting point of the profile has been programmed wrong. One of the two coordinates defining its position is missing.
SOLUTION	The starting point of a profile must be defined on the two axes forming the active plane.

0220 'Invalid axes'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The defined axes are not valid for G46.
SOLUTION	Check the following: • g.m.p. ANGAXNA (P171) and g.m.p. ORTAXNA (P172) are other than 0. • The defined axes exist and are linear.

0226 'A tool cannot be programmed with G48 active.'

DETECTION	During execution.
CAUSE	A tool change has been programmed while the "TCP transformation (G48)" function is active.
SOLUTION	A tool change cannot take place while TCP transformation is active. To make a tool change, cancel TCP transformation first.

0227 'Program Q between +/-359.9999.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	In the "Electronic threading (G33)" function, the entry angle "Q" has been programmed with a value out of the ±359.9999 range.
SOLUTION	Program an entry angle within the ±359.9999 range.

0228 'Do not program "Q" with parameter M19TYPE=0.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	In the "Electronic threading (G33)" function, an entry angle "Q" has been programmed, but the type of spindle orientation available does not allow this operation.
SOLUTION	In order to define an entry angle, spindle machine parameter M19TYPE(P43) must be set to "1".

0229 'Program maximum X'
0230 'Program minimum Y'
0231 'Program maximum Y'
0232 'Program minimum Z'
0233 'Program maximum Z'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, in the "DGWZ" instruction, the indicated limit is missing or it has been defined with a non-numerical value.
SOLUTION	Check the syntax of the block.

0234 'Wrong graphic limits'

DETECTION	During execution.
CAUSE	One of the lower limits defined with the "DGWZ" instruction is greater than its corresponding upper limit.
SOLUTION	Program the upper limit of the graphics display area greater than the lower ones.

0235 'Do not program the axis in tangential control'

No explanation required.

0236 'Do not program the longitudinal axis or the axis of the active plane'

No explanation required.

0237 'Program values between +/-359.9999.'

DETECTION	During execution.
CAUSE	A G30 offset has been programmed greater than the maximum allowed. For example G30 D380
SOLUTION	The offset must be within ±359.9999.

0238 'Do not program G30 without synchronizing the spindles in speed'

DETECTION	During execution.
CAUSE	An attempt has been made to synchronize the spindles in "G30" offset without having them synchronized in speed.
SOLUTION	First, synchronize the spindle in speed using G77S.

0239 'Do not synchronize the spindles while the "C" axis is active'

DETECTION	During execution.
CAUSE	An attempt has been made to synchronize the spindle, but the "C" axis is not active.
SOLUTION	Activate the "C" axis first.

0240 'Do not activate the "C" axis while the spindles are synchronized'

DETECTION	During execution.
CAUSE	An attempt has been made to activate the "C" axis while the spindles were synchronized.
SOLUTION	First, cancel the spindle synchronization (G78 S).

0241 'Don't program G77 S, G78 S with no spindle encoder'

DETECTION	During execution.
CAUSE	An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them does not have an encoder or Sercos feedback.
SOLUTION	Both spindles must have an encoder or Sercos feedback.

0242 'Do not synchronize spindles with M19TYPE=0'

DETECTION	During execution.
CAUSE	An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them has parameter M19TYPE=0.
SOLUTION	Both spindles must have parameter M19TYPE=1

0243 'Values 0 thru 15.'
0244 'Values between 0.00% - 100.00%.'
0245 'Values between -100.00% - 100.00%.'

No explanation required.

0246 'The feedrate cannot be negative or zero.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	If g.m.p. FEEDTYPE (P170) has a value other than ·0·, F0 cannot be programmed.
SOLUTION	The possible solutions are: • Set g.m.p. FEEDTYPE (P170) to ·0·. In this case, the motion blocks are executed at the maximum feedrate allowed. • Program F other than ·0·.

0247 'Values 0 thru 8.'

No explanation required.

FAGOR

·M· Model

Ref. 1310

BLOCK PREPARATION AND EXECUTION ERRORS

1000 'There is no enough path information.'

DETECTION	During execution.
CAUSE	The program contains too many blocks without information about the path to apply tool radius compensation, rounding, chamfer or tangential entry or exit.
SOLUTION	In order to carry out these operations, the CNC needs to know in advance the path to follow; therefore, there cannot be more than 48 blocks in a row without information about the path to follow.

1001 'Plane change in rounding/chamfering.'

DETECTION	During execution.
CAUSE	A plane change has been programmed on the path following the definition of a "controlled corner rounding G36" or "chamfer (G39)".
SOLUTION	The plane cannot be changed while executing a rounding or a chamfer. The path following the definition of a rounding or chamfer must be in the same plane that the rounding or the chamfer.

1002 'Rounding radius too large.'

DETECTION	During execution.
CAUSE	In the "Controlled corner rounding (G36)" function, the programmed rounding radius is larger than one of the paths where it has been defined.
SOLUTION	The rounding radius must be smaller than the paths that define it.

1003 'Rounding in last block.'

DETECTION	During execution.
CAUSE	A "Controlled rounding radius (G36) or "Chamfer (G39) has been defined on the last path of the program or when the CNC does not find information about the path following the definition of the rounding or chamfer.
SOLUTION	A rounding or chamfer must be defined between two paths.

1004 'Tangential output programmed wrong'

DETECTION	During execution.
CAUSE	The move following the definition of a tangential output (G38) is a circular path.
SOLUTION	The move following the definition of a tangential output must be a straight path.

1005 'Chamfer programmed wrong.'

DETECTION	During execution.
CAUSE	The move following the definition of a "Chamfer (G39)" is a circular path.
SOLUTION	The move following the definition of a chamfer must be a straight path.

1006 'Chamfer value too large.'

DETECTION	During execution.
CAUSE	In the "Chamfer (G39)" function, the programmed chamfer value is larger than one of the paths where it has been defined.
SOLUTION	The chamfer size must be smaller than the paths that define it.

1007 'G8 defined wrong.'

DETECTION During execution.

CAUSE The possible causes are:

1. When a full circle has been programmed using the function "Arc tangent to previous path (G08)"
2. When the tangent path ends in a point of the previous path or its extension (in a straight line).
3. In an irregular pocket canned cycle with islands, when programming function "G08" in the block following the definition of the beginning of the profile (G00).

SOLUTION The solution for each cause is:

1. Function "G08" does not allow programming full circles.
2. Tangent path must not end in a point of the previous path or in its extension (in a straight line).
3. The CNC does not have information about the previous path and cannot execute the tangent arc.

1008 'There is no information about the previous path'

DETECTION During execution.

CAUSE An arc tangent to the previous path has been programmed using function "G08", but there is no information about the previous path.

SOLUTION To do a path tangent to the previous one, there must be information about the previous path and it must be within the 48 blocks preceding the tangent path.

1009 'There is no information for tangent arc in pockets with islands.'

DETECTION During execution.

CAUSE Within the set of blocks defining the profile of an irregular pocket with islands, a tangent arc has been programmed, but some data is missing or there is not enough information about the previous path.

SOLUTION Check the data that defines the profile.

1010 'Wrong plane for tangent path.'

DETECTION During execution.

CAUSE A plane change has been programmed between the definition of the function "arc tangent to previous path (G08)" and the previous path.

SOLUTION A plane cannot be changed between two paths

1011 'Jog movement out of limits.'

DETECTION During execution.

CAUSE After defining an inclined plane, the tool positions at a point out of the work limits; the operator tries to move an axis with the JOG keys, the tool does not position within the area defined by the work limits.

SOLUTION Jog the axis that allows to position the tool within the work limits.

1012 'G48 cannot be programmed while G43 is active'

DETECTION During execution.

CAUSE An attempt has been made to activate TCP (G48) while tool length compensation (G43) was active.

SOLUTION To activate TCP transformation (G48), tool length compensation must be OFF because TCP already applies its own specific tool length compensation.

1013 'G43 cannot be programmed with G48 active'

DETECTION During execution.

CAUSE An attempt has been made to activate tool length compensation (G43) while TCP (G48) was active.

SOLUTION To activate tool length compensation (G43) cannot be activated while TCP transformation (G48) is ON because TCP already applies its own specific tool length compensation.

1014 'G49 cannot be programmed if it"s already active'

No explanation required.

1015 'The tool is not defined in the tool table'

DETECTION	During execution.
CAUSE	A tool change has been defined, but the new tool is not defined in the tool table.
SOLUTION	Define the new tool in the tool table.

1016 'The tool is not in the tool magazine'

DETECTION	During execution.
CAUSE	A tool change has been defined, but the new tool is not defined in position of the tool magazine table.
SOLUTION	Define the new tool in the tool magazine table.

1017 'There is no empty pocket in the tool magazine'

DETECTION	During execution.
CAUSE	A tool change has been defined and there is no empty pocket for the tool that is currently in the spindle.
SOLUTION	Perhaps, the new tool has been defined as special in the tool table and there are more than one pockets reserved to it in the magazine. In this case, that position is set for that tool and no other tool can occupy it. To avoid this error, an empty pocket (position) should be left in the tool magazine.

1018 'A tool change has been programmed without M06'

DETECTION	During execution.
CAUSE	An M06 has not been programmed after having looked for a tool and before searching again.
SOLUTION	This error occurs when having a machining center (general machine parameter TOFFM06(P28)=YES) that has a cyclic tool changer (general machine parameter CYCATC(P61)=YES). In this case, the tool change must be done with an m06 after searching for a tool and before searching for the next one.

1019 'There is no tool of the same family for replacement.'

DETECTION	During execution.
CAUSE	The real life of the requested tool exceeds its nominal life. The CNC has tried to replace it with another one of the same family, but it has not found any.
SOLUTION	Replace the tool or define another one of the same family.

1020 'Do not change the active or pending tool using high level language.'

DETECTION	During execution.
CAUSE	While programming in high level language and using the "TMZT" variable, an attempt has been made to assign the current or next tool to a magazine position.
SOLUTION	Use the "T" function to change the active tool or the next one. The "TMZT" variable cannot be used to move the active tool or the next one to the magazine.

1021 'No tool offset has been programmed in the canned cycle.'

DETECTION	During execution.
CAUSE	The "PROBE" canned cycle for tool calibration has been programmed, but no tool offset has been selected.
SOLUTION	To execute the "Tool calibration canned cycle (PROBE), a tool offset must be selected where the probing cycle information will be stored.

1022 'Tool radius programmed incorrectly'

No explanation required.

1023 'G67. Tool radius too large.'

DETECTION	During execution.
CAUSE	In the "Irregular pocket canned cycle with islands (G66)", a tool has been selected whose radius is too large for the roughing operation "G67" (2D pocket). The tool cannot get in anywhere in the pocket.
SOLUTION	Select a tool of a smaller radius.

1024 'G68. Tool radius too large.'

DETECTION	During execution.
CAUSE	In the "Irregular pocket canned cycle with islands (G66)", a tool has been selected whose radius is too large for the finishing operation "G68" (2D pocket). Somewhere in the machining operation, the distance between the outside profile and the profile of an island is smaller than the tool diameter.
SOLUTION	Select a tool of a smaller radius.

1025 'A tool with no radius has been programmed'

DETECTION	During execution.
CAUSE	In the "Irregular pocket canned cycle with islands (G66), a (G67/G68) operation has been programmed with no radius.
SOLUTION	Correct the tool definition in the tool table or select another one for that operation.

1026 'A step has been programmed that is larger than the tool diameter'

DETECTION	During execution.
CAUSE	In the "Rectangular pocket canned cycle (G87)", in the "circular pocket canned cycle (G68) or in an operation of the "irregular pocket canned cycle with islands (G66)", the "C" parameter has been programmed with a value larger than that of the tool that will be used for that operation.
SOLUTION	Correct the syntax of the block. The machining step "C" must be smaller than or equal to the tool diameter.

1027 'A tool cannot be programmed with G48 active.'

DETECTION	During execution.
CAUSE	A tool change has been programmed while the "TCP transformation (G48)" function is active.
SOLUTION	A tool change cannot take place while TCP transformation is active. To make a tool change, cancel TCP transformation first.

1028 'Do not switch axes over while G23, G48 or G49 is active'

DETECTION	During execution.
CAUSE	An attempt has been made to switch over to an axis or back (G28/G29) while function "G23", "G48" or "G49" was active.
SOLUTION	The axes cannot be swapped while function "G23", "G48" or "G49" is active.

1029 'Do not swap axes that are already swapped.'

DETECTION	During execution.
CAUSE	An attempt has been made to swap (G28) an axis that was already swapped with another one.
SOLUTION	An axis already swapped with another one cannot be swapped with a third one. It must be switched back first (G29 axis)

1030 'The "M" for the automatic gear change does not fit.'

DETECTION	During execution.
CAUSE	Using automatic gear change, 7 "M" functions and the "S" function (involving a gear change) have been programmed. In this case, the CNC cannot include the "M" for automatic gear change in that block.
SOLUTION	Program an "M" function or the "S" function in a separate block.

1031 'No subroutine is allowed with automatic gear change.'

DETECTION	During execution.
CAUSE	On machines having an automatic gear change, when programming a spindle speed "S" that involves a gear change and the "M" function of the automatic gear change has a subroutine associated with it.
SOLUTION	When having an automatic gear change, the "M" functions corresponding to the gear change cannot have a subroutine associated with it.

1032 'Spindle gear not defined in M19.'

DETECTION During execution.

CAUSE "M19" has been programmed, but none of the gear change functions "M41", "M42", "M43" or "M44" are active.

SOLUTION On power-up, the CNC does not assume any gear; Therefore, if the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions ("M41", "M42", "M43" or "M44") must be programmed.

1033 'Wrong gear change.'

DETECTION During execution.

CAUSE The possible causes are:

1. When trying to make a gear change and the machine parameters for gears (MAXGEAR1, MAXGEAR2, MAXGEAR3, or MAXGEAR4) are set wrong. All the gears have not been used and the unused ones have been set to a maximum speed of zero rpm.
2. When programming a gear change ("M41", "M42", "M43" or "M44") and the PLC has not responded with the relevant active gear signal (GEAR1, GEAR2, GEAR3 or GEAR4).

SOLUTION The solution for each cause is:

1. When not using all four gears, the lower ones must be used starting with "MAXGEAR1" and the unused gears must be assigned the value of the highest one used.
2. Check the PLC program.

1034 '"S" has been programmed, but no gear is active.'

DETECTION During execution.

CAUSE An attempt has been made to start the spindle, but no gear is selected.

SOLUTION On power-up, the CNC does not assume any gear; Therefore, when programing a spindle speed and the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions ("M41", "M42", "M43" or "M44") must be programmed.

1035 'Programmed "S" too high'

DETECTION During execution.

CAUSE An "S" has been programmed with a higher value than allowed by the last active gear.

SOLUTION Program a lower spindle speed "S" .

1036 '"S" has not been programmed in G95 or in threading'

DETECTION During execution.

CAUSE "mm(inches)/revolution (G95)" or "electronic threading (G33)"has been programmed, but no spindle speed has been selected.

SOLUTION An "S" must be programmed to work in mm/rev (G95) or for an electronic threading (G33).

1038 'The spindle has not been oriented'

DETECTION During execution.

CAUSE A threading cycle is to be executed without having oriented the active spindle (main or secondary) first.

1040 'Canned cycle does not exist'

DETECTION While executing in MDI mode

CAUSE When trying to execute a canned cycle (G8x) after interrupting a program during the execution of a canned cycle (G8x) and then changing the plane.

SOLUTION Do not interrupt the program while executing a canned cycle.

1041 'Mandatory parameter missing in canned cycle'

DETECTION During execution.

CAUSE The possible causes are:

1. In the "Irregular canned cycle with islands" some parameter is missing.

   2D POCKETS:

   - In the roughing operation "G67", either parameter "I" or "R" is missing.
   - There is no roughing operation and in the finishing operation "G68", either parameter "I" or "R" is missing.

   3D POCKETS:

   - In the roughing operation "G67", either parameter "I" or "R" is missing.
   - There is no roughing operation and in the semifinishing operation "G67", either parameter "I" or "R" is missing.
   - There is neither roughing nor semifinishing operation and in the finishing operation "G68", either parameter "I" or "R" is missing.
   - In the finishing operation "G68", parameter "B" is missing.

2. In the "Digitizing canned cycle" some parameter is missing.

SOLUTION Correct the definition of parameters.

Pocket with islands (finishing operation).

In the irregular pocket canned cycle with islands, parameters "I" and "R" must be programmed in the roughing operation. If there is no roughing operation, they must be defined in the finishing operation (2D) or in the semifinishing operation (3D). If there is no semifinishing operation (3D), they must be defined in the finishing operation. In the 3D pocket, parameter "B" must be defined in the finishing operation.

Digitizing cycles.

Check the syntax of the block. The programming formats are:

(DIGIT 1,X,Y,Z,I,J,K,B,C,D,F)

(DIGIT 2,X,Y,Z,I,J,K,A,B,C,F)

1042 'Wrong parameter value in canned cycle'

DETECTION During execution.

CAUSE The possible causes are:

1. In the "Irregular pocket canned cycle with islands", when a parameter has been defined with a wrong value in the finishing operation "G68". Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).
2. In the "Irregular pocket canned cycle with islands", when in the drilling operation (G69) parameter "B", "C" or "H" has been defined with a zero value.
3. In the rectangular (G87) or circular (G88) pocket canned cycles, either parameter "C" or a pocket dimension has been defined with a zero value.
4. In the "Deep hole drilling canned cycle with variable peck (G69), parameter "C" has been defined with zero value.
5. In the digitizing canned cycle, a parameter has been assigned a wrong value. Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).

SOLUTION Correct the definition of parameters:

Pocket with islands (finishing operation).

"Q" parameter	Only takes a value of 0, 1 or 2.
"B" parameter	Only takes values other than zero.
"J" parameter	It must be smaller than the radius of the tool used for that operation.

GRID pattern digitizing.

"B" parameter	Only takes positive values greater than zero.
"C" parameter	Only takes positive values other than zero.
"D" parameter	It only admits values of 0 or 1.

ARC pattern digitizing.

"J" and "C" parameter	Only takes positive values greater than zero.
"K", "A" and "B" parameter	It only admits positive values.

1043 'Wrong depth profile in pocket with islands.'

DETECTION During execution.

CAUSE In the "Irregular pocket canned cycle with islands" (3D):
- The depth profiles of two sections of the same contour (simple or composite) cross each other.
- A contour cannot be finished with the programmed tool (spherical path with non-spherical tool).

SOLUTION The depth profiles of two sections of the same profile cannot cross each other. On the other hand, the depth profile must be defined after the plane profile and the same starting point must be used in both profiles. Check that the tip of the selected tool is the best for the programmed depth profile.

1044 'Plane profile intersects itself in a pocket with islands'

DETECTION During execution.

CAUSE Within the set of profiles that define a pocket with islands, one of the profiles intersects itself.

SOLUTION Check the definition of the profiles. The profile of a pocket with islands cannot intersect itself.

1045 'Error when programming a drilling operation in a pocket with islands.'

DETECTION During execution.

CAUSE In the "Irregular pocket canned cycle with islands (G66), a canned cycle has been programmed that is not for drilling.

SOLUTION In the drilling operation, only canned cycle "G81", "G82", "G83" or "G69" may be programmed.

1046 'Wrong tool position before the canned cycle'

DETECTION During execution.

CAUSE When calling a canned cycle, the tool is positioned between the reference plane and the final depth coordinate of one of the operations.

SOLUTION When calling a canned cycle, the tool must be positioned above the reference plane.

1047 'Open plane profile in pocket with islands'

DETECTION During execution.

CAUSE Within the set of profiles that define a pocket with islands, one of the profiles does not start and end at the same point.

SOLUTION Check the definition of the profiles. The profiles that define the pockets with islands must be closed. The error may occurred because "G01" has not been programmed after the beginning, with "G00", of one of the profiles.

1048 'Part surface coordinate not programmed in pocket with islands'

DETECTION During execution.

CAUSE The part surface coordinate of the pocket has not been programmed at the first point of the geometry definition.

SOLUTION The data for the surface coordinate must be defined in the first definition block of the pocket profile (in absolute coordinates).

1049 'Wrong reference plane coordinate in canned cycle'

DETECTION During execution.

CAUSE In an operation of the "Irregular pocket canned cycle with islands (G66), the coordinate of the reference plane is located between the part surface coordinate and the final depth coordinate of one of the operations.

SOLUTION The reference plane must be located above the part surface. This error comes up sometimes because the part surface position has been programmed in incremental coordinates. (The pocket surface data must be programmed in absolute coordinates).

1050 'Wrong value to be assigned to a variable'

DETECTION During execution.

CAUSE Using parameters, the value assigned to a variable is too high.

SOLUTION Check the program history to make sure that this parameter does not have that value when it reaches the block where this assignment is made.

1051 'Wrong access to PLC variables.'

DETECTION During execution.

CAUSE From the CNC, an attempt has been made to read a PLC variable that is not defined in the PLC program.

1052 'Access to a variable with wrong index'

DETECTION During editing.

CAUSE While programming in high level language, an operation has been carried out either with a local parameter greater than 25 or with a global parameter greater 299.

SOLUTION The parameters used by the CNC are:

Local: 0-25.

Global: 100-299.

Other parameters out of these ranges cannot be used in operations.

1053 'Local parameters not accessible'

DETECTION While executing in the user channel.

CAUSE An attempt has been made to execute a block with an operation that uses local parameters.

SOLUTION The program that is executed in the user channel does not allow operations with local parameters (P0 to P25).

1054 'Limit of local parameters exceeded'

DETECTION During execution.

CAUSE While programming in high level language, more than 6 nesting levels have been used with the "PCALL" instruction. More than 6 calls have been made in the same loop using the "PCALL" instruction.

SOLUTION Only up to 6 nesting levels are allowed for local parameters within the 15 nesting levels of the subroutines. Calling with a "PCALL" instruction generates a new nesting level for local parameters (and a new one for subroutines).

1055 'Nesting exceeded'

DETECTION During execution.

CAUSE While programming in high level language, more than 15 nesting levels have been used with the "CALL", "PCALL" or "MCALL" instruction. More than 15 calls have been made in the same loop using the "CALL", "PCALL" or "MCALL" instruction.

SOLUTION Only 15 nesting levels allowed. Calling with the "CALL", "PCALL" and "MCALL" instructions generates a new nesting level.

1056 'RET not associated with subroutine.'

DETECTION During execution.

CAUSE The "RET" instruction has been edited, but the "SUB" instruction has not been edited before.

SOLUTION To using the "RET" instruction (subroutine), the subroutine must begin with the "SUB (subroutine number)".

1057 'Undefined subroutine'

DETECTION During execution.

CAUSE A (CALL, PCALL...) has been made to a subroutine that was not defined in the CNC memory.

SOLUTION Check that the name of the subroutine is correct and that the subroutine exists in the CNC memory (not necessarily in the same program where the call is).

1058 'Undefined probing canned cycle'

DETECTION During execution.

CAUSE Using the "PROBE" instruction, a probing cycle has been defined which is not available.

SOLUTION The available "PROBE" canned cycles are 1 to 9.

1059 'Jump to an undefined label'

DETECTION During execution.

CAUSE While programming in high level language, the "GOTO N..." instruction has been programmed, but the programmed block number (N) does not exist.

SOLUTION When programming the "GOTO N..." instruction, the block it refers to must be defined in the same program.

1060 'Undefined label'

DETECTION During execution.

CAUSE The possible causes are:

1. While programming in high level language, the instrucción "RPT N..., N..." instruction has been programmed, but a programmed block number (N) does not exist.
2. When programming "G66 ... S...E..." in an "Irregular pocket canned cycle with islands (G66) and one of the data defining the beginning or the end of the profiles is missing.

SOLUTION The solution for each cause is:

1. When programming the "RPT N..., N..." instruction, the blocks it refers to must be defined in the same program.
2. Check the program. Place the label for parameter "S" at the beginning of the profile definition and the label for parameter "E" at the end of the profile definition.

1061 'Label cannot be searched'

DETECTION While executing in MDI mode

CAUSE While programming in high level language, either an "RPT N..., N..." or "GOTO N..." instruction has been defined.

SOLUTION While operating in MDI mode, "RPT" or "GOTO" type instructions cannot be programmed.

1062 'Subroutine in an unavailable program.'

DETECTION During execution.

CAUSE A call has been made to a subroutine that it is located in a program being used by the DNC.

SOLUTION Wait for the DNC to finish using the program. If the subroutine is to be used often, it should be stored in a separate program.

1063 'The program cannot be opened.'

DETECTION During execution.

CAUSE While executing a program in infinite mode, an attempt has been made to execute another infinite program from the current one using the "EXEC" instruction.

SOLUTION Only one infinite program may be executed at a time.

1064 'The program cannot be executed'

DETECTION During execution.

CAUSE An attempt has been made to execute a program from another with the "EXEC" instruction, but the program does not exist or is protected against execution.

SOLUTION The program to be executed with the "EXEC" instruction must exist in the CNC memory and must be executable.

1065 'Beginning of compensation without straight path'

DETECTION During execution.

CAUSE The first movement in work plane after activating tool radius compensation (G41/G42) is not a linear movement.

SOLUTION The first movement after activating radius compensation (G41/G42) must be linear.

1066 'End of compensation without straight path'

DETECTION During execution.

CAUSE The first movement in work plane after deactivating tool radius compensation (G40) is not a linear movement.

SOLUTION The first movement after deactivating radius compensation (G40) must be linear.

1067 'Compensation radius too large.'

DETECTION	During execution.
CAUSE	While working with tool radius compensation (G41/G42), an inside radius has been programmed with a smaller radius than that of the tool.
SOLUTION	use a tool with a smaller radius. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path.

1068 'Step on linear path'

DETECTION	During execution.
CAUSE	When operating with tool compensation (G41/G42), the profile has a straight section that cannot be machined because the tool diameter is too large.
SOLUTION	use a tool with a smaller radius.

1069 'Circular path defined incorrectly'

No explanation required.

1070 'Step on circular path'

DETECTION	During execution.
CAUSE	When operating with tool compensation (G41/G42), the profile has a curved section that cannot be machined because the tool diameter is too large.
SOLUTION	use a tool with a smaller radius.

1071 'Plane change in tool radius compensation.'

DETECTION	During execution.
CAUSE	When operating with tool compensation (G41/G42), another work plane has been selected.
SOLUTION	To change the work plane, tool radius compensation must be off (G40).

1072 'Tool radius compensation not possible with positioning-only rotary axis.

DETECTION	During execution.
CAUSE	An attempt has been made to move a positioning-only axis with tool radius compensation (G41/G42).
SOLUTION	Tool radius compensation not allowed for positioning-only rotary axes. Use "G40" to cancel tool radius compensation.

1073 Motion block with zero speed.

DETECTION	During execution.
CAUSE	If g.m.p. FEEDTYPE (P170) has a value other than ·0·, F0 cannot be programmed.
SOLUTION	The possible solutions are: • Set g.m.p. FEEDTYPE (P170) to ·0·. In this case, the motion blocks are executed at the maximum feedrate allowed. • Program F other than ·0·.

1074 'INIPAR cannot be executed.'

DETECTION	During execution.
CAUSE	To validate the machine parameters associated with a kinematics, G48 and G49 must NOT be active.
SOLUTION	Cancel functions G48 thru G49.

1075 'G51 is incompatible helical path.'

DETECTION	During execution.
CAUSE	A helical path has been executed while function G51 was active.
SOLUTION	Cancel G51 before executing the helical path.

1076 'Coordinate angle programmed wrong.'

DETECTION	During execution.
CAUSE	When programming in angle-coordinate format, an axis movement has been programmed with an angle perpendicular to that axis. (For example, the main plane is formed by the XY axes and the X axis movement is programmed at a 90º angle).
SOLUTION	Check and correct the definition of the movement in the program. If using parameters, check that the parameters have the correct values when arriving to the definition of the movement.

1077 'Either the arc radius is too small or a full circle has been programmed'

DETECTION	During execution.
CAUSE	The possible causes are: 1. When programming a full circle using the "G02/G03 X Y R" format. 2. When programming using the "G02/G03 X Y R" format, the distance to the arc's end point is greater than the diameter of the programmed circle.
SOLUTION	The solution for each cause is: 1. This format cannot be used to make full circles. Program the coordinates of the end point different from those of the starting point. 2. The diameter of the circle must be larger than the distance to the arc's end point.

1078 'Negative radius in polar coordinates'

DETECTION	During execution.
CAUSE	Working with incremental polar coordinates, a block is executed where the end position has a negative radius.
SOLUTION	Incremental polar coordinate programming allows negative radius, but the (absolute) end point of the radius must be positive,

G74 'There is no subroutine associated with G74'

DETECTION	While executing a home search.
CAUSE	The various causes might be: 1. When trying to search home on all the axes manually, but there is no associated subroutine indicating the home searching sequence (order). 2. "G74" has been programmed, but there is no associated subroutine indicating the home searching sequence (order).
SOLUTION	The solution for each cause is: 1. An associated subroutine is required to execute the "G74" function. 2. If "G74" is to be executed from a program, the home searching order must be defined.

1080 'Plane change in tool inspection'

DETECTION	While executing the "tool inspection" option.
CAUSE	the work plane has been chanted and the original one has not been restored before resuming the execution.
SOLUTION	The plane that was active before inspecting the tool must be restored before resuming the execution.

1081 'Block not allowed in tool inspection.'

DETECTION	While executing the "tool inspection" option.
CAUSE	An attempt has been made to execute the "RET" instruction.
SOLUTION	This instruction cannot be executed in the "tool inspection" option.

1082 'The probe signal has not been received.'

DETECTION During execution.

CAUSE The possible causes are:

1. When programming a "PROBE" canned cycle, the probe has moved the maximum safety distance of the cycle without the CNC receiving the probe signal.
2. When programming the "G75" function, it has reached the end point and the CNC has not received the signal from the probe. (Only when general machine parameter PROBERR(P119)=YES).

SOLUTION The solution for each cause is:

1. Check that the probe is connected properly.

   The maximum probing distance (in PROBE cycles) depends on the safety distance "B". To increase this distance, increase the safety distance.
2. If PROBERR(P119)=NO, this error will not be issued when the end point is reached without having received the probe signal (only with "G75").

1083 'Range exceeded'

DETECTION During execution.

CAUSE The distance for the axes to travel is very long and the programmed feedrate is too low.

SOLUTION Program a higher speed for that movement.

1084 'Arc programmed wrong'

DETECTION During execution.

CAUSE The possible causes are:

1. When the arc programmed using "G02/G03 X Y I J" cannot go through the defined end point.
2. When programming an arc using "G09 X Y I J" the three points are in line or two of them are the same.
3. When trying to do a rounding tangential entry on a path that is not in the active plane.
4. When programming a tangential exit and the next path is tangent (being on its straight extension) to the path preceding the tangential exit.

   If the error comes up in the block calling the "Irregular canned cycle with islands" is because one of the cases mentioned earlier occurs in the set of blocks defining the profiles of a pocket with islands.

SOLUTION The solution for each cause is:

1. Correct the syntax of the block. The coordinates of the end point or of the radius are defined wrong.
2. The three points used to define an arc must be different and cannot be in line.
3. Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.
4. The path after a tangential exit may be tangent, but it cannot be on the extension (in a straight line) of the previous path.

1085 'Helical path programmed wrong'

DETECTION During execution.

CAUSE When programming an arc using "G02/G03 X Y I J Z K", the programmed arc is impossible. The desired height cannot be reached with the programmed helical pitch.

SOLUTION Correct the syntax of the block. The height of the interpolation and the coordinates of the end point in the plane must be related taking the helical pitch into account.

1086 'The spindle cannot be homed.'

CAUSE Spindle machine parameter REFEED1(P34) = 0.

1087 'Circle with zero radius'

DETECTION During execution.

CAUSE The possible causes are:

1. When programming an arc using "G02/G03 X Y I J", an arc has been programmed with a zero radius.
2. When operating with tool radius compensation, an inside arc has been programmed with the same radius as that of the tool.

SOLUTION The solution for each cause is:

1. Arcs with zero radius are not allowed. Program a radius other than zero.
2. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path (because to do so, the tool would have to make an arc of zero radius).

1088 'Range exceeded in zero offset.'

DETECTION During execution.

CAUSE A zero offset has been programmed and the value of the end position is too high.

SOLUTION Check that the values assigned to the zero offsets (G54-G59) are correct. If the zero offsets have been assigned values from the program using parameters, check that the parameter values are correct. If an absolute (G54-G57) and an incremental (G58-G59) zero offset has been programmed, check that the sum of both does not exceed the machine limits.

1089 'Range exceeded in zone limit.'

DETECTION During execution.

CAUSE When programming zone limits "G20" or "G21" with parameters, the parameter value is greater than the maximum allowed for that function

SOLUTION Check the program history to make sure that this parameter does not have that value when it reaches the block where the limits have been defined.

1090 'Point inside the forbidden zone 1.'

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of its channel.

CAUSE An attempt has been made to move an axis to a point located inside the work area 1 that is defined as "no entry" zone.

SOLUTION In the program history, work zone 1 (defined with G20/G21) has been set as "no entry" zone " (G22 K1 S1). To cancel this work zone, program "G22 K1 S0"

1091 'Point inside the forbidden zone 2.'

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of its channel.

CAUSE An attempt has been made to move an axis to a point located inside the work area 2 that is defined as "no entry" zone.

SOLUTION In the program history, work zone 2 (defined with G20/G21) has been set as "no entry" zone " (G22 K1 S1). To cancel this work zone, program "G22 K2 S0"

1092 'Insufficient acceleration for the speed programmed in threading.'

DETECTION During execution.

CAUSE A thread has been programmed and there isn't enough room to accelerate and decelerate.

SOLUTION Program a lower speed.

1093 'Only one Hirth axis can be moved at a time'

No explanation required.

1094 'Probe calibrated wrong'

No explanation required.

1095 'Probing axes out of alignment .'

DETECTION	During the probe calibration process.
CAUSE	An axis has moved to touch a cube and one of the axis that did not move registers a deflection greater than allowed by machine parameter MINDEFLE(P66). This is because the probing axes are not parallel enough to the axes of the machine.
SOLUTION	Correct the parallelism between the probing axes and those of the machine.

1096 'Point inside the forbidden zone 3.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located inside the work area 3 that is defined as "no entry" zone.
SOLUTION	In the program history, work zone 3 (defined with G20/G21) has been set as "no entry" zone " (G22 K3 S1). To cancel this work zone, program "G22 K3 S0"

1097 'Point inside the forbidden zone 4.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located inside the work area 4 that is defined as "no entry" zone.
SOLUTION	In the program history, work zone 4 (defined with G20/G21) has been set as "no entry" zone " (G22 K4 S1). To cancel this work zone, program "G22 K4 S0"

1098 'Work zone limits defined wrong'

DETECTION	During execution.
CAUSE	The upper limits (G21) of the defined work zone are the same or smaller than the lower ones (G20) of the same work zone.
SOLUTION	Program the upper limits (G21) of the work zone greater than the lower ones (G20).

1099 'Do not program a slaved axis.'

DETECTION	During execution.
CAUSE	When operating in polar coordinates, a movement has been programmed that involves an axis that is slaved to another one.
SOLUTION	The movements in polar coordinates are made with the main axes of the work plane; therefore, the axes that define the plane cannot be slaved to each other or to a third one. To unslave the axes, program "G78".

1100 'Travel limits of spindle 1 exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to exceed the physical turning limits of the spindle. As a result, the PLC activates the spindle mark "LIMIT+S" or "LIMIT-S". ("LIMIT+S2" or "LIMIT-S2" when working with the second spindle).

1101 'Spindle 1 locked'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	The CNC tries to output the command to the drive when the spindle input SERVOSON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the spindle parameter DWELL(P17) is not high enough.

1102 'Following error of spindle 1 out of limit'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	When the spindle is working in closed loop (M19), its following error is greater than the values indicated by spindle parameter MAXFLWE1(P21) and MAXFLWE2(P22) The possible causes for this error are: Servo drive error Faulty drive. Enable signals missing. Power supply missing. Drive adjusted incorrectly. The velocity command signal is not received. Motor error Faulty motor. Power cables. Feedback failure Defective feedback. Defective feedback cable. Mechanical failure Mechanical stiffness. Spindle mechanically locked. CNC error Defective CNC. Parameters adjusted incorrectly.

1103 'Do not synchronize spindles without homing them first'

DETECTION	During execution.
CAUSE	An attempt has been made to synchronize the spindle without homing them first.
SOLUTION	Before activating the synchronization, both spindles must be homed using the "M19" function.

1104 ' Do not program G28 or G29 while spindle synchronization is active'

DETECTION	During execution.
CAUSE	An attempt has been made to swap spindles (G28/G29) while the spindles were synchronized.
SOLUTION	First, cancel spindle synchronization (G78S).

1105 'Do not change gears while the spindles are synchronized'

DETECTION	During execution.
CAUSE	While the spindles are synchronized, a gear changing "M" function (M41 to M44) has been executed or the programmed "S" involves a gear change (with automatic gear changer).
SOLUTION	First, cancel spindle synchronization (G78S).

1106 'Travel limits of spindle 2 exceeded'

Same as error 1100, but for the second spindle.

1107 'Spindle 2 locked'

Same as error 1101, but for the second spindle.

1108 'Following error of spindle 2 out of limit'

Same as error 1102, but for the second spindle.

1109 'Axis software limit overrun'

No explanation required.

1110 'Range of the X axis exceeded'
1111 'Range of the Y axis exceeded'
1112 'Range of the Z axis exceeded'
1113 'Range of the U axis exceeded'
1114 'Range of the V axis exceeded'
1115 'Range of the W axis exceeded'
1116 'Range of the A axis exceeded'
1117 'Range of the B axis exceeded'
1118 'Range of the C axis exceeded'

DETECTION	During execution.
CAUSE	A movement has been defined with parameters and the parameter value is greater than the maximum travel distance of the axis.
SOLUTION	Check the program history to make sure that this parameter does not have that value when it reaches the block where this movement is programmed.

1119 'The X axis cannot be synchronized'
1120 'The Y axis cannot be synchronized'
1121 'The Z axis cannot be synchronized'
1122 'The U axis cannot be synchronized'
1123 'The V axis cannot be synchronized'
1124 'The W axis cannot be synchronized'
1125 'The A axis cannot be synchronized'
1126 'The B axis cannot be synchronized'
1127 'The C axis cannot be synchronized'

DETECTION	During execution.
CAUSE	The possible causes are: 1. When trying to synchronize two axes from the PLC and one axis is already slaved to another one using the "G77" function. 2. When programming or trying to move an axis that is slaved to another one.

1128 'Maximum feedrate of the X axis exceeded'
1129 'Maximum feedrate of the Y axis exceeded'
1130 'Maximum feedrate of the Z axis exceeded'
1131 'Maximum feedrate of the U axis exceeded'
1132 'Maximum feedrate of the V axis exceeded'
1133 'Maximum feedrate of the W axis exceeded'
1134 'Maximum feedrate of the A axis exceeded'
1135 'Maximum feedrate of the B axis exceeded'
1136 'Maximum feedrate of the C axis exceeded'

DETECTION	During execution.
CAUSE	The resulting feedrate of one of the axes after applying an individual scaling factor exceeds the maximum value indicated by axis machine parameter MAXFEED (P42).

1137 'Wrong feedrate parameter of the X axis'
1138 'Wrong feedrate parameter of the Y axis'
1139 'Wrong feedrate parameter of the Z axis'
1140 'Wrong feedrate parameter of the U axis'
1141 'Wrong feedrate parameter of the V axis'
1142 'Wrong feedrate parameter of the W axis'
1143 'Wrong feedrate parameter of the A axis'
1144 'Wrong feedrate parameter of the B axis'
1145 'Wrong feedrate parameter of the C axis'

DETECTION	During execution.
CAUSE	"G00" programmed with parameter G00FEED(P38)=0 or "G1 F00" with axis parameter MAXFEED(P42) = 0.

1146 'X axis locked up'
1147 'Y axis locked up'
1148 'Z axis locked up'
1149 'U axis locked up'
1150 'V axis locked up'
1151 'W axis locked up'
1152 'A axis locked up'
1153 'B axis locked up'
1154 'C axis locked up'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	The CNC tries to output the command to the drive when the spindle input SERVO(n)ON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the axis parameter DWELL(P17) is not high enough.

1155 'Maximum X axis software exceeded'
1156 'Maximum Y axis software exceeded'
1157 'Maximum Z axis software exceeded'
1158 'Maximum U axis software exceeded'
1159 'Maximum V axis software exceeded'
1160 'Maximum W axis software exceeded'
1161 'Maximum A axis software exceeded'
1162 'Maximum B axis software exceeded'
1163 'Maximum C axis software exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	A coordinate has been programmed that is out of the limits defined by axis parameters LIMIT+(P5) and LIMIT-(P6).

1164 'Work zone 1 of the X axis exceeded'
1165 'Work zone 1 of the Y axis exceeded'
1166 'Work zone 1 of the Z axis exceeded'
1167 'Work zone 1 of the U axis exceeded'
1168 'Work zone 1 of the V axis exceeded'
1169 'Work zone 1 of the W axis exceeded'
1170 'Work zone 1 of the A axis exceeded'
1171 'Work zone 1 of the B axis exceeded'
1172 'Work zone 1 of the C axis exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located out of the work area 1 that is defined as "no exit" zone.
SOLUTION	In the program history, work zone 1 (defined with G20/G21) has been set as "no exit" zone " (G22 K1 S2). To cancel this work zone, program "G22 K1 S0"

1173 'Work zone 2 of the X axis exceeded'
1174 'Work zone 2 of the Y axis exceeded'
1175 'Work zone 2 of the Z axis exceeded'
1176 'Work zone 2 of the U axis exceeded'
1177 'Work zone 2 of the V axis exceeded'
1178 'Work zone 2 of the W axis exceeded'
1179 'Work zone 2 of the A axis exceeded'
1180 'Work zone 2 of the B axis exceeded'
1181 'Work zone 2 of the C axis exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located out of the work area 2 that is defined as "no exit" zone.
SOLUTION	In the program history, work zone 2 (defined with G20/G21) has been set as "no exit" zone " (G22 K2 S2). To cancel this work zone, program "G22 K2 S0"

1182 'X axis following error beyond limits'
1183 'Y axis following error beyond limits'
1184 'Z axis following error beyond limits'
1185 'U axis following error beyond limits'
1186 'V axis following error beyond limits'
1187 'W axis following error beyond limits'
1188 'A axis following error beyond limits'
1189 'B axis following error beyond limits'
1190 'C axis following error beyond limits'

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of its channel.

Besides this, it activates the external emergency output.

CAUSE The following error of the axis is greater than the values indicated by axis parameter MAXFLWE1(P21) or maxflwe2(P22). The possible causes for this error are:

Servo drive error
- Faulty drive.
- Enable signals missing.
- Power supply missing.
- Drive adjusted incorrectly.
- The velocity command signal is not received.

Motor error
- Faulty motor.
- Power cables.

Feedback failure
- Defective feedback.
- Defective feedback cable.

Mechanical failure
- Mechanical stiffness.
- Spindle mechanically locked.

CNC error
- Defective CNC.
- Parameters adjusted incorrectly.

1191 'Difference of following errors of the slaved X axis * tool large'
1192 'Difference of following errors of the slaved Y axis * tool large'
1193 'Difference of following errors of the slaved Z axis * tool large'
1194 'Difference of following errors of the slaved U axis * tool large'
1195 'Difference of following errors of the slaved V axis * tool large'
1196 'Difference of following errors of the slaved W axis * tool large'
1197 'Difference of following errors of the slaved A axis * tool large'
1198 'Difference of following errors of the slaved B axis * tool large'
1199 'Difference of following errors of the slaved C axis * tool large'

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of its channel.

Besides this, it activates the external emergency output.

CAUSE The "n" axis is electronically coupled to another one or is a slaved Gantry axis and the difference between the following errors of the "n" axis and the one it is coupled to is greater than the value set by the machine parameter for the "n" axis MAXCOUPE(P45).

1200 'X axis travel limits exceeded'
1201 'Y axis travel limits exceeded'
1202 'Z axis travel limits exceeded'
1203 'U axis travel limits exceeded'
1204 'V axis travel limits exceeded'
1205 'W axis travel limits exceeded'
1206 'A axis travel limits exceeded'
1207 'B axis travel limits exceeded'
1208 'C axis travel limits exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to exceed the physical travel limits. As a result, the PLC activates the axis mark "LIMIT+1" or "LIMIT-1".

1209 'X axis servo error'
1210 'Y axis servo error'
1211 'Z axis servo error'
1212 'U axis servo error'
1213 'V axis servo error'
1214 'W axis servo error'
1215 'A axis servo error'
1216 'B axis servo error'
1217 'C axis servo error'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The real feedrate of the axis, after the time indicated by axis parameter FBALTIME(P12), is below 50% or over 200% of the programmed value.

1218 'Work zone 3 of the X axis exceeded'
1219 'Work zone 3 of the Y axis exceeded'
1220 'Work zone 3 of the Z axis exceeded'
1221 'Work zone 3 of the U axis exceeded'
1222 'Work zone 3 of the V axis exceeded'
1223 'Work zone 3 of the W axis exceeded'
1224 'Work zone 3 of the A axis exceeded'
1225 'Work zone 3 of the B axis exceeded'
1226 'Work zone 3 of the C axis exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located out of the work area 3 that is defined as "no exit" zone.
SOLUTION	In the program history, work zone 3 (defined with G20/G21) has been set as "no exit" zone " (G22 K3 S2). To cancel this work zone, program "G22 K3 S0"

1227 'Wrong profile intersection in pocket with islands.'

DETECTION	During execution.
CAUSE	In the "Irregular pocket canned cycle with islands (G66)", there are two plane profiles that either have the starting point or a section in common.
SOLUTION	Define the profiles again. Two plane profiles cannot start at the same point or have sections in common.

1228 'Work zone 4 of the X axis exceeded'
1229 'Work zone 4 of the Y axis exceeded'
1230 'Work zone 4 of the Z axis exceeded'
1231 'Work zone 4 of the U axis exceeded'
1232 'Work zone 4 of the V axis exceeded'
1233 'Work zone 4 of the W axis exceeded'
1234 'Work zone 4 of the A axis exceeded'
1235 'Work zone 4 of the B axis exceeded'
1236 'Work zone 4 of the C axis exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located out of the work area 4 that is defined as "no exit" zone.
SOLUTION	In the program history, work zone 4 (defined with G20/G21) has been set as "no exit" zone " (G22 K4 S2). To cancel this work zone, program "G22 K4 S0"

1237 'Do not change the entry angle inside a thread'

DETECTION	During execution.
CAUSE	A thread joint has been defined and an entry angle "Q" has been programmed between two threads.
SOLUTION	When joining threads, only the first one may have an entry angle "Q".

1238 'Range of write-protected parameters. P297, P298'

DETECTION	During execution.
CAUSE	When trying to execute the function: "Definition of inclined plane (G49)", parameters P297 and P298 are write-protected with machine parameters ROPARMIN(P51) and ROPARMAX(P52).
SOLUTION	While defining an inclined plane, the CNC updates parameters P297 and P298. Therefore, these two parameters cannot be write-protected.

1239 'Point inside the forbidden zone 5.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located inside the work area 5 that is defined as "no entry" zone.
SOLUTION	In the program history, work zone 5 (defined with G20/G21) has been set as "no entry" zone " (G22 K5 S1). To cancel this work zone, program "G22 K5 S0"

1240 'Work zone 5 of the X axis exceeded'
1241 'Work zone 5 of the Y axis exceeded'
1242 'Work zone 5 of the Z axis exceeded'
1243 'Work zone 5 of the U axis exceeded'
1244 'Work zone 5 of the V axis exceeded'
1245 'Work zone 5 of the W axis exceeded'
1246 'Work zone 5 of the A axis exceeded'
1247 'Work zone 5 of the B axis exceeded'
1248 'Work zone 5 of the C axis exceeded'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	An attempt has been made to move an axis to a point located out of the work area 5 that is defined as "no exit" zone.
SOLUTION	In the program history, work zone 5 (defined with G20/G21) has been set as "no exit" zone " (G22 K5 S2). To cancel this work zone, program "G22 K5 S0"

1249 'Variable pitch thread programmed wrong'

DETECTION	During execution.
CAUSE	We are trying to make a variable-pitch thread with the following conditions: • The "K" increment is positive and equal to or greater than 2L. • The "K" increment is positive and with one of the calculated pitches, it exceeds the maximum feedrate (parameter MAXFEED) of one of the threading axis. • The "K" increment is negative and one of the calculated pitches 0 or negative.

1250 'The K value is too large in G34'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	The ratio between the initial and final pitches of the variable-pitch thread (G34) to be executed is greater than 32767.

1251 'Two variable-pitch threads cannot be joined in round corner'

DETECTION	During motionless simulation, except when graphics are active.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	To variable-pitch threads cannot be joined in round corner unless the second one is of the type: G34 ... L0 K0.

1252 'G5 G34 without a pitch is only allowed after a variable-pitch thread'

DETECTION	During motionless simulation, except when graphics are active.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	G34...L0 K0 (blending a variable pitch thread with another one with a fixed pitch) can only be programmed after a G34 with a K value other than ·0· and round corner (G05).

1253 'Retrace function unavailable'

No explanation required.

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of its channel.

1254 'Parameter restricted to OEM programs'

DETECTION — During execution.

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of its channel.

CAUSE — An attempt has been made to use an OEM parameter P2000-P2255 in a program that has no OEM permission.

SOLUTION — Use a non-OEM parameter.

1255 'Subroutine restricted to an OEM program'

DETECTION — During execution.

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of its channel.

CAUSE — An attempt has been made to use an OEM subroutine SUB10000-SUB20000 in a program that has no OEM permission.

SOLUTION — Use a general subroutine P0000-P9999.

FAGOR

·M· Model

Ref. 1310

HARDWARE ERRORS

2000 'External emergency activated.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel.
CAUSE	PLC input I1 is set to "0" (maybe the E-stop button) or the PLC mark M5000(/EMERGEN) is set to "0".
SOLUTION	Check at the PLC why the inputs are at "0". (Possible lack of power).

2001 'X axis feedback error'
2002 'Y axis feedback error'
2003 'Z axis feedback error'
2004 'U axis feedback error'
2005 'V axis feedback error'
2006 'W axis feedback error'
2007 'A axis feedback error'
2008 'B axis feedback error'
2009 'C axis feedback error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The CNC does not receive feedback signal from the axes.
SOLUTION	Check that the connections are properly made. NOTE: This error comes up on differential axes DIFFBACK(P9) =YES and sinusoidal axes SINMAGNI(P10) other than 0 when parameter FBACKAL(P11)=ON Setting parameter FBACKAL(P11)=OFF avoids this error, but this is only temporary solution.

2010 'Spindle feedback error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The CNC does not receive feedback signal from the spindle.
SOLUTION	Check that the connections are properly made. NOTE: This error comes up on differential axes DIFFBACK(P14)=YES when parameter FBACKAL(P15)=ON. Setting parameter FBACKAL(P15)=OFF avoids this error, but this is only temporary solution.

2011 'Maximum temperature exceeded'

DETECTION	Any time.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The CNC's internal temperature has been exceeded. The causes may be: • Electrical cabinet poorly ventilated. • Axis board with some defective component.
SOLUTION	Turn the CNC and wait until it cools off. If the error persists, a component of the board may be defective. In that case, replace the board. Contact the Service Department.

2012 'There is no voltage at the axis board'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	24V are missing at the output supply of the axis board. The fuse may be blown.
SOLUTION	Power the outputs of the axis board (24v). If the fuse is blown, replace it.

2013 'There is no voltage at the I/O 1 board.'
2014 'There is no voltage at the I/O 2 board.'
2015 'There is no voltage at the I/O 3 board.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	24V are missing at the output supply of the corresponding I/O board. The fuse may be blown.
SOLUTION	Power the outputs of the corresponding I/O board (24v). If the fuse is blown, replace it.

2016 'The PLC is not ready.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The PLC program is not running. These may be the probable causes: • The PLC program is missing. • WATCHDOG error. • The program has been interrupted from monitoring.
SOLUTION	Start the PLC program. (Restart the PLC).

2017 'CNC RAM memory error.'

DETECTION	While starting the CNC or during diagnoses.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A defect has been found in the CNC's RAM memory.
SOLUTION	Replace the CPU board. Contact the Service Department.

2018 'CNC's EPROM memory error.'

DETECTION	While starting the CNC or during diagnoses.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A defect has been found in the CNC's EPROM memory.
SOLUTION	Replace the EPROM. Contact the Service Department.

2019 'PLC's RAM memory error.'

DETECTION	While starting the CNC or during diagnoses.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A defect has been found in the PLC's RAM memory.
SOLUTION	Replace the PLC board. Contact the Service Department.

2020 'PLC's EPROM memory error.'

DETECTION	While starting the CNC or during diagnoses.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A defect has been found in the PLC's EPROM memory.
SOLUTION	Replace the EPROM. Contact the Service Department.

2021 'CNC's user RAM memory error.' Press any key.'

DETECTION	While starting the CNC or during diagnoses.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A defect has been found in the CNC's user RAM memory.
SOLUTION	Contact the Service Department.

2022 'CNC's system RAM memory error.' Press any key.'

DETECTION	While starting the CNC or during diagnoses.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A defect has been found in the CNC's system RAM memory.
SOLUTION	Contact the Service Department.

2023 'PLC's RAM memory error.' Press any key.'

DETECTION	While starting the CNC or during diagnoses.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A defect has been found in the PLC's RAM memory.
SOLUTION	Contact the Service Department.

2024 'There is no voltage at the tracing board'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	24V are missing at the output supply of the tracing board. The fuse may be blown.
SOLUTION	Power the outputs of the tracing board. If the fuse is blown, replace it.

2025 'Probe feedback error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The tracing probe is not connected or any of its cables is connected wrong.
SOLUTION	Check that the probe is properly connected.

2026 'Probe's maximum travel limit overrun.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The probe has exceeded the maximum deflection allowed by machine parameter.
SOLUTION	Decrease the feedrate and check that the probe has not been damaged.

2027 'SERCOS chip RAM memory error.' Press any key.'

DETECTION	While starting the CNC or during diagnoses.
CAUSE	A defect has been found in the SERCOS chip RAM memory.
SOLUTION	Replace the SERCOS board. Contact the Service Department.

2028 'SERCOS chip version error.' Press any key.'

DETECTION	During CNC start-up.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The SERCOS chip version is old.
SOLUTION	Replace the SERCOS chip. Contact the Service Department.

2029 'Feedback error at spindle 2.'

Same as error 2010, but for the second spindle.

2030 'Feedback over-current error.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	There has been a short-circuit or the feedback device has overheated.
SOLUTION	Check cables and connections.

2034 'There is no voltage at the I/O 4 board.'
2035 'There is no voltage at the I/O 5 board.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	24V are missing at the output supply of the corresponding I/O board. The fuse may be blown.
SOLUTION	Power the outputs of the corresponding I/O board (24v). If the fuse is blown, replace it.

2036 'The type of keyboard does not match the CNC model.'

DETECTION	During CNC start-up.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The keyboard identifier is unknown.
SOLUTION	Contact the Service Department.

2037 '24 V missing at the CPU-CNC module.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	At a modular CNC 8055: 24 V missing at the CPU module of the CNC. The fuse may be blown. At a CNC 8055i: 24V missing at CNC connector X2. The fuse may be blown.
SOLUTION	At a modular CNC 8055: Apply voltage to the CPU module of the CNC (24 V). If the fuse is blown, replace it. At a CNC 8055i: Apply voltage to CNC connector X2 (24 V). If the fuse is blown, replace it.

2041 'Unsupported LCD type.'

DETECTION	During CNC start-up.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The LCD identifier is unknown.
SOLUTION	Contact the Service Department.

2042 'It is recommended to lower the order of the frequency filter.'

DETECTION	On power-up or when pressing RESET after changing the value of the axis parameter or spindle parameter ORDER.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The order value of the FAGOR filter can cause overshooting.
SOLUTION	Decrease the value of the order of the filter: • a.m.p ORDER (P70). • s.m.p. ORDER (P67).

2043 'Parameters of the frequency filter set wrong.'

DETECTION	On power-up or when pressing RESET after changing the value of some parameter of the filters.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The parameters for the frequency or order of the filter are set wrong. If it is executed with these wrong parameter values, the filter will not be active.
SOLUTION	Check the values for the frequency and order of the filter.

2044 'TURBO board incompatible with version. Replace it with TURBO2.'

	No explanation required.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.

2045 'G51 with FAGOR filters is incompatible with general parameter IPOTIME.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	If g.m.p. IPOTIME (P73) is other than ·0·, even if FAGOR filters are active (bit 15 of g.m.p. LOOKATYP=1), when programming G51, the FAGOR filters do not start working.

2046 'G51 with FAGOR filters is incompatible with parameter SMOTIME.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	If any of the axes of the main channel has a.m.p. SMOTIME (P58) other than ·0·, even having FAGOR filters active with look-ahead (bit 15 of g.m.p. LOOKATYP=1), when programming G51, the FAGOR filters do not start working.

2047 'G51 with FAGOR filters is incompatible with parameter TYPE.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	If any of the axes of the main channel has a.m.p. TYPE (P71) other than ·2·, even having FAGOR filters active with look-ahead (bit 15 of g.m.p. LOOKATYP=1), when programming G51, the FAGOR filters do not start working.

2048 'Parameter TYPE=2 is incompatible with general parameter IPOTIME.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	If FAGOR filters are active (a.m.p. TYPE=2) and g.m.p. IPOTIME (P73) is other than ·0·, the FAGOR filters do not kick in (don't start working).

2049 'Parameter TYPE=2 is incompatible with general parameter SMOTIME.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	If FAGOR filters are active (a.m.p. TYPE=2) and g.m.p. SMOTIME (P58) is other than ·0·, the FAGOR filters do not kick in (don't start working).

2051 'Too many feedback pulses.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	Axis feedrate too high due to gear ratio.
SOLUTION	Check axis gear ratio.

2052 'Too much real feedback difference.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	The possible causes are: 1. The difference between the position value of the linear encoder connected to the CNC (second feedback) and that of the motor encoder (first feedback) is greater than the value of a.m.p. FBACKDIF (P100). 2. Feedback combination being active, the counting direction of the first and second feedback is not the same or the difference between the first and second feedback is greater than 838 mm.
SOLUTION	The solutions for each case are the following: 1. Check that the counting direction of both feedbacks is the same. Disable the feature that causes the error by setting a.m.p. FBACKDIF (P100) = 0. 2. Check that the counting direction of both feedbacks is the same.

2053 'Error at the CNC parameters.'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of its channel. Besides this, it activates the external emergency output.
CAUSE	A wrong value of some parameter has been detected on system start-up. The CNC indicates which parameter has the wrong value.
SOLUTION	Assign the right value to the parameter indicated by the CNC.

PLC ERRORS

3000 ‘ (PLC_ERR without description) ‘

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels.
CAUSE	Marks ERR1 to ERR64 have been set to "1".
SOLUTION	Check at the PLC why these marks are set to "1" and act accordingly.

3001 ‘WATCHDOG in the main module (PRG).’

DETECTION	Any time.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The possible causes are: 1. The execution of the PLC’s main program has exceeded the time set in PLC parameter WAGPRG(P0). 2. The program is in an endless loop.
SOLUTION	Increase the time of PLC parameter WAGPRG(P0) or increase the PLC speed. • Insert CPU TURBO. • Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3002 ‘WATCHDOG in the periodic module (PE).’

DETECTION	Any time.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The possible causes are: 1. The execution of the PLC’s periodic program has exceeded the time set in PLC parameter WAGPER(P1). 2. The program is in an endless loop.
SOLUTION	Increase the time of PLC parameter WAGPER(P1) or increase the PLC speed. • Insert CPU TURBO. • Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3003 ‘Division by zero at the PLC’

DETECTION	Any time.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	In the PLC program, there is a line whose execution implies a division by zero.
SOLUTION	When working with registers, that register may have already acquired a zero value. Check that the register does not reach the operation with that value.

3004 'PLC error ->'

DETECTION	Any time.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error has been detected on the PLC board.
SOLUTION	Replace the PLC board. Contact the Service Department.

3005 'Contacts debugging error'

DETECTION	While debugging the PLC program.
CAUSE	When debugging the PLC program to create the PLC program in contacts (ladder), the CNC finds an error in that program.
SOLUTION	Check if it has been properly compiled.

3006 'The PLC program does not exist'

No explanation required.

3007 'Configuration file corrupted'

DETECTION	At any time, while being on the <CONTACTS> screen.
CAUSE	An error has occurred in the configuration file.
SOLUTION	Exit the <CONTACTS> screen and go back into it.

3008 'PLC program too large'

DETECTION	At any time, while being on the <CONTACTS> screen.
CAUSE	The PLC program has exceeded the maximum size limit.

SERVO ERRORS

4000 'Sercos ring error'

DETECTION — During execution.

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE — SERCOS communication has been interrupted. It may be caused by an interruption in the connection ring (optical fiber disconnected or broken) or by a wrong configuration.

1. The identifying wheel does not match the sercosid.
2. Parameter P120 (SERSPD) does not match the transmission speed.
3. The drive version is incompatible with the CNC.
4. There is an error on the SERCOS board.
5. Different transmission speed (baudrate) at the drive and at the CNC.

A drive has been turned off and back on due to a power supply failure. When starting up again, it displays the error ***4027 'The drive has started up again'***

An attempt has been made to read or write an non-existent variable or too many variables in a drive through the fast channel.

SOLUTION — To check that the connection ring is not interrupted, check that the light goes through the optical fiber. If it is due to a wrong configuration, contact the Service Department.

If the error is due to the fast channel:

- Check that all the variables to be read or written through the fast channel actually exist.
- Save the SERCOS LOG into a file and see which axis causes the error.
- Set PLC machine parameters "SRD700 and SWR800" of that drive to "0".
- Reset the CNC and verify that no errors come up.
- Set the parameters one by one to the desired value until the failure occurs.
- When locating the parameter, look that variable up in the drive manual to verify that it exists in that version and it may be accessed. If so, the error may come up because it tries read or write too many variables in that drive.

4001 'Undefined class 1 error'

DETECTION — During execution.

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE — The drive has detected an error, but it cannot identify it.

SOLUTION — Contact the Service Department.

4002 'Overload (201...203)'
4003 'Overtemperature at the drive (107)'
4004 'Overtemperature at the motor (108)'
4005 'Overtemperature at the heatsink (106)'
4006 'Voltage control error (100...105)'
4007 'Feedback error (600...606)'
4008 'Error at the power bus (213...215)'
4009 'Overcurrent (212)'
4010 'Overvoltage at the power bus (304/306)'
4011 'Undervoltage at the power bus (307)'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive. The number in brackets indicates the standard error number of the drive. Refer to the drive manual for further information.
SOLUTION	These types of error come with the messages 4019, 4021, 4022 or 4023 that indicate in which axis or spindle drive the error came up. Refer to the drive manual to check the error (number in brackets) and act accordingly.

4012 'Drive error'
4013 'Position deviation too high'
4014 'Communications error'
4015 'Travel limit overrun'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4016 'Undefined class 1 error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The drive has detected an error, but it cannot identify it.
SOLUTION	Contact the Service Department.

4017 'Drive error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4018 'Error accessing a variable'

DETECTION During execution.

CAUSE An attempt has been made to read (or write) a SERCOS variable from the CNC and:

1. That variable does not exist.
2. The maximum/minimum values have been exceeded.
3. The SERCOS variable has a variable length.
4. An attempt has been made to write a read-only variable.

SOLUTION Check that the variable to be associated with an action is of the right type.

4019 'Drive error: Axis'

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE These messages come with errors 4002 - 4011. When one of the errors mentioned above occurs, they indicate on which axis they came up.

4020 'DRIBUSID parameter value error'

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE An error occurred at the drive.

SOLUTION Refer to the drive manual.

4021 'Spindle drive error :

4022 'Spindle-2 drive error :

4023 'Auxiliary spindle drive error'

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE These messages come with errors 4002 - 4011. When one of the errors mentioned above occurs, they indicate on which spindle they came up.

4024 'Error when searching home'.

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE The home search command of SERCOS has been executed incorrectly.

4025 'Loop time exceeded: Increase P72 (looptime)'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The time it takes to calculate the feedrate of the axis is greater than the cycle time established for transmission to the drive.
SOLUTION	Increase the value of general machine parameter LOOPTIME (P72). If the error persists, contact the Service Department.

4026 'SERCOS chip RAM memory error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
SOLUTION	Contact the service department to replace the SERCOS board.

4027 'The drive has started up again'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	A drive has been turned off and back on due to a power supply failure.

4028 'The light does not reach the CNC through the optic fiber'

DETECTION	On power-up.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The signal sent by the CNC through the optical fiber does not return to the CNC.
SOLUTION	Check the condition and installation of the fiber optic cables. Check that the light going "OUT" of the CNC is going through the drives and comes "IN"to the CNC. If the cables are OK, remove the drives from the ring until the error no longer comes up.

4029 'Communication with the drive cannot be established. No response'

DETECTION On power-up.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE A drive is not responding to the signal sent by the CNC due to one of these causes:

- The drive does not recognize the sercos board.
- The drive is locked up.
- The switch number has not been properly read.
- The SERCOS transmission speed has been set differently at the drives and at the CNC. General parameter SERSPD at the CNC and QP11 at the drives.

SOLUTION Save the SERCOS LOG into a file.

See the value of axis machine parameter SERCOSID of the axis causing the error.

Check that the ring contains a drive with the switch in that position.

Reset the drive because the drive only reads the switch on power-up.

Check that the CNC and the drives have the same transmission speed. General parameter SERSPD at the CNC and QP11 at the drives.

Check that the drive does not issue sercos board. To do that look at the display of the drive. If it shows hardware errors, change the drive's sercos board.

If there are no errors at that drive, set the switch of the drive to "1", reset it, set the CNC with a single Sercos axis and connect to the CNC. If it still issues the error, change the drive.

4030 'SERCON register writing error'

DETECTION During execution.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

SOLUTION Contact the Service Department.

4032 'Handshake error'

DETECTION During the operation of the CAN bus.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE The handshake bit has been lost. To verify that the communication is correct, it continuously checks a handshake bit between the CNC and the drives.

SOLUTION Check the cables, the connections, the line terminating resistors and the CAN boards (at the CNC and at the drive).

4033 'Cyclic message of the drive lost'

DETECTION During the operation of the CAN bus.

EFFECT It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE A message of the drive has been lost (it has not reached the CNC).

SOLUTION Check the cables, the connections, the line terminating resistors and the CAN boards (at the CNC and at the drive).

4034 'SID reading error'

DETECTION	During the operation of the CAN bus.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	From a CNC channel, an attempt has been made to read a non-existent variable of the drive.
SOLUTION	Check that the variable that it is trying to read exists at the drive.

4035 'SERCOS communication saturated. Increase P178 (SERCDEL1)'

DETECTION	On SERCOS bus power-up.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The maximum bus capacity has been exceeded.
SOLUTION	Increase the Sercos transmission delay using g.m.p. SERCDEL1 (P178).

4036 'SERCOS T3 > T4. Decrease P179 (SERCDEL2)'

DETECTION	On SERCOS bus power-up.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	g.m.p. The value of SERCDEL2 (P179) is wrong.
SOLUTION	Contact the Service Department.

4050 'ERROR 1: internal (Fatal error): Internal RAM check failed'
4051 'ERROR 2: internal (Fatal error): Internal program malfunctioning problem'
4052 'ERROR 3: Under-voltage of the power bus' There is no function'
4053 'ERROR 4: The emergency stop cannot stop the motor in the determined time period.
4054 'ERROR 5: Program code checksum error'
4055 'ERROR 6: Sercos board error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4056 'ERROR 100: Internal +5 V out of range'
4057 'ERROR 101: Internal -5 V out of range'
4058 'ERROR 102: Internal +8 V out of range'
4059 'ERROR 103: Internal -8 V out of range'
4060 'ERROR 104: Internal +18 V out of range'
4061 'ERROR 105: Internal -18 V out of range'
4062 'ERROR 106: Heatsink overheated'
4063 'ERROR 107: VeCon card overheated'
4064 'ERROR 108: Motor overheated'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4065 'ERROR 200: Overspeed'
4066 'ERROR 201: Motor overload'
4067 'ERROR 202: Drive overload'
4068 'ERROR 211: internal (Fatal error): DSP program execution error'
4069 'ERROR 212: Over-current'
4070 'ERROR 213: Undervoltage at the IGBT power driver'
4071 'ERROR 214: Short-circuit'
4072 'ERROR 215: Over-voltage at the power bus (Hard)'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4073 'ERROR 300: Heatsink of the power supply module overheated'
4074 'ERROR 301: Ballast circuit of the power supply module heatsink overheated'
4075 'ERROR 302: Short-circuit at the ballast circuit of the power supply module'
4076 'ERROR 303: Ballast circuit supply voltage out of range'
4077 'ERROR 304: Over-voltage at the power bus detected by the power supply module'
4078 'ERROR 305: Protocol error on the interface between the power supply module and the driver'
4079 'ERROR 306: Over-voltage at the power bus (Soft, trigger prior to hardware)'
4080 'ERROR 307: Under-voltage of the power bus'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4081 'ERROR 400: No SERCOS board detected'
4082 'ERROR 401: Internal SERCOS error'
4083 'ERROR 403: MST fault'
4084 'ERROR 404: MDT fault'
4085 'ERROR 405: Wrong phase (> 4)'
4086 'ERROR 406: Wrong phase up'
4087 'ERROR 407: Wrong phase down'
4088 'ERROR 408: Phase change without "ready" acknowledgement
4089 'ERROR 409: Change to an uninitialized phase'
4090 'ERROR 410: Noise resets sercon'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4091 'ERROR 500: Incoherent parameters'
4092 'ERROR 501: Parameter checksum error'
4093 'ERROR 502: Wrong parameter value'
4094 'ERROR 503: The table for default parameter values for each motor is wrong'
4095 'ERROR 504: Wrong parameter in SERCOS phase 2'
4096 'ERROR 505: Different RAM and Flash parameters'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4097 'ERROR 600: Communication error with the second feedback'
4098 'ERROR 601: Communication error with the rotor encoder'
4099 'ERROR 602: Motor feedback B signal saturation'
4100 'ERROR 603: Motor feedback A signal saturation'
4101 'ERROR 604: Saturation of A and/or B signal values'
4102 'ERROR 605: A and/or B signal values too low'
4103 'ERROR 606: Too much dispersion of the rotor sensor signals'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4104 'ERROR 700: RS232 board error'
4105 'ERROR 701: Internal: Wrong VeCon board identification'
4106 'ERROR 702: Expansion board identification error'
4107 'ERROR 703: I/O board identification error'
4108 'ERROR 704: Analog board identification error'
4109 'ERROR 705: Power board identification error'
4110 'ERROR 706: X3 encoder simulation board identification error'
4111 'ERROR 707: X4 motor feedback board identification error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4112 'ERROR 801: Encoder not found'
4113 'ERROR 802: Error when communicating with the encoder'
4114 'ERROR 803: Encoder not initialized'
4115 'ERROR 804: Defective encoder'
4116 'ERROR 805: No encoder has been detected on the motor'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4117 'ERROR 7: SERCON clock error'
4118 'ERROR 8: SERCON data error'
4119 'ERROR 203: Torque overload error'
4120 'ERROR 411: Telegram reception error'
4121 'ERROR 109: Over-voltage at digital inputs'
4122 'ERROR 110: Low heatsink temperature'
4123 'ERROR 607: Direct feedback A and/B signal saturation'
4124 'ERROR 608: A and/or B signal values of direct feedback too low'
4125 'ERROR 609: Temperature sensor error'
4126 'ERROR 150: Travel limits exceeded'
4127 'ERROR 152: Velocity command module exceeded'
4128 'ERROR 153: To much position command shift'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4129 'ERROR 154: Too much feedforward velocity command'
4130 'ERROR 155: Too much feedforward acceleration command'
4131 'ERROR 156: To much following error'
4132 'ERROR 157: Too much difference between the positions of the two feedbacks'
4133 'ERROR 250: 'Error when searching home'
4134 'ERROR 251: DriveControlledHoming command error'
4135 'ERROR 253: I0 not detected in 2 revolutions'
4136 'ERROR 254: Wrong reading of distance-coded reference marks (I0)'
4137 'ERROR 308: Over-current at energy return circuit'
4138 'ERROR 309: Short-circuit at the High Side IGBT'
4139 'ERROR 310: Low voltage at the driver of the High Side IGBT'
4140 'ERROR 311: Short-circuit at the Low Side of the IGBT'
4141 'ERROR 312: Low voltage at the driver of the Low Side IGBT'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4142 'ERROR 313: Consumption over-current'
4143 'ERROR 314: I2t protection of the crowbar resistor'
4144 'ERROR 806: Error when searching home with Sincoder'
4145 'ERROR 807: Wrong C and D feedback signals'
4146 'ERROR 412: Delayed synchronism message'
4147 'ERROR 413: Handshake error at the drive'
4148 'ERROR 9: Loss of non-volatile data'
4149 'ERROR 10: Damaged non-volatile data'
4150 'ERROR 31: Internal error'
4151 'ERROR 506: .MOT file not found'
4152 'ERROR 507: Motor not found in .MOT file'
4153 'ERROR 508: List of wrong parameters in phase 4'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4154 'ERROR 808: No encoder has been detected in Feedback2'
4155 'ERROR 809: Error when communicating with the Feedback2 encoder'
4156 'ERROR 810: Feedback2 encoder not initialized'
4157 'ERROR 811: Defective Feedback2 encoder'
4158 'ERROR 255: Error when changing feedbacks after executing the PC150 command'
4159 'ERROR 812: Feedback2 encoder detected'
4160 'ERROR 206: Velocity command too high'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4176 'ERROR 205: The motor has no voltage for the required torque'
4177 'ERROR 315: The power supply has not started up correctly'
4178 'ERROR 610: Wrong absolute signals'
4179 'ERROR 611: The axis moves on power-up and the position cannot be read'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4180 'ERROR 256: Erroneous home signal distance per feedback turn'
4181 'ERROR 160: Emergency ramp following error'
4182 'ERROR 111: Undertemperature of the motor'
4183 'ERROR 509: The activation code that activates the "open" option at the drive has not been entered.
4184 'ERROR 818: Error on the absolute track'
4185 'ERROR 819: CPU error'
4186 'ERROR 820: Error at the adjustment potentiometers'
4187 'ERROR 821: Image capturing sensor (CCD) error'
4188 'ERROR 822: Supply voltage out of range'
4189 'ERROR 823: Parameter error'
4190 'ERROR 158: Excessive position deviation when estimating the electrical position with the GC7 command'
4191 'ERROR 159: Wrong counting direction when executing the GC3 command'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4192 'ERROR 216: Internal'
4193 'ERROR 316: It took too long chargingn the DC bus of a compact drive'
4194 'ERROR 813: 'Error when initializing the electrical position'
4195 'ERROR 814: Wrong absolute signals'
4196 'ERROR 815: The axis is moving on drive power-up and the absolute position cannot be read correctly'
4197 'ERROR 816: Unstable C and D motor feedback signals'
4198 'ERROR 817: CRC check error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4200 'ERROR 9001: Parameter checksum error'
4201 'ERROR 9002: AD circuit damaged'
4202 'ERROR 9003: Speed overflow''
4203 'ERROR 9004: Over-current'
4204 'ERROR 9005: Position counter overflow'
4205 'ERROR 9006: Error pulse overflow (Pn504)'
4206 'ERROR 9007: Electronic changer configured wrong or pulse frequency overflow'
4207 'ERROR 9008: First current detection channel damaged'
4208 'ERROR 9009: Second current detection channel damaged'
4209 'ERROR 9010: Incremental encoder damaged'
4210 'ERROR 9012: Over-current'
4211 'ERROR 9013: Servomotor over-voltage'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4212 'ERROR 9014: Servomotor under-voltage'
4213 'ERROR 9015: Crowbar resistor error.
4214 'ERROR 9016: Regenerating circuit error'
4215 'ERROR 9017: Resolver error:
4216 'ERROR 9018: IGBT temperature alarm'
4217 'ERROR 9020: Phase not connected at power supply'
4218 'ERROR 9021: Instantaneous power supply missing'
4219 'ERROR 9041: Reserved'
4220 'ERROR 9042: Servomotor type error'
4221 'ERROR 9043: Servodrive type error'
4222 'ERROR 9044: Reserved'
4223 'ERROR 9045: Multi-turn absolute encoder data error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4224 'ERROR 9046: Multi-turn absolute encoder data error'
4225 'ERROR 9047: Battery voltage under 2.5V'
4226 'ERROR 9048: Battery voltage under 3.1V'
4227 'ERROR 9050: Serial encoder communication error'
4228 'ERROR 9051: Speed alarm on absolute encoder'
4229 'ERROR 9052: Absolute encoder damaged'
4230 'ERROR 9053: Serial encoder calculation error'
4231 'ERROR 9054: Parity bit error or serial encoder end bit error'
4232 'ERROR 9055: Serial encoder communication data error'
4233 'ERROR 9056: Serial encoder end bit error'
4234 'ERROR 9058: Serial encoder EEPROM data empty'
4235 'ERROR 9059: Serial encoder EEPROM data format error'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

4236 'ERROR 9060: Communication module not detected'
4237 'ERROR 9061: CPU or communication module error'
4238 'ERROR 9062: Servodrive not receiving periodic data from communication module'
4239 'ERROR 9063: Communication module not receiving response from servodrive'
4240 'ERROR 9064: Bus and communication module disconnected'
4241 'ERROR 9066: Wrong CAN communication'
4242 'ERROR 9067: Timeout of the master station'
4243 'ERROR 9069: The synchronism signal monitoring cycle is longer than what it was set for'

DETECTION	During execution.
EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	An error occurred at the drive.
SOLUTION	Refer to the drive manual.

CAN ERRORS

5003 'Application error'

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE — Internal CANopen error.

SOLUTION — Contact the Service Department.

5004 'CAN bus error'

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE — The error type is indicated with a code:

- 2 Transmission queue full, the message cannot be sent.
- 128 Bus Off, the bus has been deactivated due to too many errors.
- 129 CAN warning, there are more than 96 errors at the bus, step prior to the buss off error.
- 130 Loss of message received or too many messages received. Usually due to wrong speed for the cable length.
- 131 The CNC has switched to an inoperative state in the bus (internal).

SOLUTION — The solution for each cause is:

- 2 Check the connection between the CNC and first node.
- 128 Check cables and connections.
- 129 Check cables and connections.
- 130 Check machine parameter IOCANSPE (P88).
- 131 Check cables and connections.

5005 'Presence control error detected by the CNC'

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE — The CNC detects that the node has reset itself or is connected wrong.

SOLUTION — Check cables and connections.

5006 'Error because the node has been reset'

EFFECT — It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·.

If it is in execution, it interrupts the execution of the part program of the CNC of all channels.

Besides this, it activates the external emergency output.

CAUSE — The node has been reset due to a power supply failure.

SOLUTION — Check the power supply voltage at the indicated node, the ground connection and the load of the outputs.

5007 'Corrected error'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	It is activated when an error situation disappears and shows whether there are any more left. If there is none, it resets the node connections.

5022 'Internal software'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Internal node software error.
SOLUTION	Access the Status screen \ Can \ Versions and reload the software.

5027 'Communication'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Node communication error.
SOLUTION	Contact the Service Department.

5028 'Lost messages'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The node has lost messages.
SOLUTION	Check cables and connections.

5029 'Presence control error detected by the node'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The presence control done by the CNC node has failed.
SOLUTION	Check cables and connections.

5030 'Protocol error'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The node has received a message that it cannot interpret.
SOLUTION	Contact the Service Department.

5031 'PDO not processed due to its length'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The node has received a process message whose length does not match.
SOLUTION	Contact the Service Department.

5032 'PDO too long'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	The node has received a process message longer than the one programmed.
SOLUTION	Contact the Service Department.

5036 'Output over-current'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Excessive consumption (over current) has been detected in the outputs of the indicated node. As a precaution, the system deactivates all the outputs of this module setting them to zero volts.
SOLUTION	Check the consumption and possible short-circuits at the outputs of the module.

5037 'Power supply voltage error'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	A power supply failure has been detected at the indicated node, it has no power or it is under +24V.
SOLUTION	Check the supply voltage at the outputs and the consumption of the module's supply voltage.

5039 'No response (identifier).'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Error in the node configuration.
SOLUTION	Check cables and connections.

5041 'It has no digital inputs'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Error in the node configuration.
SOLUTION	Check cables and connections.

5045 'Writing the transmission mode TPDO1.'
5046 'Writing the reception mode RPDO1.'
5047 'Writing the reception mode RPDO2.'
5048 'Writing - Life Time Factor'
5049 'Writing - Guard Time'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Error in the node configuration.
SOLUTION	Check cables and connections.

5051 'PT100 broken or not connected'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
SOLUTION	Check that the PT100 is connected and not broken.

5052 'Too many errors at the bus'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Error in the node configuration.
SOLUTION	Check cables and connections.

5055 'Writing the reception mode RPDO3'
5058 'Writing the reception mode RPDO4'
5061 'Writing the transmission mode TPDO2.'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Error in the node configuration.
SOLUTION	Check cables and connections.

5062 'It could not disable the PT100 1'
5063 'It could not disable the PT100 2'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Error in the node configuration.

5064 'It could not enable the analog inputs'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Error in the node configuration.

5065 'No communication with CAN drives'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	CAN communication has been interrupted.
SOLUTION	Check cables and connections.

5066 'Error reading parameter SRR700, SWR800 SID'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Using PLC parameters 700/800, a CAN variable has been requested that does not exist at the drive.
SOLUTION	Check that the variables that it is trying to read exist at the drive.

5067 'Too many PLC parameters SRR700.'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Too many parameters are requested from the drive.
SOLUTION	Set some PLC parameters SRR700 (P28-P67) to 0 to request fewer variables from the drive.

5068 'Too many PLC parameters SWR800.'

EFFECT	It stops the movement of the axes and the spindle, eliminating all the enable signals and canceling all the analog outputs of the CNC. When detected from the position loop, it opens the position loop and sets the LOPEN mark to ·1·. If it is in execution, it interrupts the execution of the part program of the CNC of all channels. Besides this, it activates the external emergency output.
CAUSE	Too many parameters are requested from the drive.
SOLUTION	Set some PLC parameters SWR800 (P68-P87) to 0 to request fewer variables from the drive.

FAGOR

·M· Model

Ref. 1310

TABLE DATA ERRORS

'CHECKSUM ERROR GENERAL PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR 2nd SPINDLE PARAM. Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR AUX. SPINDLE PARAM. Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR SERIAL LINE 1 PARAM. Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR SERIAL LINE 2 PARAM. Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR HD/ETHERNET PARAM. Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR USER PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR OEM PARAM. Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR PLC PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR ZERO OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR PASSWORD TABLE Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR AXIS * PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:TOOL TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR TOOL OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR MAGAZINE TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR M FUNCTION TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR LEADSCREW * TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR CROSS COMP. TABLE * Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR GEOMETRY TABLE Load CARD A? (ENTER/ESC)'

DETECTION	During CNC start-up.
CAUSE	Certain table data has been lost (possible RAM error) and there is a table saved in CARD A.
SOLUTION	Pressing [ENTER] copies the table saved in CARD A to RAM memory. If the error persists, contact the service department.

'ERROR:GENERAL PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AUX. SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-1 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'CHECKSUM ERROR:HD/ETHERNET PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:USER PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:OEM PARAMETERS Initialize? (ENTER/ESC)'
'ERROR:PLC PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:ZERO OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CODE TABLE CHECKSUM Reset? (ENTER/ESC)'

'ERROR:AXIS PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:TOOL TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL MAGAZINE TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:M FUNCTION TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AXIS LEADSCREW TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CROSS COMP. TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR: GEOMETRY TABLE CHECKSUM Reset? (ENTER/ESC)'

DETECTION	During CNC start-up.
CAUSE	Certain table data has been lost (possible RAM error) and there is no table saved in CARD A.
SOLUTION	Pressing [ENTER] loads the tables with CNC's default values. If the error persists, contact the Service Department.

'CHECKSUM ERROR GENERAL PARAMETERS Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR SPINDLE PARAMETERS Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR SPINDLE 2 PARAMETERS Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR AUX SPINDLE PARAMETERS Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR SERIAL LINE 1 PARAM. Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR SERIAL LINE 2 PARAM. Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR HD/ETHERNET PARAM. Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR USER PARAMETERS Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR OEM PARAMETERS Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR PLC PARAMETERS Load HARD DISK? (ENTER/ESC)'

'CHECKSUM ERROR ZERO OFFSET TABLE Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR ZERO CODE TABLE Load HARD DISK? (ENTER/ESC)'

'ERROR:* AXIS PARAMETER CHECKSUM HARD DISK? (ENTER/ESC)'

'CHECKSUM ERROR:TOOL TABLE Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR:TOOL OFFSET TABLE Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR:TOOL MAGAZINE TABLE Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR:M FUNCTION TABLE Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR * AXIS LEADSCREW TABLE Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR CROSS COMP. TABLE * Load HARD DISK? (ENTER/ESC)'
'CHECKSUM ERROR GEOMETRY TABLE Load HARD DISK? (ENTER/ESC)'

DETECTION	During CNC start-up.
CAUSE	Certain table data has been lost (possible RAM error) and there is a table saved in HARD DISK.
SOLUTION	Pressing [ENTER] copies into RAM the table saved in the HARD DISK. If the error persists, contact the Service Department.

'Wrong * leadscrew table. Press key'

DETECTION During CNC start-up.

CAUSE There is some erroneous data in the parameters of the leadscrew compensation table.

SOLUTION The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).
- The error difference between consecutive points cannot be greater than the distance between them.

'Wrong * cross compensation table. Press key'

DETECTION During CNC start-up.

CAUSE There is some erroneous data in the parameters of the cross compensation table.

SOLUTION The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).

'Incorrect cross compensation table parameters'

DETECTION During CNC start-up.

CAUSE The parameters indicating the axes involved in the cross compensation are defined wrong.

SOLUTION Maybe a nonexistent axis has been defined or the affected axis (to be compensated) and the one affecting it are the same.

'Wrong axis or spindle parameters sercosid'

DETECTION During CNC start-up.

CAUSE The servosid parameters have not been entered correctly.

SOLUTION The rules of sercosid parameters are:

- They must begin with number 1.
- They must be consecutive.
- They cannot be repeated.

FAGOR

·M· Model

Ref. 1310

ERRORS OF THE MC WORK MODE

9001 'CENTER PUNCHING: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9002 'CENTER PUNCHING: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9003 'CENTER PUNCHING: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9004 'CENTER PUNCHING: P=0'

DETECTION	During execution.
CAUSE	The center punching depth "P" has not been defined.
SOLUTION	The center punching depth "P" must be other than zero.

9005 'CENTER PUNCHING: ø=0'

DETECTION	During execution.
CAUSE	The point diameter "ø" has not been defined.
SOLUTION	The point diameter "ø" must be positive and other than zero.

9006 'CENTER PUNCHING: α=0'

DETECTION	During execution.
CAUSE	The angle of the tip of the drill bit has not been «α»
SOLUTION	The angle of the tip of the drill bit «α» must be positive and other than zero.

9007 'DRILLING 1: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9008 'DRILLING 1: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9009 'DRILLING 1: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9010 'DRILLING 1: P=0'

DETECTION	During execution.
CAUSE	The drilling depth "P" has not been defined.
SOLUTION	The drilling depth "P" must be other than zero.

9011 'DRILLING 2: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9012 'DRILLING 2: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9013 'DRILLING 2: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9014 'DRILLING 2: P=0'

DETECTION	During execution.
CAUSE	The drilling depth "P" has not been defined.
SOLUTION	The drilling depth "P" must be other than zero.

9015 'DRILLING 2: B=0'

DETECTION	During execution.
CAUSE	The withdrawal distance "B" after each penetration has not been defined.
SOLUTION	The distance "B" it withdraws after each penetration must be other than zero.

9016 'THREADING: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9017 'THREADING: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9018 'THREADING: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9019 'THREADING: P=0'

DETECTION	During execution.
CAUSE	The tapping depth "P" has not been defined.
SOLUTION	The tapping depth "P" must be other than zero.

9020 'REAMING: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9021 'REAMING: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9022 'REAMING: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9023 'REAMING: P=0'

DETECTION	During execution.
CAUSE	The reaming depth "P" has not been defined.
SOLUTION	The reaming depth "P" must be other than zero.

9024 'BORING 1: F=0'

DETECTION During execution.
CAUSE A feedrate "F" has been defined with a wrong value.
SOLUTION Program a positive feedrate "F" other than zero.

9025 'BORING 1: S=0'

DETECTION During execution.
CAUSE A spindle speed "S" has been defined with a wrong value.
SOLUTION Program a positive spindle speed "S" other than zero.

9026 'BORING 1: T=0'

DETECTION During execution.
CAUSE The tool number "T" has not been defined.
SOLUTION The tool number "T" must be other than zero.

9027 'BORING 1: P=0'

DETECTION During execution.
CAUSE The boring depth "P" has not been defined.
SOLUTION The boring depth "P" must be other than zero.

9028 'DRILLING 3: F=0'

DETECTION During execution.
CAUSE A feedrate "F" has been defined with a wrong value.
SOLUTION Program a positive feedrate "F" other than zero.

9029 'DRILLING 3: S=0'

DETECTION During execution.
CAUSE A spindle speed "S" has been defined with a wrong value.
SOLUTION Program a positive spindle speed "S" other than zero.

9030 'DRILLING 3: T=0'

DETECTION During execution.
CAUSE The tool number "T" has not been defined.
SOLUTION The tool number "T" must be other than zero.

9031 'DRILLING 3: P=0'

DETECTION During execution.
CAUSE The drilling depth "P" has not been defined.
SOLUTION The drilling depth "P" must be other than zero.

9032 'BORING 2: F=0'

DETECTION During execution.
CAUSE A feedrate "F" has been defined with a wrong value.
SOLUTION Program a positive feedrate "F" other than zero.

9033 'BORING 2: S=0'

DETECTION During execution.
CAUSE A spindle speed "S" has been defined with a wrong value.
SOLUTION Program a positive spindle speed "S" other than zero.

9034 'BORING 2: T=0'

DETECTION During execution.
CAUSE The tool number "T" has not been defined.
SOLUTION The tool number "T" must be other than zero.

9035 'BORING 2: P=0'

DETECTION During execution.
CAUSE The boring depth "P" has not been defined.
SOLUTION The boring depth "P" must be other than zero.

9036 'RECTANGULAR POCKET 1: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9037 'RECTANGULAR POCKET 1: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9038 'RECTANGULAR POCKET 1: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9039 'RECTANGULAR POCKET 1: P=0'

DETECTION	During execution.
CAUSE	The pocket depth "P" has not been defined.
SOLUTION	The pocket depth "P" must be other than zero.

9040 'RECTANGULAR POCKET 1: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9041 'RECTANGULAR POCKET 1: Tool diameter larger than pocket'

DETECTION	During execution.
CAUSE	The tool diameter is larger than one of the pocket's "H" or "L" dimensions.
SOLUTION	Choose a tool of smaller diameter to mill the pocket.

9042 'RECTANGULAR POCKET 1: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9043 'RECTANGULAR POCKET 2: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9044 'RECTANGULAR POCKET 2: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9045 'RECTANGULAR POCKET 2: P=0'

DETECTION	During execution.
CAUSE	The pocket depth "P" has not been defined.
SOLUTION	The pocket depth "P" must be other than zero.

9046 'RECTANGULAR POCKET 2: Wrong penetration angle'

DETECTION	During execution.
CAUSE	A penetration angle smaller than 0º and greater than 90º has been programmed.
SOLUTION	Program a penetration angle "β" and "θ" between 0º and 90º.

9047 'RECTANGULAR POCKET 2: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9048 'RECTANGULAR POCKET 2: Tool diameter larger than pocket'

DETECTION	During execution.
CAUSE	The tool diameter is larger than one of the pocket's "H" or "L" dimensions.
SOLUTION	Choose a tool of smaller diameter to mill the pocket.

9049 'RECTANGULAR POCKET 2: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9050 'CIRCULAR POCKET 1: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9051 'CIRCULAR POCKET 1: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9052 'CIRCULAR POCKET 1: P=0'

DETECTION	During execution.
CAUSE	The pocket depth "P" has not been defined.
SOLUTION	The pocket depth "P" must be other than zero.

9053 'CIRCULAR POCKET 1: Wrong penetration angle'

DETECTION	During execution.
CAUSE	A penetration angle smaller than 0º and greater than 90º has been programmed.
SOLUTION	Program a penetration angle "β" and "θ" between 0º and 90º.

9054 'CIRCULAR POCKET 1: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9055 'CIRCULAR POCKET 1: Tool diameter larger than pocket'

DETECTION	During execution.
CAUSE	The tool radius is greater than the pocket radius "R".
SOLUTION	Choose a tool of smaller diameter to mill the pocket.

9056 'CIRCULAR POCKET 1: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9057 'CIRCULAR POCKET 2: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9058 'CIRCULAR POCKET 2: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9059 'CIRCULAR POCKET 2: P=0'

DETECTION	During execution.
CAUSE	The pocket depth "P" has not been defined.
SOLUTION	The pocket depth "P" must be other than zero.

9060 'CIRCULAR POCKET 2: Wrong penetration angle'

DETECTION	During execution.
CAUSE	A penetration angle smaller than 0º and greater than 90º has been programmed.
SOLUTION	Program a penetration angle "β" and "θ" between 0º and 90º.

9061 'CIRCULAR POCKET 2: Tool radius larger than Ri'

DETECTION	During execution.
CAUSE	A tool has been selected with a radius greater than Ri (inside radius).
SOLUTION	Select a tool with a smaller diameter.

9062 'CIRCULAR POCKET 2: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9063 'CIRCULAR POCKET 2: Tool diameter larger than pocket'

DETECTION	During execution.
CAUSE	The tool radius is greater than the pocket radius "R".
SOLUTION	Choose a tool of smaller diameter to mill the pocket.

9064 'CIRCULAR POCKET 2: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9065 'CIRCULAR POCKET 2: Ri > Re'

DETECTION	During execution.
CAUSE	An inside radius (Ri) has been programmed greater than the outside (Re).

9066 'RECTANGULAR BOSS: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9067 'RECTANGULAR BOSS: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9068 'RECTANGULAR BOSS: P=0'

DETECTION	During execution.
CAUSE	The boss depth "P" has not been defined.
SOLUTION	The boss height "P" must be other than zero.

9069 'RECTANGULAR BOSS: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9070 'RECTANGULAR BOSS: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9071 'CIRCULAR BOSS: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9072 'CIRCULAR BOSS: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9073 'CIRCULAR BOSS: P=0'

DETECTION	During execution.
CAUSE	The boss depth "P" has not been defined.
SOLUTION	The boss height "P" must be other than zero.

9074 'CIRCULAR BOSS: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9075 'CIRCULAR BOSS: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9076 'PROFILE POCKET: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9077 'PROFILE POCKET: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9078 'PROFILE POCKET: P=0'

DETECTION	During execution.
CAUSE	The pocket depth "P" has not been defined.
SOLUTION	The pocket depth "P" must be other than zero.

9079 'PROFILE POCKET: Wrong penetration angle'

DETECTION	During execution.
CAUSE	A penetration angle smaller than 0º and greater than 90º has been programmed.
SOLUTION	Program a penetration angle "β" and "θ" between 0º and 90º.

9080 'PROFILE POCKET: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9081 'PROFILE POCKET: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9082 '3D PROFILE POCKET: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9083 '3D PROFILE POCKET: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9084 '3D PROFILE POCKET: P=0'

DETECTION	During execution.
CAUSE	The pocket depth "P" has not been defined.
SOLUTION	The pocket depth "P" must be other than zero.

9085 '3D PROFILE POCKET: Wrong penetration angle'

DETECTION	During execution.
CAUSE	A penetration angle smaller than 0º and greater than 90º has been programmed.
SOLUTION	Program a penetration angle "β" and "θ" between 0º and 90º.

9086 '3D PROFILE POCKET: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9087 '3D PROFILE POCKET: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9088 '3D PROFILE POCKET: Tool radius FINISHING STOCK less than R'

DETECTION	During execution.
CAUSE	The radius of the finishing tool is smaller than R (finishing tool tip radius).
SOLUTION	Select a tool with a larger diameter.

9089 'SURFACE MILLING: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9090 'SURFACE MILLING: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9091 'SURFACE MILLING: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9092 'SURFACE MILLING: P=0'

DETECTION	During execution.
CAUSE	The depth "P" of the surface milling has not been defined.
SOLUTION	The surface milling depth "P" must be other than zero.

9093 'PROFILE MILLING 1: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9094 'PROFILE MILLING 1: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9095 'PROFILE MILLING 1: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9096 'PROFILE MILLING 1: P=0'

DETECTION	During execution.
CAUSE	The milling depth "P" has not been defined.
SOLUTION	The milling depth "P" must be other than zero.

9097 'PROFILE MILLING 1: Null profile'

DETECTION	During execution.
CAUSE	The profile to be machined has not been defined.
SOLUTION	The profile must consist of at least two points besides the entry one and the exit one.

9098 'PROFILE MILLING 2: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9099 'PROFILE MILLING 2: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9100 'PROFILE MILLING 2: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9101 'PROFILE MILLING 2: P=0'

DETECTION	During execution.
CAUSE	The milling depth "P" has not been defined.
SOLUTION	The milling depth "P" must be other than zero.

9102 'SLOT MILLING: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9103 'SLOT MILLING: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9104 'SLOT MILLING: P=0'

DETECTION	During execution.
CAUSE	The milling depth "P" has not been defined.
SOLUTION	The milling depth "P" must be other than zero.

9105 'SLOT MILLING: L=0'

DETECTION	During execution.
CAUSE	The slot length "L" has not been defined.
SOLUTION	The length of the slot "L" must be other than zero.

9106 'SLOT MILLING: Tool diameter smaller than Δ'

DETECTION	During execution.
CAUSE	The programmed milling step "Δ" is larger than the tool diameter.
SOLUTION	Program a milling step "Δ" smaller than the tool diameter or choose a tool of larger diameter.

9107 'SLOT MILLING: Tool diameter larger than slot'

DETECTION	During execution.
CAUSE	The diameter of the tool is larger than the programmed slot.
SOLUTION	Select a tool with a smaller diameter.

9108 'SLOT MILLING: Tool diameter FINISHING STOCK less than δ'

DETECTION	During execution.
CAUSE	The programmed finishing stock "δ" is larger than the tool diameter.
SOLUTION	Program a finishing stock "δ" smaller than the tool diameter or choose a tool of larger diameter.

9109 'LINEAR POSITIONING: Wrong I'

DETECTION	During execution.
CAUSE	The "I" distance between positionings has the wrong value and it does not allow an integer number of machining operations.
SOLUTION	Check that the data entered is correct.

9110 'ARC POSITIONING: β not valid'

DETECTION	During execution.
CAUSE	The distance "β" between positionings has the wrong value and it does not allow an integer number of machining operations.
SOLUTION	Check that the data entered is correct.

9111 'RECTANGULAR PATTERN POSITIONING: Ix/Iy not valid'

DETECTION	During execution.
CAUSE	One of the distances "Ix/Iy" between positionings has the wrong value and it does not allow an integer number of machining operations.
SOLUTION	Check that the data entered is correct.

9112 'GRID PATTERN POSITIONING: Ix/Iy not valid'

DETECTION	During execution.
CAUSE	One of the distances "Ix/Iy" between positionings has the wrong value and it does not allow an integer number of machining operations.
SOLUTION	Check that the data entered is correct.

9113 'PROBE 1: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9114 'PROBE 1: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9115 'PROBE 1: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9116 'RECTANGULAR POCKET 2: δ not valid'

DETECTION	During execution.
CAUSE	The finishing stock "δ" is too large. It is impossible to leave this residual stock using the programmed tool.
SOLUTION	The finishing stock "δ" must be smaller.

9117 'PART CENTERING: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9118 'PART CENTERING: L=0'

DETECTION	During execution.
CAUSE	The part length "L" has not been defined.
SOLUTION	The part length "L" must be other than zero.

9119 'PART CENTERING: H=0'

DETECTION	During execution.
CAUSE	The part width ""H" has not been defined.
SOLUTION	The part width "H" must be other than zero.

9120 'PART CENTERING: Δz=0'

DETECTION	During execution.
CAUSE	The distance for the probe to go up in Z for its movements over the part has not been defined.
SOLUTION	The "Δz" distance must be other than zero.

9121 'PART CENTERING: Dr=0'

DETECTION	During execution.
CAUSE	The withdrawal distance, after the part searching probing movement for measuring has not been defined.
SOLUTION	The "Dr" distance must be other than zero.

9122 'PART CENTERING: Fs=0'

DETECTION	During execution.
CAUSE	The part searching probing feedrate "Fs" has not been defined.
SOLUTION	Program a positive feedrate "Fs" other than zero.

9123 'PART CENTERING: F=0'

DETECTION	During execution.
CAUSE	The probing feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9124 'PROBE CALIBRATION: F=0'

DETECTION	During execution.
CAUSE	The measuring feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9125 'PROBE CALIBRATION: Fs=0'

DETECTION	During execution.
CAUSE	The probe searching feedrate "Fs" has not been defined.
SOLUTION	Program a positive feedrate "Fs" other than zero.

9126 'PROBE CALIBRATION: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9127 'PROBE CALIBRATION: Ds=0'

DETECTION	During execution.
CAUSE	The probe approach distance "Ds" has not been defined.
SOLUTION	Program an approach distance "Ds" other than zero.

9128 'PROBE CALIBRATION: Dr=0'

DETECTION	During execution.
CAUSE	The tool withdrawal distance "Dr" has not been defined.
SOLUTION	Program a withdrawal distance "Dr" other than zero.

9129 'THREAD MILLING: F=0'

DETECTION	During execution.
CAUSE	A feedrate "F" has been defined with a wrong value.
SOLUTION	Program a positive feedrate "F" other than zero.

9130 'THREAD MILLING: S=0'

DETECTION	During execution.
CAUSE	A spindle speed "S" has been defined with a wrong value.
SOLUTION	Program a positive spindle speed "S" other than zero.

9131 'THREAD MILLING: T=0'

DETECTION	During execution.
CAUSE	The tool number "T" has not been defined.
SOLUTION	The tool number "T" must be other than zero.

9132 'THREAD MILLING: P=0'

DETECTION	During execution.
CAUSE	The milling depth "P" has not been defined.
SOLUTION	The milling depth "P" must be other than zero.

9133 'THREAD MILLING: Nominal diameter=0'

DETECTION	During execution.
CAUSE	The nominal diameter of the thread "ϕ" has not been defined.
SOLUTION	The nominal diameter of the thread "ϕ" must be other than zero.

9134 'THREAD MILLING: K=0'

DETECTION	During execution.
CAUSE	The thread depth "K" (distance from the crest to the root) has not been defined.
SOLUTION	The thread depth "K" must be other than zero.

9135 'THREAD MILLING: B=0'

DETECTION	During execution.
CAUSE	The thread pitch "B" has not been defined.
SOLUTION	The thread pitch "B" must be other than zero.

9136 'THREAD MILLING: Ds=0'

DETECTION	During execution.
CAUSE	The approach distance to the thread start "Ds" has not been defined.
SOLUTION	The "Dr" approach distance must be other than zero.

9137 'THREAD MILLING: Edges=0'

DETECTION During execution.
CAUSE The number of edges of the cutter "N" has not been defined
SOLUTION The number of edges of the cutter "N" must be other than zero.

9138 'BORE MILLING: F=0'

DETECTION During execution.
CAUSE A feedrate "F" has been defined with a wrong value.
SOLUTION Program a positive feedrate "F" other than zero.

9139 'BORE MILLING: S=0'

DETECTION During execution.
CAUSE A spindle speed "S" has been defined with a wrong value.
SOLUTION Program a positive spindle speed "S" other than zero.

9140 'BORE MILLING: T=0'

DETECTION During execution.
CAUSE The tool number "T" has not been defined.
SOLUTION The tool number "T" must be other than zero.

9141 'BORE MILLING: P=0'

DETECTION During execution.
CAUSE The milling depth "P" has not been defined.
SOLUTION The milling depth "P" must be other than zero.

9142 'BORE MILLING: Diameter=0'

DETECTION During execution.
CAUSE The drill bit diameter "ϕ" has not been defined.
SOLUTION The diameter of the drill bit "ϕ" must be other than zero.

9143 'BORE MILLING: B=0'

DETECTION During execution.
CAUSE The helical penetration pitch "B" has not been defined.
SOLUTION The helical penetration pitch "B" must be other than zero.

FAGOR

·M· Model

Ref. 1310