CNC 8055 ·T·

Error solution

Ref. 1310

FAGOR ⌐

FAGOR AUTOMATION

FAGOR

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

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

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

I N D E X

FAGOR ⬤

·T· Model

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. In the "Pattern repeat canned cycle (G66)", "Roughing canned cycle along the X axis (G68)" or "Roughing canned cycle along the Z axis (G69)" when parameter "S" (beginning of 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. 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. 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 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.
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 or G15 C.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An attempt has been made to execute an operation on the "C" axis, but the axis is not active.
SOLUTION	In order to operate with the "C" axis, it must be activated first using the "G15" function.

FAGOR

·**T**· Model

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.

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 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 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.

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 profile, a high level block has been programmed.
SOLUTION	Profiles 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.

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.

FAGOR ▰

·T· Model

Ref. 1310

0029 'G84-85: X Z Q R C [D L M F H] I K.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the canned cycle for "Turning of curved sections (G84)" or for "Facing curved sections (G85)" 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. A parameter has been programmed which does not match the calling format.
SOLUTION	The following parameters must be programmed in this cycle:

    X-Z Profile starting point
    Q-R Profile end point
    C   Depth of pass.
    I-K   Distance from the starting point to the arc center.

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

0030 'G86-87: X Z Q R I B [D L] C [J A].'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the canned cycle for "longitudinal threadcutting (G86)" or for "face threadcutting (G87)" 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. A parameter has been programmed which does not match the calling format.
SOLUTION	The following parameters must be programmed in this cycle:

    X-Z Starting point of the thread.
    Q-R End point of the thread.
    I   Thread depth.
    B   Depth of pass.
    C   Thread pitch.

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

0031 'G88-G98: X Z Q R [C D K].'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the canned cycle for "grooving along X (G88)" or "grooving along Z (G89)" 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. A parameter has been programmed which does not match the calling format.
SOLUTION	The following parameters must be programmed in this cycle:

    X-Z Starting point of the groove.
    Q-R End point of the groove.

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

0032 'G66: X Z I C [A L M H] S E.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the "Pattern repeat canned cycle with islands (G66)" 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. A parameter has been programmed which does not match the calling format.
SOLUTION	The following parameters must be programmed in this cycle:

    X-Z Profile starting point
    I   Remaining stock.
    C   Depth of pass.
    S   Block where the profile geometry description begins.
    E   Block where the profile geometry description ends.

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

0033 'G68-G69: X Z C [D L M F H] S E .'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the canned cycle for "roughing along X (G68)" or "roughing along Z (G69)" 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. A parameter has been programmed which does not match the calling format.

SOLUTION   The following parameters must be programmed in this cycle:

    X-Z   Profile starting point
    C   Depth of pass.
    S   Block where the profile geometry description begins.
    E   Block where the profile geometry description ends.

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

0034 'G81-G82: X Z Q R C [D L M F H].'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the canned cycle for "Turning of straight sections (G81)" or for "Facing straight sections (G82)" 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. A parameter has been programmed which does not match the calling format.

SOLUTION   The following parameters must be programmed in this cycle:

    X-Z   Profile starting point
    Q-R   Profile end point
    C   Depth of pass.

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

0035 'G83: X Z I B [D K H C L R].'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters have been programmed wrong in the "Axial drilling/tapping cycle (G83)". 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. A parameter has been programmed which does not match the calling format.

SOLUTION   The following parameters must be programmed in this cycle:

    X-Z   Machining position.
    I   Machining depth.
    B   Type of operation.

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

FAGOR ⬎

·T· Model

0036 'G60-G61: X Z I B Q A J [D K H C] S [L R].'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the canned cycle for "face drilling or tapping (G60)" or for "longitudinal drilling or tapping (G61)" 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. A parameter has been programmed which does not match the calling format.

SOLUTION	The following parameters must be programmed in this cycle:

    X-Z   Machining position.
    I   Machining depth.
    B   Type of operation.
    Q   Angular position of the first machining operation.
    A   Angular step between machining operations.
    J   Number of machining operations.
    S   Live tool turning speed and direction.

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

0037 'G62-G63: X Z L I Q A J [D] F S.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	The parameters of the canned cycle for "longitudinal slot milling (G62)" or "face slot milling (G62)" 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.

SOLUTION	The following parameters must be programmed in this cycle:

    X-Z   Slot position.
    L   Slot length.
    I   Slot depth.
    Q   Angular position of the first slot.
    A   Angular step between slots.
    J   Number of slots.
    F   Feedrate.
    S   Live tool turning speed and direction.

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

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	During the execution in programs transmitted via DNC.
CAUSE	An attempt has been made to execute a block syntactically incorrect (G1 X20K-15)
SOLUTION	Correct the syntax of the block.

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	A block has been edited whose execution involves the movement of a nonexistent axis.
SOLUTION	Check that the axis name being edited is correct.

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.

FAGOR

·T· Model

Ref. 1310

·**13**·

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")

FAGOR

·T· Model

Ref. 1310

·15·

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	In the "Pattern repeat canned cycle (G66)", "Roughing canned cycle along the X axis (G68)" or "Roughing canned cycle along the Z axis (G69)", it has been programmed that the profiles are located in another program (parameter "Q"), but the program does not exist.
SOLUTION	Parameter "Q" defines which program contains the profile definitions of the cycles. 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 tables.
CAUSE	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	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.

FAGOR ⬛

·T· Model

Ref. 1310

·16·

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.

FAGOR

·**T· Model**

0083 'Incomplete operation.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	While programming in high level language, the "IF" instruction has been edited without the condition between brackets.
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>.

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).

FAGOR 🔴

·**T**· **Model**

Ref. 1310

·**19**·

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.

FAGOR ⬂

·T· Model

Ref. 1310

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.

FAGOR

·**T**· Model

Ref. 1310

·**21**·

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 X Z R F 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.

FAGOR ⬥

·T· Model

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".

FAGOR

·T· Model

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.

FAGOR

·**T**· **Model**

Ref. 1310

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.

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.

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".

FAGOR ⬤

·T· Model

Ref. 1310

·26·

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.

FAGOR

·T· Model

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.

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.

0186 'The "C" axis does not exist.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An attempt has been made to activate the "C" axis, but the machine does not have this feature.

0187 'G66, G68, G69 are not allowed when machining with the "C" axis.'

DETECTION	During execution.
CAUSE	An attempt has been made to execute a "G66", "G68" or "G69" canned cycle while the "C" axis is active.
SOLUTION	Cancel the "C" axis to work with these canned cycles.

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.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An attempt has been made to start the live tool "M45 S...", but the machine does not have this feature.

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, 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 or Z axis is slaved or synchronized.
SOLUTION	To execute the "PROBE"¨ instruction, the X-Z axes must not be slaved or synchronized. To unslave the axes, program "G78".

0196 'Axes X 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 or Z axis is missing.
SOLUTION	To operate with the "PROBE" instruction, the X, Z axes must be defined.

0197 'G15 must be programmed before the "C" axis.'

DETECTION	While editing at the CNC or while executing a program transmitted via DNC.
CAUSE	An attempt has been made to execute an operation on the "C" axis, but the axis is not active.
SOLUTION	In order to operate with the "C" axis, it must be activated first using the "G15" function.

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.

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.

FAGOR

·T· **Model**

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).

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.

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".

FAGOR

·T· Model

0229 'Program maximum Z'
0230 'Program inside R'
0231 'Program outside R'

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.

FAGOR ◢

·T· Model

Ref. 1310

·32·

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
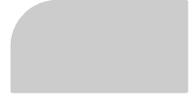
·T· Model

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.

FAGOR ⟍

·T· Model

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.

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 'No radius has been programmed for G15.'

DETECTION	During execution.
CAUSE	The Z-C plane has been selected as a new work plane, but the radius of the cylinder to be machined has not been defined.
SOLUTION	In order to work in the Z-C plane, first define the radius of the cylinder on which to machine using function "G15 R..."

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.

FAGOR

·T· Model

Ref. 1310

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.

1028 'Do not switch axes over or back while G15 is active'

DETECTION	During execution.
CAUSE	An attempt has been made to switch over to an axis or back (G28/G29) while function "G15" was 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.

FAGOR ▰

·T· Model

Ref. 1310

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).

1037 'No "S" has been programmed for G96.'

DETECTION	During execution.
CAUSE	The "Constant Surface Speed (G96)" function has been programmed, but no cutting speed has been defined, a previous one does not exist or no spindle gear is selected.
SOLUTION	In order to work at constant surface speed (G96), a cutting speed "S" must be already programmed and a spindle gear must be active.

1038 'The spindle has not been oriented'

DETECTION	During execution.
CAUSE	An attempt has been made to execute a threading cycle (G86 or G87) as a thread repair without already having oriented the active spindle (main or secondary).

1039 'No "F" has been programmed in G94.'

DETECTION	During execution.
CAUSE	An attempt has been made to execute a live tool cycle (G60, G61, G62 and G63) and there is feedrate selected in G94 (mm/min).
SOLUTION	First, select the feedrate "F" in mm/min (G94).

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.

1042 'Wrong parameter value in canned cycle'

DETECTION	During execution.
CAUSE	When defining a canned cycle, a parameter has been defined with the wrong value. Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).
SOLUTION	Correct the definition of parameters:

- In the "Pattern repeat cycle":
  - Parameter "C" only takes positive values greater than zero.
  - Parameter "A" only admits values of 0 or 1.
  - Parameter "J" only takes positive values greater than zero.
- In the canned cycle for "roughing along the Z axis" or "roughing along the X axis", parameter "C" only takes positive values greater than zero.
- In the " Axial drilling/tapping canned cycle":
  - Parameter "I" only admits values other than zero.
  - Parameter "B" only takes positive values or zero.
- In the canned cycle for "facing curved sections" or "turning curved sections", parameter "C" only takes positive values greater than zero.
- In the "Face threading canned cycle" or "Longitudinal threading canned cycle", parameter "I", "B", "E" or "C" has been defined with a zero value.
- In the canned cycle for "grooving along the Z axis" or "grooving along the X axis", parameter "C" only takes positive values greater than zero.
- In the canned cycle for "Face drilling/tapping" or "Longitudinal drilling/tapping":
  - Parameter "I" only admits values other than zero.
  - Parameter "B" only takes positive values or zero.
  - Parameter "J" only takes positive values greater than zero.
- In the canned cycle for "slot milling on the face" or "slot milling on the side", the slot dimension cannot be zero and parameters "I" and "J" only take positive values greater than zero.

1043 'Wrong tool for programmed profile.'

DETECTION	During execution.
CAUSE	The selected tool cannot machine anywhere on the profile.
SOLUTION	Choose a more appropriate tool to machine the profile.

1044 'A profile has been programmed that intersects itself.'

DETECTION	During execution.
CAUSE	In the set of profiles, there is one that intersects itself.
SOLUTION	Check the definition of the profiles. A profile cannot intersect itself.

1045 'Wrong cutter geometry angle.'

DETECTION	During execution.
CAUSE	The cutter's geometry angles have been assigned a wrong value.
SOLUTION	Correct the tool geometry data.

1046 'Wrong tool position before the canned cycle'

DETECTION	During execution.
CAUSE	The canned cycle calling point is defined wrong.
SOLUTION	The canned cycle calling point must be off the part and at a distance greater than the one defined as finishing stock on both axes. (Cycles that do not have a finishing stock will use the safety distance).

1047 'Wrong location code in canned cycle'

DETECTION	During execution.
CAUSE	The location code (shape) of the tool is not the right one.
SOLUTION	Choose a tool with the right location code (shape).

FAGOR

·T· Model

Ref. 1310

1048 'Wrong cutter width'

DETECTION	During execution.
CAUSE	A grooving operation has been defined with a cutter of zero width.
SOLUTION	Check the definition of the cutter dimensions (NOSEW). The cutter width must be other than zero.

1049 'Incompatible tool position and location code in profile cycle.

DETECTION	During execution.
CAUSE	The canned cycle calling point is defined wrong or the tool location code (shape) is the right one to execute the machining operation.
SOLUTION	The canned cycle calling point must be off the part and at a distance greater than the one defined as finishing stock on both axes. Besides, the tool's location code must allow executing the profile without running into the part.

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 variable.'

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 4.

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. In the "Pattern repeat canned cycle (G66)", "Roughing canned cycle along the X axis (G68)" or "Roughing canned cycle along the Z axis (G69)" a profile definition has been programmed, but one of the two data defining the beginning "S" or end "E" 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.

FAGOR

·**T**· Model

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.

FAGOR

·T· Model

Ref. 1310

·42·

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·.

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 XZ 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 Z R" format. 2. When programming using the "G02/G03 X Z 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 possible causes are: 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.

FAGOR

·T· Model

Ref. 1310

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 upon the "Pattern repeat canned cycle (G66)", "Roughing canned cycle along X (G68)" or "Roughing canned cycle along Z (G69)" is because in the set of blocks that define the profiles, one of the cases mentioned earlier occurs.
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.
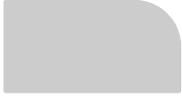
1087 'Circle with zero radius'

DETECTION	During execution.
CAUSE	The possible causes are:

1. When programming an arc using "G02/G03 X Z I K", 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.

FAGOR 🟥

·T· Model

Ref. 1310

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).

FAGOR

·T· Model

Ref. 1310

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).

FAGOR ⟍

·T· Model

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).

FAGOR

·T· Model

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"

FAGOR ⤳

·T· Model

Ref. 1310

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).

FAGOR ⬤

·T· Model

Ref. 1310

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 one programmed.

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"

FAGOR ⬢

·T· Model

Ref. 1310

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	Parameters P297 and P298 are write-protected by means of machine parameters ROPARMIN(P51) and ROPARMAX(P52).

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"

FAGOR

·T· Model

Ref. 1310

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 ⬤

·T· Model

Ref. 1310

·55·

·T· Model

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.

FAGOR ❘

·T· Model

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).

FAGOR ➡

·T· Model

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.

FAGOR

·T· **Model**

Ref. 1310

·59·

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.

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.

FAGOR ⬤

·**T**· **Model**

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.

FAGOR 🔴

·T· Model

Ref. 1310

·61·

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.

FAGOR 〰

·T· Model

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.

FAGOR ❚

·T· Model

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.

FAGOR

·T· Model

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.

FAGOR ➡

·T· Model

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.

FAGOR ⬈

·T· Model

Ref. 1310

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.

FAGOR

·T· Model

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.

FAGOR ⬳

·T· Model

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).

FAGOR 🔴

·**T**· **Model**

Ref. 1310

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 established time frame'

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.

FAGOR

·T· Model

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.

FAGOR ◢

·T· **Model**

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.

FAGOR ⬤

·T· Model

Ref. 1310

·74·

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.

FAGOR ◢

·T· Model

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.

FAGOR ⟱

·T· Model

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.

FAGOR 🔴

·T· Model

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.

FAGOR ⬎

·T· Model

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.

FAGOR

·T· Model

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.

FAGOR 🔴

·T· Model

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.

FAGOR

·**T**· **Model**

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.

FAGOR ⬤

·T· Model

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.

FAGOR ➔

·T· Model

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 ▰

·T· **Model**

Ref. 1310

FAGOR

·T· Model

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.

FAGOR ⬧

·T· Model

'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.

·T· Model

'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

·T· Model

Ref. 1310

FAGOR

·T· Model

ERRORS OF THE TC WORK MODE

9001 'Cycle without ROUGHING or FINISHING'

DETECTION	While executing a drilling, facing, taper turning, rounding or profile cycle.
CAUSE	Neither the roughing nor the finishing tools have been seleted.
SOLUTION	Select the roughing tool (If T=0, there is no roughing) and the finishing tool (if T=0, there is no finishing).

9002 'ROUGHING: Value of Δ=0'

DETECTION	While executing a drilling, facing, taper turning, rounding or profile cycle.
CAUSE	The tapping depth "Δ" has not been defined.
SOLUTION	Program a cutting pass greater than zero.

9003 'ROUGHING: Value of F=0

DETECTION	While executing a drilling, facing, taper turning, rounding or profile cycle.
CAUSE	No roughing feedrate "F" has been selected.
SOLUTION	Program a positive feedrate "F" other than zero.

9004 'ROUGHING: Value of S=0'

DETECTION	While executing a drilling, facing, taper turning, rounding or profile cycle.
CAUSE	No roughing spindle speed "S" has been selected.
SOLUTION	Program a positive spindle speed "S" other than zero.

9005 'FINISHING: Value of F=0

DETECTION	While executing a drilling, facing, taper turning, rounding or profile cycle.
CAUSE	No finishing feedrate "F" has been selected.
SOLUTION	Program a positive feedrate "F" other than zero.

9006 'FINISHING: Value of S=0'

DETECTION	While executing a drilling, facing, taper turning, rounding or profile cycle.
CAUSE	No finishing spindle speed "S" has been selected.
SOLUTION	Program a positive spindle speed "S" other than zero.

9007 'GEOMETRY: Value of Zi=Zf

DETECTION	While executing a turning, facing, taper turning or grooving cycle.
CAUSE	The Z coordinate of the starting and end points are the same.
SOLUTION	The Z coordinate of the starting and end points must be different.

9008 'GEOMETRY: Value of X=Ø'

DETECTION	While executing a turning, facing, taper turning or grooving cycle.
CAUSE	The coordinates of the initial and final diameter are the same.
SOLUTION	The X coordinate of the starting and end points must be different from that of the final diameter.

9009 'FINISHING: Impossible finishing stock'

DETECTION	While executing a turning, facing or grooving cycle.
CAUSE	The finishing stock is greater than the total machining depth.
SOLUTION	The finishing stock must be smaller than the total machining depth.

9010 'GEOMETRY: Wrong angle value'

DETECTION	While executing a taper turning cycle.
CAUSE	The taper angle is smaller than 0º or greater than 90º.
SOLUTION	The taper angle must be between 0º and 90º.

FAGOR

·**T· Model**

Ref. 1310

9011 'GEOMETRY: Wrong radius value'

DETECTION	While executing a rounding cycle.
CAUSE	No rounding radius has been defined.
SOLUTION	Program a rounding radius other than zero.

9012 'THREADING: Value of T=0'

DETECTION	While executing a threading cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9013 'THREADING: Value of S=0'

DETECTION	While executing a threading cycle.
CAUSE	The spindle speed "S" has not been defined.
SOLUTION	Program a positive spindle speed "S" other than zero.

9014 'THREADING: Value of P=0'

DETECTION	While executing a threading cycle.
CAUSE	The thread pitch has not been defined.
SOLUTION	Program a thread pitch greater than zero.

9015 'THREADING: Value of H=0'

DETECTION	While executing a threading cycle.
CAUSE	The thread depth has not been defined.
SOLUTION	Program a thread depth other than zero.

9016 'THREADING: Value of $\Delta=0$'

DETECTION	While executing a threading cycle.
CAUSE	The tapping depth "Δ" has not been defined.
SOLUTION	Program a cutting pass greater than zero.

9017 'THREADING: Value of $\sigma >$ (Zf-Zi)'

DETECTION	While executing a threading cycle.
CAUSE	The distance to the end of the thread is greater than the length of the thread.
SOLUTION	Program a distance to the end of the thread smaller than the length of the thread.

9018 'GEOMETRY: Value of Xi=Xf'

DETECTION	While executing a threading cycle.
CAUSE	The X coordinate of the starting and end points are the same.
SOLUTION	The X coordinate of the starting and end points must be different.

9019 'THREADING: Value of $\sigma >$ (Xf-Xi)'

DETECTION	While executing a threading cycle.
CAUSE	The distance to the end of the thread is greater than the length of the thread.
SOLUTION	Program a distance to the end of the thread smaller than the length of the thread.

9020 'ROUGHING: Wrong location (shape) code'

DETECTION	While executing a grooving cycle.
CAUSE	No groove roughing can be done with the selected location code.
SOLUTION	Select a tool with the right location code.

9021 'FINISHING: Wrong location (shape) code'

DETECTION	While executing a grooving cycle.
CAUSE	No groove finishing can be done with the selected location code.
SOLUTION	Select a tool with the right location code.

9022 'GEOMETRY: Null profile'

DETECTION	While executing a level-1 profile cycle.
CAUSE	No profile has been defined because the starting point and the end point are the same.
SOLUTION	The starting point and the end point of the profile must be different.

9023 'DRILLING: Value of T=0'

DETECTION	While executing a drilling or multiple drilling cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9024 'DRILLING: Value of Δ=0'

DETECTION	While executing a drilling or multiple drilling cycle.
CAUSE	The tapping depth "Δ" has not been defined.
SOLUTION	Program a cutting pass greater than zero.

9025 'DRILLING: Value of L=0'

DETECTION	While executing a drilling or multiple drilling cycle.
CAUSE	The drilling depth has not been defined.
SOLUTION	Program a drilling depth other than zero.

9026 'DRILLING: Value of F=0

DETECTION	While executing a drilling or multiple drilling cycle.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9027 'DRILLING: Value of S=0'

DETECTION	While executing a drilling or multiple drilling cycle.
CAUSE	The spindle speed "S" has not been defined.
SOLUTION	Program a positive spindle speed "S" other than zero.

9028 'TAPPING: Value of T=0'

DETECTION	While executing a tapping or multiple tapping cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9029 'TAPPING: Value of L=0'

DETECTION	While executing a tapping or multiple tapping cycle.
CAUSE	The threading depth has not been defined.
SOLUTION	Program a tapping depth other than zero.

9030 'TAPPING: Value of F=0

DETECTION	While executing a tapping or multiple tapping cycle.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9031 'TAPPING: Value of S=0'

DETECTION	While executing a tapping or multiple tapping cycle.
CAUSE	The spindle speed "S" has not been defined.
SOLUTION	Program a positive spindle speed "S" other than zero.

9032 'GEOMETRY: The final diameter is not external'

DETECTION	While executing a turning or grooving cycle.
CAUSE	An outside turning has been defined, but the final diameter is larger than that of the starting diameter.
SOLUTION	In an outside turning, the final diameter must be smaller than the starting one.

FAGOR 🔴

·T· Model

Ref. 1310

·**93**·

9033 'GEOMETRY: The final diameter is not internal'

DETECTION	While executing a turning or grooving cycle.
CAUSE	An inside turning has been defined, but the final diameter is smaller than that of the starting diameter.
SOLUTION	In an inside turning, the final diameter must be larger than the starting one.

9034 'GEOMETRY: Wrong quadrant'

DETECTION	While executing a taper turning cycle.
CAUSE	A taper has been defined in the wrong quadrant.
SOLUTION	Select the right quadrant using the relevant icon.

9035 'A negative safety distance is not allowed in this cycle'

DETECTION	While executing a taper turning, rounding or profiling cycle.
CAUSE	A negative safety distance has been defined.
SOLUTION	In taper canned cycles, the safety distance must be positive.

9036 'ROUGHING: Wrong tool for GROOVING.'

DETECTION	While executing a grooving cycle.
CAUSE	The geometry of the selected tool is not right for groove roughing.
SOLUTION	Select a tool with the right geometry.

9037 'FINISHING: Wrong tool for GROOVING.'

DETECTION	While executing a grooving cycle.
CAUSE	The geometry of the selected tool is not right for groove finishing.
SOLUTION	Select a tool with the right geometry.

9038 'GEOMETRY: Wrong angle for GROOVING.'

DETECTION	While executing a grooving cycle.
CAUSE	The angle of the groove walls is lower than 0º or higher than 90º.
SOLUTION	The angle of the groove walls must be between 0º and 90º.
	'GEOMETRY: The sides of the groove intersect each other.'

9039 'GEOMETRY: The sides of the groove intersect each other.'

DETECTION	During execution.
CAUSE	The two walls of the groove intersect each other.
SOLUTION	Check the cycle data. The walls of the groove cannot intersect each other.

9040 'MULTIPLE CYCLE: Value of β=0'

DETECTION	While executing a multiple drilling, multiple tapping or slot multiple milling cycle.
CAUSE	The angular step between machining has not been defined.
SOLUTION	Program an angular step other than zero.

9041 'MULTIPLE CYCLE: Value of N=0'

DETECTION	While executing a multiple drilling, multiple tapping or slot multiple milling cycle.
CAUSE	The number of machining operations has not been defined.
SOLUTION	The minimum number of machining operations is 1.

9042 'MULTIPLE SLOT MILLING CYCLE: Value of T=0'

DETECTION	While executing a multiple slot milling cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9043 'MULTIPLE SLOT MILLING CYCLE: Value of F=0

DETECTION	While executing a multiple slot milling cycle.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9044 'MULTIPLE SLOT MILLING CYCLE: Value of S=0'

DETECTION	While executing a multiple slot milling cycle.
CAUSE	No live tool turning speed "St" has been defined.
SOLUTION	Program a positive turning speed "St" other than zero.

9045 'MULTIPLE SLOT MILLING CYCLE: Value of I=0'

DETECTION	While executing a multiple slot milling cycle.
CAUSE	The slot milling depth has not been defined.
SOLUTION	Program a slot milling depth other than zero.

9046 'MULTIPLE SLOT MILLING CYCLE: Value of L=0'

DETECTION	While executing a multiple slot milling cycle.
CAUSE	The slot milling length has not been defined.
SOLUTION	The slot length must be other than zero.

9048 'C AXIS PROFILE CYCLE: Value of T=0'

DETECTION	While executing a profile cycle on the C axis.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9049 'C AXIS PROFILE CYCLE: Value of Δ=0'

DETECTION	While executing a profile cycle on the C axis.
CAUSE	The tapping depth "Δ" has not been defined.
SOLUTION	Program a cutting pass greater than zero.

9050 'C AXIS PROFILE CYCLE: Value of I=0'

DETECTION	While executing a profile cycle on the C axis.
CAUSE	The total machining depth has not been defined.
SOLUTION	Program a machining depth other than zero.

9051 'C AXIS PROFILE CYCLE: Value of F=0

DETECTION	While executing a profile cycle on the C axis.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9052 'C AXIS PROFILE CYCLE: Value of S=0'

DETECTION	While executing a profile cycle on the C axis.
CAUSE	No live tool turning speed "St" has been defined.
SOLUTION	Program a positive turning speed "St" other than zero.

9053 'Value of Smax=0'

DETECTION	While executing any cycle.
CAUSE	The maximum spindle speed "Smax" has not been defined.
SOLUTION	Program a maximum positive spindle speed "Smax" other than zero.

9054 'DRILLING: Value of KΔ>1'

DETECTION	While executing a drilling or multiple drilling cycle.
CAUSE	The gear ratio "KΔ" with a value greater than 1.
SOLUTION	Program a gear ratio "KΔ" with a value between 0 and 1.

9055 'GEOMETRY: The tool is too big for the groove.'

DETECTION	While executing a grooving cycle.
CAUSE	The diameter of the selected tool is larger than the width of the programmed groove.
SOLUTION	Select another tool with a smaller diameter.

FAGOR ➔

·T· **Model**

Ref. 1310

9056 'GEOMETRY: Value of Z=R.'

DETECTION	While executing a grooving cycle.
CAUSE	When milling slots with Z axis penetration (levels 2 and 4), a slot has been programmed without penetration.
SOLUTION	Program the external coordinate "Z" and the depth coordinate "R" with different values.

9057 'CUTOFF: Value of T=0'

DETECTION	While executing a cut-off cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9058 'CUTOFF: Value of F=0

DETECTION	While executing a cut-off cycle.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9059 'CUTOFF: Value of S=0'

DETECTION	While executing a cut-off cycle.
CAUSE	The spindle speed "S" has not been defined.
SOLUTION	Program a positive spindle speed "S" other than zero.

9060 'CUTOFF: Value of Fr=0'

DETECTION	While executing a cut-off cycle.
CAUSE	The cut-off feedrate "Fr" has not been defined.
SOLUTION	Program a positive cut-off feedrate "Fr" other than zero.

9061 'CUTOFF: Value of Fr > F'

DETECTION	While executing a cut-off cycle.
CAUSE	The cut-off feedrate "Fr" is higher than the feedrate "F".
SOLUTION	Program a smaller cut-off feedrate "Fr" than the feedrate "F".

9062 'DRILLING: Value of minimum increment Δ'

DETECTION	While executing a drilling cycle.
CAUSE	The minimum increment is larger than the drilling peck "Δ"
SOLUTION	Program a smaller minimum increment than the drilling peck "Δ"

9063 'TOOL CALIBRATION: Value of T=0'

DETECTION	While executing a tool calibration cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9064 'PROBE CALIBRATION: Value of T=0'

DETECTION	While executing a probe calibration cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9065 'RECTANGULAR POCKET: F=0'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9066 'RECTANGULAR POCKET: S=0'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	No live tool turning speed "S" has been defined.
SOLUTION	Program a positive spindle speed "S" other than zero.

9067 'RECTANGULAR POCKET: P=0'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	The total pocket depth has not been defined.
SOLUTION	Program a total pocket depth greater than zero.

9068 'RECTANGULAR POCKET: Wrong penetration angle'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	The defined penetration angle is wrong.
SOLUTION	Program a penetration angle greater than 0º and smaller than or equal to 90º.

9069 'RECTANGULAR POCKET: Tool diameter smaller than Δ'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	The tool diameter is smaller than the maximum roughing pass "Δ".
SOLUTION	The tool diameter must be the same as or larger than the maximum roughing pass "Δ".

9070 'RECTANGULAR POCKET: Tool diameter larger than pocket'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	The tool diameter is larger than the pocket.
SOLUTION	The tool diameter must be smaller than the pocket's "L" and "H" dimensions of the pocket.

9071 'RECTANGULAR POCKET: Tool diameter FINISHING STOCK less than δ'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	The diameter of the finishing tool is smaller than the finishing stock "δ".
SOLUTION	The diameter of the tool must be greater than the finishing stock "δ".

9072 'CIRCULAR POCKET: F=0'

DETECTION	While executing a circular pocket cycle.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9073 'CIRCULAR POCKET: S=0'

DETECTION	While executing a circular pocket cycle.
CAUSE	No live tool turning speed "S" has been defined.
SOLUTION	Program a positive spindle speed "S" other than zero.

9074 'CIRCULAR POCKET: P=0'

DETECTION	While executing a circular pocket cycle.
CAUSE	The total pocket depth has not been defined.
SOLUTION	Program a total pocket depth greater than zero.

9075 'CIRCULAR POCKET: Wrong penetration angle'

DETECTION	While executing a circular pocket cycle.
CAUSE	The defined penetration angle is wrong.
SOLUTION	Program a penetration angle greater than 0º and smaller than or equal to 90º.

9076 'CIRCULAR POCKET: Tool diameter smaller than Δ'

DETECTION	While executing a circular pocket cycle.
CAUSE	The tool diameter is smaller than the maximum roughing pass "Δ".
SOLUTION	The tool diameter must be the same as or larger than the maximum roughing pass "Δ".

9077 'CIRCULAR POCKET: Tool diameter larger than pocket'

DETECTION	While executing a circular pocket cycle.
CAUSE	The tool diameter is larger than that of the pocket.
SOLUTION	The diameter of the tool must be smaller than the diameter of the pocket (2Rc).

FAGOR ⬤

·T· Model

Ref. 1310

9078 'CIRCULAR POCKET: Tool diameter FINISHING STOCK less than δ'

DETECTION	While executing a circular pocket cycle.
CAUSE	The diameter of the finishing tool is smaller than the finishing stock "δ".
SOLUTION	The diameter of the tool must be greater than the finishing stock "δ".

9079 'RECTANGULAR POCKET: δ not valid'

DETECTION	While executing a rectangular pocket cycle.
CAUSE	The defined finishing stock "δ" is not valid.
SOLUTION	Reduce the value of the finishing stock "δ".

9080 'POSITIONING: Value of T=0'

DETECTION	While executing a positioning cycle.
CAUSE	The tool number has not been defined.
SOLUTION	The tool number must be other than zero.

9081 'PROFILE POCKET: F=0'

DETECTION	While executing a profile pocket cycle.
CAUSE	The feedrate "F" has not been defined.
SOLUTION	Program a positive feedrate "F" other than zero.

9082 'PROFILE POCKET: S=0'

DETECTION	While executing a profile pocket cycle.
CAUSE	No live tool turning speed "S" has been defined.
SOLUTION	Program a positive spindle speed "S" other than zero.

9083 'PROFILE POCKET: Wrong penetration angle'

DETECTION	While executing a profile cycle.
CAUSE	The defined penetration angle is wrong.
SOLUTION	Program a penetration angle greater than 0º and smaller than or equal to 90º.

9084 'PROFILE POCKET: Tool diameter smaller than Δ'

DETECTION	While executing a profile pocket cycle.
CAUSE	The tool diameter is smaller than the maximum roughing pass "Δ".
SOLUTION	The tool diameter must be the same as or larger than the maximum roughing pass "Δ".

9085 'PROFILE POCKET: Tool diameter FINISHING STOCK less than δ'

DETECTION	While executing a profile pocket cycle.
CAUSE	The diameter of the finishing tool is smaller than the finishing stock "δ".
SOLUTION	The diameter of the tool must be greater than the finishing stock "δ".

9081 'PROFILE POCKET: P=0'

DETECTION	While executing a profile pocket cycle.
CAUSE	The total pocket depth has not been defined.
SOLUTION	Program a total pocket depth greater than zero.

·T· Model

FAGOR

·T· Model