7. Trouble-shooting
7.1 DIAGNOSIS operating mode

The DIAGNOSIS operating mode helps operators and
service personnel to find and identify errors in the
control system and machine. By means of the tool
diagnosis, the tool state can be interrogated and
influenced using the diagnosis variables.
The parameters can be protected against alteration
by entering a password (see section 6).

Select operating mode:

Press the operating mode key.

The operating modes menu is dis-
played on the monitor.

Press the DIAGNOSTIC MODE
softkey.

The main level of the diagnosis
menu is displayed:

SET TIME	ERROR OUTPUT	VERSION
PASS-WORD	SPS-MEMORY OUTPUT	SET PASS-WORD >
INPUTS	VARIABLE EXPRESS. CYCLIC	OUTPUTS

Press "continue" key.

The subsequent menu on the same
level is displayed:

TEST TEXT	TEST PICTURE	SET VARIABLE UNDEFIN
LANGUAGE 1	LANGUAGE 2	>
MEMORY OUTPUT	MEMORY OUTPUT CYCLIC	MEMORY INPUT

Time input
By pressing the SET TIME softkey
date and time can be programmed.
This input remains active after
switching off the machine.

Press the SET
TIME softkey.

The control system requests the
values by dialog.

Enter values.

Confirm.

Display:
e.g. 27-OKT-1989   16:25

Error output
The last ten errors can be
displayed.

Press the ERROR
OUTPUT softkey.

The error numbers, text and overflow
display of the last ten errors are
displayed.

Press the "delete" key

... ERROR: corresp. error message 9#

Press the "delete" key

... ERROR: corresp. error message 8#

etc.

**Software production
date**
This function can be used to dis-
play ths software production date.

Press the VERSION
softkey.

INPUT MODULE NUMBER:

Enter digits (e.g.: 45)

e.g. ZIFBEA 25-SEP-1986

Confirm.

Confirm.

Proceeds to next module.

**Release of the protected parameters
by entering a password**
If parameters protected by a password are to be altered
or newly entered, then the valid password must be entered
once. Following this, the corresponding parameters can be altered
in the PARAMETER operating mode. This release produced by
entering a password is active until an incorrect password is entered
or the control system is switched off. After the control switch is
switched on again or an incorrect password is entered, the password
has to be entered onceagain if protected parameters are to be changed.
The parameteres are **not** protected when
the password **9999** is active.

<table>
<tr><td></td><td>Press the softkey
PASSWORD</td><td>INPUT PASSWORD
(KEY 1 ... 9)</td></tr>
<tr><td></td><td>Enter digits of the password</td><td>Protected parameters are
released.</td></tr>
</table>

SPS-memory output
Memory output from SPS AT186 is displayed on
the monitor in hexadecimal or binary code after
entering segment and offset address in hexadecimal code.

<table>
<tr><td></td><td>Press the SPS-MEMORY
OUTPUT softkey.</td><td>INPUT ADDRESS (0...FFFFH):</td></tr>
<tr><td></td><td>Enter digits.</td><td></td></tr>
<tr><td>></td><td>Switchover to
letter menu e.g.: 12 E</td><td></td></tr>
<tr><td></td><td>Enter letters</td><td></td></tr>
<tr><td></td><td>Confirm.</td><td>e.g.   SPS 0200.0010
00FF 00000000 11111111</td></tr>
</table>

The corresponding input and output bytes of the SPS
are displayed.

<table>
<tr><td></td><td>Press the softkey VALUE FORWARD
or VALUE BACKWARD</td><td>Either the following or the previous
address is displayed.</td></tr>
</table>

Setting a new password
Input or alteration of certain parameters
can be blocked by entering a 4-character
password. If the existing password is to be
replaced by a new one, proceed as follows:

⊞	Press the SET PASS- WORD softkey.	INPUT PASSWORD (PRESS KEY 1...9)
⊞	Enter original pass- word in digits.	INPUT NEW PASSWORD (PRESS KEY 1...9)
⊞	Enter new password using the digits on keypad.	REPEAT PASSWORD (KEY 1...9)
⊞	Enter digits	If digit sequence 9999 is entered, the parameters are no longer protected.

Note: The password 999 was programmed by the manufacturer before the delivery
         of the control system. This password **does not**, however, **protect the parameters.**

Display of input bytes

⊞	Press the INPUT softkey	ENTER INPUT NUMBER:
⊞	Enter digits (e.g. 2).	
▬	Confirm.	e.g. 12 10010011
⊞	Press the VALUE FORWARD or VALUE BACKWARD softkey.	x xxxx xxxx (cyclic display)
∧	Press	Return to the diagnosis menu. The display remains present until another cyclic output or the DIAGNOSIS operating mode is called.

Output of variables
The contents of a variable are
displayed cyclically.

Press the VARIABLE
EXPRESS. CYCLIC softkey.

Enter digits.

Confirm.   e.g. V11 - 0.000000

Press the VALUE FORWARD or   Displays further variables.
VALUE BACKWARD softkey.

Note
If this variable is undefined, a question mark will be displayed; in the event of
format errors, the value of the variable will be displayed video-inverse.

#Display of output bytes

Press the OUTPUT   ENTER OUTPUT NUMBER:
softkey

Enter digits (e.g. 2)

Confirm   e.g. 02   10110011

Press the VALUE FORWARD or   0x xxxx xxxx
VALUE BACKWARD softkey.

Press   Return to the diagnosis menu.
                                      The display remains present until
                                      another cyclic output or the
                                      DIAGNOSIS operating mode is
                                      called.

Text test (for internal testing purposes only)

Press the TEST TEXT
softkey.

INPUT TEXT NUMBER: .

Enter digits.

Confirm.

The corresponding text from the
language PROM is displayed on the
monitor.

Picture test (for internal testing purposes only)
If pictures erase outputs on the
monitor screen (picture 2, 5, 16...19
etc.), the control system must be re-
set (switched off and on again).

Press the TEST PICTURE
softkey.

INPUT PICTURE NUMBER:

Enter digits.

Confirm

The corresponding support graphics
are displayed.

Cancelling variable definitions
This function is used to cancel
the definition of a variable.
Proceed as follows:

Press the SET VARIABLE
UNDEFIN. softkey.

Confirm.

Variables V1 to V299 are set in the state of
"undefined" while the other variables are
set to zero.

Switchover of the language displayed on the screen
Texts guiding the operator can be displayed in three
languages (these languages can be chosen at will from the
total of the languages offered by GILDEMEISTER
AUTOMATION).
When the control system is switched on, the language
selected lastly will always be active.

Press the softkey   The texts are displayed in the second
LANGUAGE 2   language ordered by the client.

Press the softkey   The texts are displayed in the first
LANGUAGE 1.   language ordered by the client.

Note: After a new language was selected to be displayed on the screen,
           the control system must be switched off and on again in order for
           the total screen texts to be displayed in the new language.

Memory output
The segment and offset addresses
must be entered in Hex format.

Press the MEMORY OUT-   INPUT SEGMENTAL OFFSET:
PUT softkey.

Enter digits.

Switch over to   e.g.: 12 AF. A435

Letters menu.

Confirm   12 AF: A435 FF FF FF
                                    FF FF FF FF FF FF FF
                                    "..........."
                                    Ten memory addresses are dumped
                                    in Hex ASCII.

Cyclic memory output

	Press the MEMORY OUTPUT CYCLIC.	INPUT SEGMENTAL OFFSET:
Enter digits.	
> Switchover to	
Letters menu e.g. 12 E	INPUT NUMBER OF BYTES:
Confirm.	
Enter digits (0-4).	
Confirm.	012 E: 0000 9BDD
Press the VALUE FORWARD or VALUE BACKWARD.	The previous or subsequent address of a segment is displayed.

Memory input
After pressing the softkey MEMORY INPUT,
a submenu is displayed offering the following
functions:

- deleting the program memory
- deleting the parameter memory
- deleting the working memory
- setting a memory

These functions are designed for the service
personnel of the GILDEMEISTER AG exclusively
and are therefore protected by a password
(not the same password as for parameters).
The Memory input function alters
memory addresses which may impair
correct functioning of the control system.

7.   Trouble-shooting
7.2 DIAGNOSIS operating mode in side-line operation

The DIAGNOSIS operating mode can be called up in side-line
operation from the EDITOR, MANUAL CONTROL, AUTOMATIC
CONTROL SINGLE BLOCK, DNC and PARAMETER operating modes.

Press the DIAGNOSIS softkey
in the respective operating mode.

The DIAGNOSIS main menu is
displayed on the screen.

SET TIME	ERROR OUTPUT	VERSION
PASS-WORD	SPS-MEMORY OUTPUT	SET PASS-WORD >
INPUTS	VARIABLE EXPRESS. CYCLIC	OUTPUTS

Note:
If DIAGNOSIS has been selected for side-line operation,
the user can only **call up information** in the diagnosis functions
(memory, variables etc.). Modifications, however, can **not** be
carried out. Any attempts to do so will lead to the display of an
error message.

There are two possible ways to exit the DIAGNOSIS
side-line operating mode:

Press the "next higher
menu level" key

Return to the initially selected operating
mode.

or

Press operating mode key.

Call-up of the operating mode menu.

7. Trouble-shooting
7.3 DIAGNOSIS / TOOL DIAGNOSIS operating mode

Tool diagnosis variables
The ELTROPILOT L - control system is provided with
an extended tool diagnosis, which enables the
operator, even during the course of the program,
to

- get information on the state of particular
  tools

- to influence, by hand, the state of particular
  tools (e.g. identifications of inactivity)

- to influence the state via the part program
  (measuring pieces, measuring tools) and to
  interrogate it.

In order to call up the tool diagnosis, proceed
as follows:

[>]	Press the "continue" key	The second subsequent menu of the operating mode DIAGNOSIS is displayed.
	Press the TOOL DIAGNOSIS VARIABLE softkey.	TOOL NUMBER:
	Enter digits. e.g.: 1	
	Confirm	In addition to the tool diagnosis variables an explanatory text and the service life of the corresponding tool are displayed.

BIT	16	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1
N1401 T1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0

N1201   T1   S0   PZ   0 : 0 : 0   AZ   0 : 0 : 0   PS 0.   AS 0 >
        AT 0

BIT 1 - 8   tool inactivated by:

1 general
2 expired service life or quantity
3 tool in-process   4 part in-process
5 part post-process 6 rupture   7 wear
8 free

BIT 9 - 16   tool:

9 used   10 active   11 first using
12 total tool life < 6% or last workpiece
13 free
14 free   15 in turret   16 in magazine

If the bits 1 - 16 are set, i.e. a "1" is displayed, then the state
applies which is mentioned for the corresponding bit.

The diagnosis variables can be divided into two sections according to
their contents:
Bit 1 - 8:   identifications of inactivity
Bit 9 - 16: general identifications

Their meaning in particular is:

Bit 1:   If the corresponding tool is identified as inactive, due to
                an event specified in detail under the following bits, then
                this bit is always generally identified as inactive.

Bit 2:   The corresponding tool is inactivated due to expired service
                life or because the maximum number of pieces that can be machined
                has been reached. This identification is automatically set by the
                tool organization.

Bit 3:   Due to the measured values of the tool in-process-measurement
                the corresponding tool was declared unusable via the part program.

Bit 4:   The corresponding tool was declared unusable via the part program
                due to the measured values of the part in-process measurements
                (see also the following example).

Bit 5:   The corresponding tool was declared unfit via the part
                   program due to the measured values of the part post-process
                   measurements.

Bit 6:   *Due to the measuring of the cutting force, a tool rupture was*
                   signalled by the tool organization.

Bit 7:   *By measuring the cutting force, indications of wear of the corres-*
                   ponding tool were detected which do not allow to continue machining
                   with the tool affected.

Bit 8:   *Tool was inactivated by (optional occupation)*

Bit 9:   This bit is used by the tool organization to mark a tool which has been
                   used before for a machining process.

Bit 10:   This bit is set by the tool organization when the corresponding
                   tool is just being used for machining.

Bit 11:   If the corresponding tool is being used for the first time,
                   then this bit is set by the tool organization

Bit 12:   This bit is set if less than 6% of the total service life of the
                   corresponding tool group (tool with all exchange tools) is
                   available or if only one more part can be machined with these tools.

Bit 13:   free

Bit 14:   *free application (bit is set by the part program).*

Bit 15:   If the corresponding tool is located in the turret, then this bit
                   is set by the part program.

Bit 16:   This bit is set by the part program if the corresponding tool is situated
                   in the tool magazine.

Note: If, in the MANUAL operating mode, the values for the
                   service lives of unusable tools are reset to their original values
                   after the cutting plates have been renewed, then the corresponding
                   identifications of inactivity are cancelled, that is, the bits are
                   set to zero.

                   The text printed *in italics* signifies that the functions of the
                   corresponding bits have not yet been realizied.

If these identifications are to be changed by hand by the operator,
then the cursor can be moved in front of the corresponding bit.

Press softkey MAINTAIN
VALUE.

Cursor moves to the next
bit.

Press the ACCEPT NEW
VALUE softkey.

CHANGE VALUE OR CONFIRM

Enter digits (1 or 0)

Confirm

The newly entered identification is
transferred into the display.

Press the TERMINATE
INPUT softkey.

Return to the superior menu level.

The following example shows a two-point-measurement on the diameter
of a workpiece, together with surveillance of the tolerance and, if
necessary, identification of inactivity of the corresponding tool,
or modification of the corresponding D-correction via the part
program, respectively.

Part program

```
N49...
N50   V { V983 = 50 }
N51   V { V984 = 0.02 }
N52   V { V985 = 2.5 }
N53   V { V986 = 25 }
N54   T10
N55   G0   X100   Z25   M...
N56   L27
N57   G61   H { ABS (V992) <= V984 } N60
N58   V { T1 [4] = 1 }
N59   G61   N61
N60   V { D1 [X] = D1 [X] + V992 }
N61...
```

Measuring cycle %27

```
N 1  G94  F150  M5
N 2  G910
N 3  G0  X { V983 + V985 + 10 }
N 4  G912
N 5  G1  X { V983 + V985 - 10 }
N 6  G913
N 7  V { V990 = V983 + V985 - V980 * 2 }
N 8  G910
N 9  G0  X { V983 + V985 + 20 }  Z50
N10  G0  X { V983 + V985 + 10 } * (-1)
N11  G0  Z { V986 }
N12  G912
N13  G1  X { ( V983 + V985 - 10) * (-1)
N14  G913
N15  V { V991 = V983 + V985 + V980 * 2 }
N16  V { V992 = ( V990 + V991 ) / 2 }
N17  M30
```

Explanation (part program)

N49	Machining process of the part (block N1 to N49)
N50	The nominal value (diameter 50 mm) is allocated to the variable V983
N51	The range of measurement tolerance 0.02 mm is allocated to the variable V984
N52	The radius of the probe is allocated to the variable V985 as diameter dimension (2.5 mm)
N53	The measuring point Z (Z= 25 mm) is allocated to the variable V986
N54	Swivelling in of the measuring tool T10
N55	Pre-positioning in rapid traverse to the diameter 100 and measuring point Z and switching on of the part measuring electronic system
N56	Call up of the subprogram L27 activating the measuring cycle
N57	If the absolute value of the arithmetic mean value of the difference between nominal and measured value (measuring differences) is less or equal to the tolerance range 0.02, then due to the jump condition the part program is continued with block N60. If the value is larger then the measuring tolerance, then the program is continued with block N58.
N58	The service life of tool T1 is set to zero due to the event "inactive due to part mesuring", that is, the tool is declared unusable.
N59	Unconditional jump to block N61

N60   Adding the contents of the variable V992 (mean value of measuring differences)
           to the D correction value D1 (X)

N61   Continuation of the machining process of the part

% 27 (measuring cycle as subprogram)

N 1   Feedrate 150 mm/min and stopping spindle movement

N 2   Activating hardware for part measurement and switching on the
           collision surveillance (measurement 1)

N 3   Pre-positioning in X-direction to 5mm before nominal dimension + radius
           of probe (measurement is to be executed above the center of turning)

N 4   Switchover of the probe input activated by G910 to detection of actual value

N 5   Moving probe in X direction (measurement 1) and transferring the
           measured value to variable V980.
           Note: The measured value is stored as a radius dimension

N 6   Finishing measurement 1

N 7   The difference between nominal and measured value is allocated to the
           variable V990

N8   Activating the hardware for part measuring and switching on the collision
           surveillance (measurement 2)

N9   Moving the probe back in rapid traverse

N10   Pre-positioning in X direction to 5 mm before nominal dimension + radius of
           probe (measurement is to be executed below the center of turning)

N11   Pre-positioning in Z direction

N12   Switchover of the probe input activated by G910 to detection of
           the actual value

N13   Moving the probe in X-direction (measurement 1) and transferring the
           measured value to variable V980.
           Note: The measured value is stored as radius dimension

N14   Terminating measurement 2

N15   The difference between nominal and measured value is allocated to the variable V991

N16   The arithmetic mean value of both measuring differences is allocated
           to the variable V992

N17   End of program of measuring cycle

7.4 Error diagnosis

This error description is intended to assist the "operator at the system" to isolate and eliminate the errors reported in messages which are generated because of an operator error involving the control system or machine or because of a software error in the associated parts programs.

It also lists errors in the software of the control system and describes their cause.

Error list
The error list contains four types of errors:

Operator errors
Whenever the control system recognizes an error in operation it generates an error message; such messages are identified by BF... on the following list. A message indicating the operator error appears on the input line of the monitor for 4 seconds, after which the entry can be repeated. Operator errors are not stored in the control system's error memory.

General errors
General errors recognized by the control system are identified by AF... on the following list. Most such errors can be corrected by the operator since they are related to the development of the program and/or occur when operating the control system or machine. These error messages appear on the error line of the control system monitor and are stored in the control system's error memory.

Software errors
The error messages identified by SWF... on the following list are software errors in the control system which the operator can usually not correct by himself. Nevertheless, wherever such is possible the following list will provide a short description of the causes of these errors. If the description of the error cause refers to a specific hardware module which cannot be initialized or addressed by the control system, the operator should make certain that the board in question is properly seated in the board rack and he should check all of the card's connecting cables. Where appropriate, he should switch the control system off and on and then repeat the operation. If the error message is repeated, it is advisable to notify Gildemeister Service.

Error messages of this type are also stored in the control system's error memory.

SYSTEM ERRORS

If errors recognized by the control system result in a situation in which continued operation can no longer be guaranteed, the machine is shut down (EMERGENCY OFF). These errors are displayed on the error lines as SYSTEM ERROR or abort errors. Work can only be continued by switching the machine off and back on again. Report any unidentified system errors or abnormal termination messages to your GILDEMEISTER representative. System errors which possibly occur because of hardware damages are characterized by SF.

ERROR MEMORY

All error messages except operator errors (BF) are stored in the control system's error memory in the order received.

If no EMERGENCY OFF exists, the oldest error is displayed, i.e. the first error transferred to the error memory. If applicable, the number of errors following the one being displayed also appears at the same time (e.g. 2# displayed to the right of the error message means that two more errors follow the one currently being indicated).

If EMERGENCY OFF exists, 0005 EMERGENCY OFF is displayed, not the oldest error.

Note
When the control system is transformed into the EMERGENCY OFF state no deletion of the screen contents ensues, consequently the cause of this states can be better diagnosed.

DELETING THE ERROR MESSAGE

Proceed as follows to delete an error message from the control system's error memory:

Press DELETE key

The "deleted" error is no longer displayed and the next oldest error appears on the error line. The display of any following errors is decremented by 1.

FIVE-DIGIT ERROR MESSAGES

If an error refers to just one particular slide, a 5-digit error message appears in which the first digit is the slide number. If the actual error message in such a case has only 3 digits, a blank appears between slide number and error number.

ERROR LIST

To present a complete picture, the following listing of the error
messages contains all of the error messages, those for the lathe and
those for the transport axis system.

AF 005   EMERGENCY OFF

   General error

      The operator has triggered the function "EMERGENCY OFF" by pressing
      the mushroom button, or the control system produced this state because
      there is no guarantee of continuing to work properly without
      endangering the machine and operator.

   Remedy

      If the EMERGENCY OFF mushroom button has been pressed (screen contents
      remain to ensure better diagnosis of the cause), the machine can be
      put back into service as follows once the cause for actuating
      EMERGENCY OFF has been eliminated:

      -   Release the EMERGENCY OFF button by turning it clockwise

      -   Press the key Control system ON

      -   Select the manual control operating mode

      -   Open and close the sliding protective door

      -   Approach reference point(s) again

      -   Continue work operation

      -   Move chuck, quill, tailstock and similar machine options (if
          installed) into defined position by actuating them.

AF 0006   Positive limit switch X overtravelled

AF 0007   Negative limit switch X overtravelled

AF 0008   Positive limit switch Z overtravelled

AF 0009   Negative limit switch Z overtravelled

General error

The pertinent software limit switch +X, -X, +Z or -Z was not set correctly in parameter N200, consequently the appropriate hardware limit switch on the machine has been actuated during traversing of the slide.

In the manual control mode the pertinent slide was moved to the appropriate hardware limit switch with the manual direction switch without any preceding reference run to the machine's corresponding hardware limit switch. The software limit switches programmed under parameter N200 are not active without a reference run.

Remedy

Moving clear of the limit switch (see Chapter 5 of the Operating and Programming Instructions).

Check parameters for the limit switch and correct where applicable.

If traversing movements of slides are carried out in the manual control mode without a preceding reference run, do so carefully.

AF 0010   Undefined limit switch overtravelled

General error

If the physical limit switch cannot be identified, this error message appears. The error is to be evaluated in connection with error numbers AF 0006 - AF 0009.

AF 0011   Limit switch in +Y direction overtravelled

AF 0012   Limit switch in -Y direction overtravelled

General error

The software limit switch +Y or -Y in parameter N200 was not set correctly, consequently the corresponding hardware limit switch on the machine was tripped during traversing of the slide.

In the manual control operating mode the slide in question was moved to the corresponding hardware limit switch of the machine without a preceding reference run. The software limit switches programmed under parameter N200 are not active without a reference run.

Remedy   7-21

Move clear of the limit switch (see Chapter 5 of the Operating and
Programming Instructions).

Check parameters for the limit switch and correct where applicable.

If traversing movements of slides are carried out in the MANUAL
control mode without a preceding reference run, do so carefully.

The error numbers 0400 - 899 are reserved for machine adjustment (SPS) errors.

SWF 0911 String empty

Software error

(ZIWORD) has been applied to a string with blanks

SWF 0912 String has no digits

Software error

(ZIHWOR) string contains an invalid hexadecimal symbol;
(ZIREAL, ZIWORD) string contains an invalid symbol in 1st position.

SWF 0913 Negative number forbidden

Software error

(ZIDEZR) format specification does not allow any negative numbers.

SWF 0914 Format too small

Software error

(ZIDEZR) number is larger than format specification permits.

SWF 0915 Digit expected

Software error

(ZIREAL) string contains an invalid symbol after the minus sign.

BF 0916 Character cannot be edited

Operator error

The character utilized cannot be used at this position, since it is already being used by the control system in a different context.

Remedy

Change NC block.

BF 0917 Comparison operator forbidden

Operator error

The comparison operators <,>, ==, <>, <= or >= cannot be used in the allocation of a variable. They are only permitted in connection with the function G61 when a jump condition is being defined.

Remedy

Refer to specifications for the allocation of a variable in Section 8.1 of the Operating and Programming Instructions.

SWF 0918 Prefix forbidden

Software error

(ZIWOCP, ZIWOEX) string contains a minus sign although this is not allowed according to the format specification.

Remedy

Change NC block.

BF 0919 Decimal point forbidden

Operator error

In the allocation of a variable the decimal point is only to be used with the decimal numbers of the value allocation to separate the integer from the non-integer portion.

Remedy

Refer to rules for allocation of a variable in Section 8.1 of the Operating and Programming Instructions.

SWF 0920 Faulty digit sequence

Software error

(ZIWOEX) string contains too many digits. Standardize string for editing to an allocated format < 255 bytes.

SWF 0921 Format too small

Software error

(ZIWOEX) digit string is longer than allowed by format specification.

SWF 0922 Format specification incomplete

Software error

(ZIWOCP) For editing, the string is placed in a uniform format. If this is not possible or is done incorrectly, the error message is generated.

BF 0923 Invalid format

Operator error

(ZIDEZW) number is larger than allowed by format specification.
The number to be converted cannot be represented in the specified format.

BF 0925 Digit forbidden

Operator error

The control system expects a letter at this position in the variable allocation.

Remedy

Refer to rules for allocation of a variable in Section 8.1 of the Operating and Programming Instructions.

BF 0926 Digit forbidden

Operator error

It is not possible to enter digits at this position after (, ., [,].

Remedy

Change NC block.

BF 0927 Digit expected

Operator error

The control system expected one or more digits to be entered at this position.

Remedy

Change NC block.

BF 0928   NC syntax error

Operator error

It is not permissible to enter or output text via printer or master computer via softkey INPUT/OUTPUT at this position in the variable allocation.

Remedy

Refer to the programming of variables in Section 8.1 of Operating and Programming Instructions.

BF 0929   "=" expected

Operator error

An equal sign must be programmed at this position.

BF 9030   Minus sign forbidden

Operator error

In a variable allocation it is not permissible to enter a minus sign before (, .,], (,), [or after ., {, >, =,], (,), }.

Remedy

Refer to rules for the allocation of variables in Section 8.1 of the Operating and Programming Instructions.

BF 0931   Variable reference forbidden

Operator error

The use of D, E, M, T, V, X, Y, Z is forbidden at this position.

Remedy

Refer to rules for the allocation of variables in Section 8.1 of the Operating and Programming Instructions.

BF 0932   End of string

Operator error

The cursor is already at the extreme right.

Remedy

Refer to rules for the allocation of variables in Section 8.1 of the Operating and Programming Instructions.

BF 0933 "(" forbidden

BF 0934 ")" forbidden

BF 0935 "=" forbidden

Operator error

Use of the left bracket, right bracket and equal sign is forbidden at
this position in the variable allocation.

Remedy

Refer to rules for the allocation of variables in Section 8.1 of the
Operating and Programming Instructions.

BF 0936 Simplified geometry programming forbidden

Operator error

The simplified geometry programming can only be used for the calculation of contour
elements during turning operations. Contour elements of milling operations in the XY,
YZ plane or in connection with the C-axis cannot be calculated using simplified
geometry programming.

Remedy

Contour elements to be executed have to be unambiguously specified by
the operator.

BF 0937 "+, -, /, *, ^" forbidden

Operator error

Arithmetic operations are forbidden at this position in a variable allocation.

Remedy

Refer to rules for the allocation of variables in Section 8.1 of the Operating and
Programming Instructions.

BF 0938 Function forbidden

Operator error

The last function called is not permitted at this position.

BF 0939 Variable reference incomplete

Operator error

The allocation of a variable is incomplete as entered and therefore cannot be executed.

Remedy

Refer to rules for the allocation of variables in Section 8.1 of the Operating and Programming Instructions.

BF 0940 Brackets not in pairs

Operator error

Left bracket or right bracket is missing.

Remedy

Refer to rules for the allocation of variables in Section 8.1 of the Operating and Programming Instructions.

BF 0941 Variable allocation incomplete

Operator error

Call for the allocation of a variable. No allocation has taken place (e.g. VII and confirmation key pressed).

Remedy

The variable has to be allocated a defined value (e.g. VII = 25).

BF 0942 Variable allocation too long

Operator error

The allocation of a variable is not to be more than 253 characters long. If this length is exceeded, the error message is generated.

Remedy

Shorten the allocation of a variable as necessary.

BF 0944 Start of string

Operator error

Cursor is already at the extreme left.

Remedy

> Refer to rules for the allocation of variables in Section 8.1 of the Operating and Programming Instructions.

BF 0960 Operating mode forbidden during automatic parallel operation

Operator error

> The operating mode manual control or reference run cannot be called as a parallel operating mode since this would produce an abnormal interruption of the automatic mode.

Remedy

> Perform desired operations after completely executing the program.

BF 0961 Option missing

Operator error

> The graphic representation option (graphic simulation of programs) is missing.

Remedy

> If the option was purchased by the customer but mistakenly not released, notify Gildemeister Service.

> Option can also be subsequently purchased.

BF 0962 Operating mode permitted only after moving clear

Operator error

> A slide has run against a limit switch -X, +X, -Z, +Z. The selected function is not permitted until the slide has been moved clear (slide has to have left the relevant limit switch).

Remedy

> For a description of the moving clear function, refer to "Moving clear of a limit switch" in Chapter 5 of the Operating and Programming Instructions.

AF 0963 Selected operating mode forbidden because of EMERGENCY OFF

General error

> The machine is in the EMERGENCY OFF state. All functions are therefore suppressed.

Remedy

Restarting after EMERGENCY OFF: Refer to Chapter 5 of the Operating
and Programming Instructions.

SWF 0964 Test sum error in the parameter

Software error

Parity error in the parameter memory.

SWF 0969 Test sum error in the parameter

Software error

Parity error in the parameter memory.

BF 0990   Spindle jogging during cycle ON forbidden

Operator error

Automatic operating mode. Use of manual control mechanisms while a cycle is active
is not permitted. When executing a parts program with cycle ON, this error message
appears if the spindle jogging key is pressed. The command for spindle jogging is
ignored.

Remedy

Do not carry out the jogging function until the parts program has been
completely executed; this requires a change of operating mode.

SWF 0997 Keyboard operation without test characters

Software error

The SET card runs without test characters

SWF 1101 Unknown function code

Software error

Function code cannot be interpreted by the control system.

AF 1102   Serial interface currently occupied

General error

The external device (e.g. punching unit) has been switched off but an
attempt is being made to conduct an external data transmission.

Remedy

All of the units involved in the data transmission must be kept ready for operation.

BF 1401 Too many programs active

Operator error

User list is fully occupied. Too many programs/subprograms have been allocated to a single slide.

Remedy

A maximum of only 6 programs can be opened per slide.

SWF 1402 Program is missing

Software error

User does not exist. (SPPRLO) user is not stored on the user list.

BF 1403 Not enough storage space

Operator error

The NC parts program memory is full. This error message is output when a new program is to be opened but practically no more storage space exists (less than 17 bytes).

Remedy

Copy programs not needed onto external data memory or add memory expansion.

BF 1404 Program access blocked

Operator error

Reading access to write program. A program with subprogram is being executed in automatic mode. Simultaneously, the subprogram is called up in the editor for editing; this triggers the error message.

Remedy

Editing can be performed in this program after it has been completely executed.

BF 1405 Program access blocked

Operator error

A program is being accessed from two different stations (NC control system/master computer).

Remedy

Coordinate the time of program accesses so that the program is not accessed from two stations simultaneously.

BF 1406 Program directory full

Operator error

The maximum number of opened programs permitted is 200. Where very large programs are involved, however, the total memory capacity may be used before the maximum number of programs (200) is reached. The error message is then: "Not enough storage space".

Remedy

Copy unneeded programs onto external data memory or delete them, as applicable. If the memory capacity is not sufficient, retrofit memory expansion.

SWF 1407 Program not opened

Software error

Program not opened for writing.

BF 1408 Program missing

Operator error

Program number not found in the parts program memory (e.g. delete editor/program). A nonexistent program number has been entered and an attempt made to delete it, or a nonexistent subprogram is being called by a main program.

Remedy

Check the facts by listing the programs (editor/program list).

AF 1409 Block missing, block number not found

General error

The selected block does not exist in this program. Perhaps the search was not started at the program beginning, the program end is missing or the jump target of function G61 (conditional jump) is missing.

Remedy

Check program on basis of these criteria.

BF 1410 Program access blocked

Operator error

Writing access to reading program. For example, a program is being run in automatic mode and an attempt is simultaneously being made in the editor to edit it.

Remedy

Editing can be carried out in this program after it has been completely executed.

SWF 1413 invalid characters in the program memory

Software error

Block structure faulty. The block number string is not of valid length.

BF 1414 Not enough storage space

Operator error

Insertion not possible. The remaining storage space available is insufficient to edit a program or expand an existing program.

Remedy

If the memory capacity is 32 kBytes, it can be expanded to 64 kBytes.
If it is already 64 kBytes, storage on an external data medium is advisable.

BF 1415 No block number available

Operator error

All possible block numbers are in use. All available 9999 blocks are occupied. The program management can administer max. 300 NC blocks.
This error message also appears if an attempt is made to reorganize a program with more than 300 blocks.

Remedy

Change program structure, if appropriate.

SWF 1416 Invalid characters in the program memory

Software error

Incorrect test sum. The stored program has been damaged. The test sum comparison reports an error.

Action: Selecting a program generates a new interpretation.

BF 1417 Comment block deleted

Operator error

If a parts program is created via an external programming system, comment blocks can be written above the actual program. If such a block is subsequently deleted, it no longer appears but the space in the program continues to be reserved. If an attempt is made to delete this space with the aid of the NC editor, this error message appears. Of no significance for the program run.

SWF 1418 Block too long

Software error

The distance between two automatically generated block characters is greater than the permissible block length.

BF 1419 Too many blocks

Operator error

This error occurs if a program with more than 300 NC blocks is supposed to be restructured in the editor/program list/program block organization. A program cannot consist of more than 300 blocks. Block numbers from 1 to 9999 can be used.

Remedy

If this error message appears, the programmer should combine several NC blocks into a single NC block, if such exist and programming of the functions does not require individual NC blocks.

BF 1424 Program with same number

Operator error

A program is supposed to be opened with a program number which already exists in the parts program memory.

Remedy

Select a different program number

BF 1425 Search word not found

Operator error

Mode: Search for NC word. The character (chain) being sought has not been found in the selected program.

Remedy

Where applicable, repeat search from beginning of program.

BF 1427 Block number too long

Operator error

If a program is created at an external programming station or with DNC and then read in via the serial interface, no syntax check is carried out during the transmission. Block numbers which are not in the required format may therefore be transferred (e.g. 5 digits). The first time the program is called up, the interpreter detects the format error and generates this error message.

Remedy

Change the program (max. permissible length of block numbers is 4 digits).

BF 1428 Block number not found

Operator error

Mode: Search for editor/block number. The block number being sought does not exist in this program. In case of program organization/block reorganization: the block number to be renamed was not found.

Remedy

Check the program.

BF 1429 Program missing

Operator error

The program called up is not in the program memory, e.g. a nonexistent subprogram is being called in a main program.

Remedy

Call for a listing of the program memory contents (program list).

BF 1451 Block number already exists

Operator error

Mode: New editor/block. The number entered is for a block which already exists in this program.

Remedy

Confirm after "Block new" to have the control system define the block number or select a different block number.

BF 1452 Block number missing

Operator error

The block number sought does not exist in selected program or search has not been started at program beginning.

SWF 1453 Block-by-block operation mode not selected

Software error

The block-by-block operation mode was not transferred properly.

BF 1454 No NC word selected during word operation

Operator error

Change operating mode editor/NC. The cursor is not on a word that could be changed, or the block is empty.

Remedy

Use the cursor to accurately select the NC word to be changed.

BF 1455   Format of program numbers not permissible

Operator error

The program number must consist of min. 1 and max. 8 characters (letters). The string is not to be separated by any other character.

If a program is created at a programming station or with DNC and read in via the serial interface, no syntax check is carried out during the transmission. Program numbers which are not in the required format may therefore be transferred (e.g. 9 digits). The first time the program is called, the interpreter detects the format error and generates the error message.

or:

After selecting the softkey "Program selection" there is no program number in the display and the confirmation key is pressed.

or

After selecting "Delete program" there is no program number in the display and the confirmation key is pressed.

Remedy

Change the program number or select the program to be executed.

BF 1456   Block operation not permissible

Operator error

A program without content is in the editor or the cursor is above or below a program (first or last block respectively) and the "Change block" function is being called.

Remedy

Use the cursor to select the block to be changed.

BF 1457   Non-corresponding measuring system

Operator error

The parameter N1 (measuring system) is programmed to determine whether the entered values are to be calculated in the metric system or in inches. The programs are given a corresponding code in the control system. If the program code and the parameter N1 are different (different measuring systems), the error message appears.

Remedy

Select the correct measuring system under parameter N1 (0 = metric; 1= inches).

BF 1458 Option missing

Operator error

The option "Materials file" has been called but has not been released or does not exist

Remedy

If the option was purchased by the customer but mistakenly not released, notify Gildemeister Service.

Option can also be subsequently purchased.

BF 1459 Editing blocked

Operator error

The editing switch on the control console is not pointing to editing ON, therefore the selection program cannot be changed.

Remedy

Set the editing switch to the editing ON position.

BF 1461 Key forbidden here

Operator error

The key just actuated is not permitted in the selected menu.

SWF 1700 Projection data missing

Software error

The projection data for the M- and G-functions is not available.

BF 1701 Length of block exceeded

Operator error

The max. block length (string length 64 - block end code) is exceeded when the NC word is inserted.

Remedy

Program the desired machining process in one or more NC blocks.

SWF 1702 Projection data missing

    Software error

        Faulty projection data

BF 1703 NC function already exists

    Operator error

        This error occurs when two T-functions are called up directly or
        indirectly in a single block,

        e.g. N1 692 X... Z... I... K... T3 M3 T345621

        G92 already contains a T-function. The same error occurs in case of
        double feed entries, double deletion level or double subprogram call.

    Remedy

        Check and alter the program with regard to this fact.

BF 1704 NC function already exists

    Operator error

        Same function (without address parameters) found. In contrast to error
        BF 1703, the functions meant here contain no additional address parameters.

    Remedy

        Check and alter the program with regard to this fact.

BF 1705 NC parameter already exists

    Operator error

        The function G92, which contains a T-function, was programmed after a
        T-function.

    Remedy

        Check and alter the program with regard to this fact.

Bf 1707 Competing NC function

    Operator error

        Mutually exclusive MC functions were called up, e.g. M3/M4.

Remedy

Refer to the meanings of the individual functions in the code list of
the Operating and Programming Instructions.

BF 1708 NC parameter already exists

Operator error

Occurs in the case of G-functions which directly or indirectly contain
a traversing path and are programmed in one block.

Remedy

Program the G-functions in question in a separate NC block.

BF 1709 Too many NC functions

Operator error

More than three G-, M- or $-functions have been called up in one block.

Remedy

Distribute the machining process to be programmed to more than one NC block.

BF 1710 Function value not entered

Operator error

A digit input is still expected.

BF 1711 Function value forbidden

Operator error

An impermissible value or no value was entered (just confirmed) for a
function in the operating mode editor/change block/change NC word.

Remedy

The meaning of the individual addresses is explained in the detailed
description of the G-function in question which is in Section 3.2 of
the Operating and Programming Instructions.

AF 2010   Transmission disrupted or end of punched tape

General error

External data traffic/program block input mode. The control system expects an external data input within a defined time. If this is not received, for technical reasons for example (faulty line connection, incorrect pin assignment or not ready to transmit/receive), the error message is output.

Remedy

Check and, where applicable, correct line connection, pin assignment and external equipment.

SWF 2303 Negative number forbidden

Software error

Format specification does not permit any negative numbers.

SWF 2304 Format too small

Software error

The number is larger than permitted by the format specification.

SWF 2313 Format not permitted

Software error

Decimal conversion of a binary number. The number to be converted cannot be represented in the indicated format.

AF 2404   Abort via stop key

Software error

External data traffic mode. Data transmission was terminated by manually actuating the stop key.

Remedy

Press the delete key and then continue.

AF 2405 Transmission disrupted

General error

> External data traffic mode. If no data connection is set up after a
> predefined time during separate routine input/out and program block
> output, the attempt is interrupted and the error message appears.

Remedy

> Check and, where applicable, correct the line connections or readiness
> to transmit/receive. Check and, where applicable, correct baud rate and parity.

BF 3011 Too many words in the parameter

Operator error

> If a parameter block is created at a programming station or with DNC
> and is read in via the serial interface, no syntax check is carried
> out during the transmission. It is therefore possible for parameter
> numbers to be transmitted which are not in the required format. The
> first time the program is called, the interpreter detects the format
> error and generates the error message. Parameter numbers which do not
> correspond to the format are recognized as incorrect in the editor and
> are rejected.

> or

> A machine dimension larger than M9 has been programmed in the parts program.

Remedy

> Correct parameters which have incorrect syntax (refer to notes on
> parameter punched tape/format check)

> or

> Restrict machine dimension in parts program to M1 – M9, since only
> these nine machine dimensions are available (parameters N751 to N759).

AF 3026 Missing tool ID number or T = zero on tool reference list

General error

> The selected tool identification number does not appear on the tool
> list or parameter address T is not occupied (zero).

Remedy

> For details of tool calculation, please refer to the Operating and
> Programming Instructions: Chapter 6, "Program description", or Section
> 3.4, "Tool data/Tool exchange".

SF 3040   Basic error in the projection data

Hardware error

During each initialization of the control the basic projection data (PRODATS) are examined. The control is waiting for defined characteristics in the adresses. In the event of non-correspondng versions of basic projection data these characteristics are not to find by the control at the defined locations.
So the error message is produced (same in the event of defective Prodat-Eproms).

SWF 3159 Test sum error in the parameter

Software error

The test sum calculation or check on the parameters was faulty.

SWF 3160 Parameter value is invalid

Software error

A format error was committed during the decimal conversion of a value.

AF 3161   Serial interface is currently occupied

General error

External parameter input mode. For example the interface is already occupied due to an external parameter input and is therefore no longer capable of handling a parameter output via the same interface. A further cause of error may be the double actuation of the same softkey (e.g. external parameter input).

Remedy

Repeat the parameter output after the parameter input has been completed.

AF 3162   Unknown parameter block

General error

The parameter called is larger than the max. permissible value (9999). This may have occurred during data transmission via DNC or punched tape, since impermissible values are not rejected during read-in.

Remedy

Check the parameters.

AF 3200   Unknown function code

General error

> An invalid function code has been transferred during the external
> parameter transmission.

Remedy

> Check the parameters.

AF 3201   No value selected

General error

> The softkey "Change value" was actuated without having first selected
> the value to be changed.

Remedy

> Select the value to be changed.

AF 3210   Unknown parameter block

General error

> Parameter input/parameter number mode. The parameter indicated does
> not exist.

Remedy

> The Operating and Programming Instructions (parameter list) provide
> information about the available parameters.

AF 3220   Parameter input blocked

General error

> The parameters cannot be changed during automatic mode or tool
> inspection, as this might produce an undefined operating state.

Remedy

> After completely executing the selected program, switch to the
> operating mode Parameter and then change the appropriate parameters.

AF 3250   Key forbidden here

General error

> Error message appears directly after pressing the key which is not
> allowed.

SWF 3318 NC block in the program memory too long

Software error

The distance between two block end characters exceeds the maximum permissible block length.

SWF 3322 Control character in program memory missing

Software error

The control system automatically generates a block-start/block-end character. If some software error has prevented this, this error message appears.

SWF 3323 Incorrect control character in program memory

Software error

Gap-end characters are found inside a block and replaced by block-end characters which cause this error message.

AF 3324 Program with same number

General error

An attempt has been made to open a program with a program number already stored in the program memory.

Remedy

Determine existing program numbers on the basis of the program list and issue an unassigned program number.

AF 3325 Search word not found

General error

The desired character (chain) was not found in the selected program.

Remedy

Start search at the beginning of the program.

AF 3401 C-axis not implemented

General error

Hardware imperative for the operation of a C-axis do not exist on the machine/control system, although the software for the C-axis option is available (released).

Remedy

Retrofit the C-axis option (hardware).

SWF 3402 EAGK not ready

Software error

Input/output basic card for tool/workpiece measurement or desired value for driven tool is not read for operation.

AF 3403 NC is not configured as central control

General error

The software of the control system must be on an AP186, i.e. on a P186 configured as a central control.

AF 4304 Position-regulated spindle stop not supported by SPS

General error

In the current control system configuration the option position-regulated spindle stop is not supported by the SPS (= SPC, stored-program controller) for the spindle in question. Nevertheless one of the parameters (N517, N557 or N637) is at "1", i.e. the position-regulated spindle stop is selected via parameter.

Remedy

Set appropriate parameter to zero (deselect position-regulated spindle stop). If the option was purchased, contact Gildemeister Service.

AF 3501 DNC option does not exist

General error

The DNC option is not included in the current control system configuration but it can be obtained.

Remedy

Retrofit the option.

SWF 3502 DNC interface not initialized

Software error

The option DNC is available, but the interface has not been initialized. The card is missing, is faulty or is not correctly plugged into the rack. If one or two applies, please call Gildemeister Service.

AF 3503   DNC operating mode in parameter switched off

General error

The parameter N15 allows the release or blocking of DNC operation when the DNC option is available.

Remedy

Depending on the programming of the parameter block N15, the following applies:

0 = DNC switched off
1 = DNC switched on

AF 3504   DNC interface currently busy

General error

A data exchange is taking place via the DNC interface at the moment, therefore it is not available for another use.

Remedy

Try again when data exchange has been completed.

AF 3505   DNC data transmission disrupted

General error

It was not possible to set up a data transmission within a fixed specified time. Cause may rest in a fault data line, for example, or a data protocol which is not permissible.

Remedy

Check the possible sources of errors identified as causes.

AF 3506   Invalid data during transmission request

General error

The format of the data protocol of the master computer is incorrect.

Remedy

Check the data protocol; restart machine if necessary. If the error message recurs, notify Gildemeister Service.

AF 3507   Input abort entered at keyboard

General error

> The operator has used the softkey keyboard to request an abnormal termination of data transmission. Error message is to be regarded as information for the operator.

AF 3508   Output abort entered at keyboard

General error

> The operator has used the softkey keyboard to request an abnormal termination of data transmission. Error message is to be regarded as information for the operator.

SWF 3510 No data in the input FIFO of DNC

Software error

> During DNC terminal operation no further characters have been transferred by FLR during a waiting period (approx. 1 sec). Failure of FLR or connection from FLR to CNC.

BF 3517   Slide number unknown

Operator error

> The slide selected does not exist.

Remedy

> It is only permissible to select the slides which are actually on the machine.

SWF 4010 Unknown function code in manual control mode

Software error

> It was not possible to process the function code "Manual direction keys" which was read in.

Remedy

> The error can be cancelled by changing the operating mode, but the manual direction keys are no longer ready for operation.

BF 4041   C-axis was not swivelled

Operator error

> The C-axis has to be swivelled before the commands Jog C-axis, Override handwheel or Set up C-axis.

Remedy

Swing in the C-axis (M14 or M314).

BF 4042 Spindle does not reach position

Operator error

The spindle cannot reach the programmed position. The problem may be the improperly operating drive amplifier (main drive) or the main drive transmitter.

Remedy

Check the drive amplifier or drive transmitter.

AF 4051 Handwheel already in use

General error

The handwheel cannot be used for two functions simultaneously, e.g. for magnifying lens function and feed override.

Remedy

Deselect the old handwheel assignment and activate the desired new handwheel movement.

AF 4060 Tool number not in permissible range

General error

The specified tool number is outside the permissible range. Max. 64 tools can be calculated.

or

The D-correction has been set to zero in the D-correction active scratch cycle/ handwheel, i.e. the control system cannot allocate the values determined.

Remedy

Rules for handling the 2/4/6-digit tool numbers: Refer to Section 3.4 of the Operating and Programming Instructions.

AF 4062 Only rotational feed allowed; spindle inactive

General error

If a continuous feed is to be carried out in the manual control mode, only a feed in [mm/rev] is permitted. The spindle must move before the direction is selected.

Remedy

Select required feed or speed from menu and then actuate the
continuous feed function again.

AF 4063   No reference point reached

General error

If a machine dimension from the parameters N751 – N759 is to be
approached, the reference point must be approached first.

Remedy

Approach the reference point as described in Chapter 2 of the
Operating and Programming Instructions, "Reference mode".

AF 4064   Parameter input blocked

General error

The parameters are not to be changed during the automatic mode or tool
inspection, since this would produce an undefined operating state.

Remedy

After completely executing the selected program, change to the
parameter mode and change the parameters in question.

AF 4065   Function forbidden

General error

If the operator is in a certain operating sequence which is not related to tool set-up,
an inspection cycle for example, and he changes to the manual control mode, the
following functions are activated in the tool set-up submode:

tool measuring, key
tool change point
set zero point
protective zones N201
shift zero point
back-up tool

This error message then appears, since activating the tool set-up
functions would cause the original operating sequence to be
terminated.

Remedy

Activating the above-named functions is only permitted when the tools
are to be set up. Any preceding operating sequences which are not
related to the set-up process must be completely finished.

AF 4071   Invalid input

General error

Machine dimensions parameters N751 - N759. The M[...] value used in the parts
program is only to be 1 to 9, since it refers to the machine dimensions M1 to M9 which
are stored under parameter blocks N751 to N759.

Remedy

Change parts program accordingly.

BF 4072   Spindle not in position regulated

Operator error

The spindle must be placed in position regulation with the function
M19 before set-up with the handwheel.

Remedy

Press the softkey "M-function" to switch the spindle to position
regulation with M19.

BF 4073   Select spindle

Operator error

Manual control/C-axis mode. Selected C-axis and spindle do not match.

Remedy

The following allocation must be established:

C-axis 1 - spindle H
C-axis 2 - spindle 3

BF 4074   Maximum speed from parameter

Operator error

The programmed C-axis speed is faster than the maximum speed stored in
parameter N126. In this case the control system selects the parameter
speeds and sends the error message.

Remedy

If applicable, increase the speed in parameter N126.

AF 5002 Preset quantity achieved

General error

This message provides operator information. The programmed quantity
has been reached (max. quantity = 9999).

SWF 5100 Unknown function code

Software error

An impermissible code has been read in from the keyboard.

SWF 5201 Handwheel not available

Software error

The attempt to switch the handwheel to a new function resulted in the
discovery that it is already occupied.

BF 5202 Tool correction number unknown

Operator error

The D-correction memory in parameters N1101 to N1180 are accessed with
the aid of the tool correction number. Therefore only numbers from 1
to 80 are possible. D0 means that no D-correction is effective.

Remedy

Change D-number in the program appropriately. Refer to Section 3.4 of
the Operating and Programming Instructions.

BF 5203 No tool inspection when "CYCLE ON"

Operator error

Selecting tool inspection during the cycle ON state would bring about
a program interruption which is probably not intended at this point in time.

Remedy

If a tool inspection is to be performed, the automatic program sequence must first be interrupted with feed stop. Afterwards the unit must be switched over to the automatic parallel mode. Only then is it permissible to select the softkey tool inspection.

BF 5204 No automatic parallel mode tool inspection

Operator error

The unit was not switched over to the automatic parallel mode before selecting the softkey tool inspection.

Remedy

If a tool inspection is to be carried out, the automatic program sequence must first be interrupted by feed stop. Afterwards the unit has to be switched over to the automatic parallel mode. Only then is it possible to select the softkey tool inspection.

AF 5205 Slides have an unequal number of synchronous points

General error

The error may occur when

1) synchronous points in the NC program are skipped due to slide-dependent jump instructions from a slide.

b) the two slides in synchronous cycles (G81, G82, G817, G818, G827 and G828) do not have tools with the same cutting radii.

Remedy

a) Examine program sequence with regard to jump function G61 and its program branches.

b) Examine the tool files (N1001 to N1064) of the tools involved in the synchronous cycles with regard to their tool nose radii.

AF 5207 Traverse path missing during path synchronization

General error

The control system requires a traverse path between two G63 functions.

Remedy

Correct the parts program.

AF 5208 Synchronous coordinates cannot be overtravelled

General error

The control system checks whether a process of synchronization to coordinates is to be conducted during unilateral synchronization. If this is the case and the block currently being processed is the synchronous block, the slide is switched off until the synchronous coordinates are overtravelled. If the synchronous block cannot overtravel the programmed coordinates or the synchronous block has already been processed but the coordinates were not overtravelled, this error message is issued.

Remedy

The function sequence is described in greater detail and with examples in Section 3.2 of the Operating and Programming Instructions.

AF 5211 Not enough storage space

General error

The operator list is fully occupied. Max. 22 programs can be opened (5 slides for each of 4 programs plus one program for the editor and one for the display).

Remedy

Check the program structure. The subprograms opened for the various slides are counted as part of the number of programs. Where applicable, skip a few subprogram calls and put the appropriate program in the main program.

AF 5212 Program missing

General error

User does not exist. The program being sought is not on the user list (directory of all programs in the memory).

Remedy

Check the program memory, perhaps the wrong program number was given.

AF 5213 Not enough storage space

General error

NC parts program memory is fully occupied. This error is output when a new program is to be opened but virtually no storage space is available (less than 17 bytes).

Remedy

> Copy programs not needed onto external data memory or retrofit memory expansion.

AF 5214   Program access blocked

General error

> A program is being accessed from several sides. For example, a program currently being processed in the editor "parallel mode" is called by a main program as a subprogram while being executed in the automatic mode.

Remedy

> Do not process programs which the control system is accessing during the automatic mode in the editor mode at the same time.

SWF 5215 Program access blocked

Software error

> Multiple reading is not possible. This module is already being accessed from another station.

AF 5216   Not enough storage space

General error

> Program directory is full. The maximum permissible number of opened programs is 200. With very large programs, however, it may arise that all the storage space is occupied before the maximum number of programs (200) is reached. The error message is then "Not enough storage space".

Remedy

> Transfer programs to external data medium or retrofit memory expansion.

AF 5218   Program missing

General error

> Program number not found in the parts program memory. An automatic sequence is to be started, but no program has been selected or the one selected does not exist. A nonexistent subprogram may have been called by a main program.

Remedy

> Select a program which is available in the parts program memory and features all necessary subprograms.

AF 5219 Block missing

General error

The selected NC block does not exist in this program. Perhaps the search was not started at program beginning, the program end is missing or the NC block defined as the jump target with the function G61 does not exist.

Remedy

Check the program with regard to these facts.

AF 5220 Program access blocked

General error

A program is running in automatic mode and an attempt is simultaneously being made in the editor to edit in this program.

Remedy

Process the program before or after the automatic run.

BF 5221 Invalid characters in the program memory

Operator error

This error occurs during the operating sequence when the operator makes contradictory inputs. The error may arise, for example, when the softkey "NC word selection" is pressed in the single block mode and then the confirmation key is pressed without first entering an NC word.

Remedy

Closely follow the logic of the individual steps in the operating sequence.

SWF 5222 Program access blocked

Software error

The program directory does not refer to the program contents.

SWF 5223 Invalid characters in program memory

Software error

Faulty block structure; the block number string is not the valid length.

AF 5224 Not enough storage space

General error

Insertion is not possible; the remaining storage space available is insufficient to edit a program or add anything.

Remedy

If the memory capacity is 32 kBytes, it can expanded to 64 kBytes. If it is already 64 kBytes, storage on external data media is advisable.

AF 5225 No block number free

General error

All possible block numbers are in use and/or all available 9999 NC blocks are in use.

The program organization can manage max. 300 NC blocks. This error message also appears if a program with more than 300 blocks is to be restructured.

Remedy

If applicable, change the program structure.

SWF 5226 Invalid characters in the program memory

Software error

The test sum is faulty. The stored program has been damaged. The test sum comparison is reporting an error.

Possible action

Selecting the program again produces a re-interpretation.

AF 5227 Program selection block via edit switch

General error

Automatic mode of operation. The operator can only process the program already selected; intentional restriction of operator options.

AF 5235 Search word not found

General error

Mode of operation: single block, selection of NC word; the code entered (word, letter) was not found.

Remedy

If applicable, repeat search from beginning of program.

BF 5240 Jog mode not allowed when spindle stationary

Operator error

Jog mode cannot be selected when single block is activated.

Remedy

Change over to manual control mode to operate the spindle in the jog mode.

AF 5250 Wrong measuring system

General error

Programming the parameter N1 (measuring system) defines whether the values entered are to be calculated in the metric system or in inches. The programs receive a corresponding internal code. If the program code and the parameter N1 are not the same (different measuring systems), the error message appears.

Remedy

Select the correct measuring system (0 = metric; 1 = inches) under parameter N1.

SWF 5251 Error in program memory

Software error

Faulty memory structure

AF 5252 SPS does not acknowledge the specified workpiece

General error

The SPS (stored-program controller) takes over the counting of workpieces. In this process the SPS must acknowledge the quantity specified by the NC. If it does not acknowledge, this error message appears. This does not generate a program stop or abnormal interruption.

AF 5270 Exchange tool is not being monitored

General error

No tool-life monitoring has been defined for the exchange tool.

Remedy

The type of monitoring has to be defined individually for each tool by appropriately programming the tool-dependent tool-life parameters N1201 to N1264. Here the type of monitoring for original and defined exchange tool must be identical.

For details, refer to Chapter 6 of the Operating and Programming Instructions, parameters N1200 to N1264.

AF 5275 Type of monitoring: exchange tool not equal to original tool

General error

The exchange tool and the original tool have different types of monitoring.

Remedy

Either tool-life or quantity monitoring must be programmed for all tools from a single tool group (original tool and all defined exchange tools), i.e. the type of monitoring must be identical.

For details, refer to Chapter 6 of the Operating and Programming Instructions, parameters N1200 to N1264.

AF 5276   Function cannot be processed

Cause of error

While interpreting the NC program the control has encountered an NC function which it cannot process in this program structure based on the NC program to be processed. Reasons for this include:

1) In order to process the function triggering the error message, other functions must have been executed beforehand in the program run.

2) The function in question can only be called if a function activated earlier in the program run has been completely processed.

3) The function in question cannot be executed repeatedly. First of all it has to be cancelled (e.g. G921 with G981).

Correction of error

Check and correct the criteria named in the NC parts program at the point at which the error message occur.

SWF 5300 Block number digit missing

Software error

A block-start character exists but no block number.

SWF 5301 Block-start character or jump target missing

Software error

No block-start character N exists.

SWF 5302 Unknown NC word

Software error

An unknown NC word has been detected during the preliminary interpretation.

AF 5303   Correction number is larger than 80

General error

The number of tool corrections D is limited to 80 (N1101 to N1180). If a tool file number higher than 80 is accidentally programmed and no D-correction is entered, the control system will automatically search for the D-correction matching the tool (here greater than 80). Due to differing module run times, the error AF 5303 may appear before AF 4060 (tool number not in the valid range).

Remedy

It is only permissible to program tool file numbers up to 64, since the control system only has tool files N1001 to N1064. The D-corrections D1 to D80 (D0 for deselection of D-correction) may be allocated in this process. Check and correct the T-call in the parts system for adherence to rules for the handling of 2/4/6-digit tool numbers and the associated tool file numbers (refer to Section 3.4 of the Operating and Programming Instructions).

AF 5304   Tool place number forbidden

General error

This error message appears if values greater than 255 are entered under tool place number in parameters N1601 to N1664.

Remedy

As a general rule, it is only permissible to program position numbers which also exist on the turret disc (8-, 10-, 12-, and 16-fold turrets). With other tool systems it is possible to program the tool code number here which is managed by the machine adjustment (SPS); refer to data provided by machine manufacturer.

AF 5306 M-number is larger than 999

General error

If a program is created at a programming station or with DNC and is read in via the serial interface, no syntax check is carried out during the transmission. It is therefore possible that M-numbers may be transmitted which are not in the required format. During the first program run the interpreter detects the format error and generates this error message. In the editor, parameter numbers which do not conform in terms of format are recognized as faulty and are rejected.

Remedy

Review the parts program with regard to M-functions, possibly on the basis of the code list. Change if appropriate.

AF 5307 Too many G-words in one block

General error

If a program is created at a programming station or with DNC and is read in via the serial interface, no syntax check is carried out during the transmission. It is therefore possible that a number of G-words (more than 3) may be transmitted which do not fit in the required block format. During the first program run the interpreter detects the format error and generates this error message. In the editor, parameter numbers which do not conform in terms of format are recognized as faulty and are rejected.

Remedy

Distribute the information to be programmed among several of NC blocks.

AF 5308 Too many M-words in one block

General error

If a program is created at a programming station or with DNC and is read in via the serial interface, no syntax check is carried out during the transmission. It is therefore possible that a number of M-words (more than 3) may be transmitted which do not fit in the required block format. During the first program run the interpreter detects the format error and generates this error message. In the editor, parameter numbers which do not conform in terms of format are recognized as faulty and are rejected.

Remedy

Distribute the information to be programmed among several of NC blocks.

AF 5309   Too many subprogram calls in one block

General error

If a program is created at a programming station or with DNC and is
read in via the serial interface, no syntax check is carried out
during the transmission. It is therefore possible that a number of
subprogram calls (more than 3) may be transmitted which do not fit in
the required block format. During the first program run the
interpreter detects the format error and generates this error message.
In the editor, parameter numbers which do not conform in terms of
format are recognized as faulty and are rejected.

Remedy

Distribute the information to be programmed among several of NC blocks.

AF 5310   Subprogram number not indicated

General error

If a program is created at a programming station or with DNC and is
read in via the serial interface, no syntax check is carried out
during the transmission. It is therefore possible that a subprogram
without a subprogram number may be transmitted. During the first
program run the interpreter detects the format error and generates
this error message. In the editor, parameter numbers which do not
conform in terms of format are recognized as faulty and are rejected.

Remedy

Subprogram call L must be followed by a program number (max. 8 digits)
which exists in the program memory. Correct the parts program appropriately.

AF 5311   Subprogram number too long

General error

If a program is created at a programming station or with DNC and is
read in via the serial interface, no syntax check is carried out
during the transmission. It is therefore possible that a subprogram
with a subprogram number which is too long (max. 8 digits) may be
transmitted. During the first program run the interpreter detects the
format error and generates this error message. In the editor,
parameter numbers which do not conform in terms of format are
recognized as faulty and are rejected.

Remedy

Correct parts program and/or parts program memory with regard to this fact.

SWF 5312 NC word content not valid

Software error

Unsuccessful conversion of a string (M- and G-functions) into a binary number.

AF 5313   Neither 1 nor 3 jump targets indicated

General error

Either 1 or 3 jump targets must be indicated when programming the function C61 (conditional/unconditional branches).

Case 1: No jump target was indicated while creating an unconditional branch, e.g. G61 N...

Case 2: No jump target was indicated while creating a conditional branch, e.g. G61 N H(...) N...

Case 3: 3 jump targets were not indicated while creating a conditional branch (jump according to prefix),
e.g. G61 N H(...) N... N... N...

Remedy

The three cases are illustrated with examples in the detailed description of the function G61 in Section 3.2 of the Operating and Programming Instructions.

AF 5314   Unknown NC word in block

Software error

If a program is created at an external programming station or with DNC and is read in via the serial interface, no syntax check is carried out during the transmission. It is therefore possible that unknown NC words may be entered and transmitted. During the first program run the interpreter detects the format error and generates this error message. In the editor, parameter numbers which do not conform in terms of format are recognized as faulty and are rejected.

Remedy

Check the NC word; perhaps it was keyed in incorrectly.

AF 5315 Subprogram nesting level too low

General error

The nesting level of the programs is max. 4 levels

Remedy

Change program structure.

AF 5316 Threading cycle programmed with feed/min

General error

Thread-cutting has been programmed in the program with feed/min (G94).

Remedy

Feed/rev (G95) must be active while creating a thread.

AF 5317 No T-word programmed

General error

No value has been programmed under the address parameter T when programming the function G92.

Remedy

The control system requires a tool number here which may consist of two, four or six digits. For details, refer to Section 3.4 of the Operating and Programming Instructions.

AF 5318 Access to tool file with T00

General error

The invalid tool call T00 was programmed with the function G92.

Remedy

When programming the tool number with two or four digits, the control system uses the tool number to directly access the tool file and the position on the turret. Consequently only the tool files N1001 to N1064 and the turret positions 1 to 12 (16) can be accessed. With identity number programming the control system expects a six-digit input (refer to Section 3.4 of the Operating and Programming Instructions).

AF 5319 Unknown G-function

General error

> A G-function has been called in the program which is not located in the G-function index and is therefore invalid.
>
> or
>
> A G-function programmed in a separate block has been written in a block with other functions.

Remedy

> Chapter 9 of the Operating and Programming Instructions contains a listing of all valid G-functions.

AF 5320 Q-value is larger than 32767

General error

> If a program is created at a programming station or with DNC and read in via the serial interface, no syntax check is conducted during the transmission. It is therefore possible that Q-values larger than 32767 are also transmitted. During the first program call the interpreter detects the format error and generates this error message. During the program input in the editor mode, values which too large are recognized as incorrect and are not accepted by the control system.

Remedy

> Check the program; perhaps a input error is involved.

AF 5321 Tool file number forbidden

General error

> If the parameter N1600 is 0, the tool number T has to be entered in only 2- or 4-digit format. If the parameter block N1600 is 1, a 6-digit tool identity number has to be programmed.

Remedy

> With a 2-digit T entry, turret position equals tool file number, which automatically eliminates the possibility of tool numbers higher than the maximum number of tool positions on the turret disc. If the tool number has four digits, the first two digits identify the tool file (max. 64) and the two others the position on the turret disc.
>
> With a 6-digit tool number entry (identity number programming) the control system expects six positions; these may also include preceding zeroes (refer to Section 3.4 of the Operating and Programming Instructions).

AF 5322 Tool number forbidden

General error

In connection with the function G930 (preselection of chain changer) a six-digit tool number was programmed and the parameter N1600 is zero (no tool identity number).

Remedy

Set parameter N1600 to 1; this selects the 6-digit identity number programming (refer to Chapter 6 of the Operating and Programming Instructions).

AF 5323 Milling cycle option missing

General error

A milling or drilling cycle (G74, G77, G770, G78 or G79) has been requested, however the current control system configuration does not contain this option.

Remedy

If the option was purchased by the customer and mistakenly not released, inform Gildemeister Service and this will be done. This option can generally be subsequently purchased by the customer.

SWF 5324 Text-end character missing

Software error

There is no automatic generated text-start or text-end character.

AF 5326 Tool life has expired

General error

The tool life defined in parameters B1201 to N1264 has expired.

Remedy

Install new replacement tool and increase tool life or define similar tools as exchange tools (refer to parameters N1200 to N1264 in Chapter 6 of the Operating and Programming Instructions).

AF 5327 Tool file number for tool life prohibited

General error

A tool file number higher than 64 has been programmed under the parameter address T.

Remedy

An unambiguous allocation to the tool files (N1001 to N1064) must be established.

AF 5329   NC block number is too long

General error

If a program is created at a programming station or with DNC and is read in via the serial interface, no syntax check is conducted during the transmission. It is therefore possible that block numbers with more than four digits are also transmitted. During the first program call the interpreter detects the format error and generates this error message. During the creation of a program in the editor mode, parameter numbers not in accordance with the format are recognized as incorrect and their input is blocked.

Remedy

Change block number of the externally created program; maximum block number length = 4 (9999).

AF 5330   Too many coordinates in block

General error

The error may occur in the G-functions listed below if a program is created at a programming station or with DNC and is read in via the serial interface. During the first program call the interpreter detects the syntactical error and generates this error message.

G100 X C   In addition, one Z was
G101 X C   programmed for each.
G102 X C I J
G103 X C I J

G110 Z C   In addition, one X was
G111 Z C   programmed for each.
G112 Z C K J
G113 Z C K J

Remedy

Change program read in from the outside. Use the function G1 Z... (frontface) or G1 X... (circumference) for the infeed to the part.

AF 5331   C-axis not swivelled

General error

Before a task is assigned to the C-axis, it has to be swivelled, e.g. tip the C-axis.

Remedy

Swivel C-axis with M14.

AF 5332 C-axis 1 already swivelled

General error

The C-axis 1 has already been allocated to slide 1 or 2 by the $-slide
code and an attempt has also been made to allocate it to slide 2 or 1.

Remedy

The C-axis can only be allocated to one slide at a time for machining.
If the machining is to change from one slide to the other, this is
done with the functions G925 or G985. Before the "transfer" of the C-
axis to another slide, the current C-axis angle is transferred into a
memory in the control system; this cancels the allocation of C-axis to
compound slide. Thereafter M97 is used to synchronize the two compound
slides. Refer to the description of the functions G925 and G985 or the
general comments before function G100 in Section 3.2 of the Operating
and Programming Instructions.

AF 5333 C-axis option not set

General error

A function of the C-axis was selected, however the C-axis option is
not included in the current configuration of the control system.

Remedy

If the option was purchased by the customer and mistakenly not
released, inform Gildemeister Service. The option will then be
released. This option can generally be subsequently purchased by the
customer.

AF 5334 Q-value does not exist

General error

Function G62, "unilateral synchronization". No slide number has been
indicated under the address Q (contains the number of the slide which
is to be awaited).

Remedy

The corresponding slide number is to be indicated under the address Q.
The slide named here must exist on the machine (refer to the detailed
description of function G62 in Section 3.2 of the Operating and
Programming Instructions).

AF 5335 Q-value invalid (> slide number)

General error

> Function G62, "unilateral synchronization". The value indicated for Q (number of the slide which is to be awaited) is larger than the maximum number of slides and is therefore invalid.

Remedy

> Indicate the slide number in question under the address Q. The slide named here must exist on the machine. Refer to the detailed description of function G62 in Section 3.2 of the Operating and Programming Instructions.

AF 5336 U-value does not exist

General error

> Function G62, "unilateral synchronization". A non-existent program was indicated under the address U (number of the subprogram in which the indicated slide is to be awaited).

Remedy

> Indicate under address U the number of the corresponding subprogram. This program has to exist in the main program as a subprogram and has to be in the program memory of the control system (refer to the detailed description of function G62 in Section 3.2 of the Operating and Programming Instructions).

AF 5337 N-value does not exist

General error

> Function G62, "unilateral synchronization". No value has been specified for N (block number which the machining operation of the specified slide Q must reach in the defined program Q so that the slide in whose program G62 was requested will to continue to operate). If a non-existent block has been specified, the error message 5509 appears.

Remedy

> Indicate under address N the number of the corresponding subprogram. The block named here must exist in the program defined under U (refer to the detailed description of function G62 in Section 3.2 of the Operating and Programming Instructions).

AF 5338   No U-number indicated

General error

Function G62, "unilateral synchronization". No value was indicated under address U, the number of the program in which the specified slide Q is to be awaited.

Remedy

Indicate the corresponding program number under address U. The program named here has to exist in the main program as a subprogram and has to be in the parts program memory (refer to the detailed description of function G62 in Section 3.2 of the Operating and Programming Instructions).

AF 5339   Too many U-digits

General error

If a program is created at a programming station or with DNC and is read in via the serial interface, no syntax check is conducted during the transmission. It is therefore possible that program numbers with more than eight digits are also transmitted. During the first program call the interpreter detects the format error and generates this error message. In the editor, program numbers not in accordance with the format are recognized as incorrect and are rejected.

The maximum program number length is 8 digits, therefore U is to have max. 8 digits.

Remedy

Change program number structure; program numbers are to have max. 8 digits.

AF 5340   Enter tool number as 6-digit tool identification

General error

The parameter N1600 is at 1, therefore a 6-digit input is required.

Remedy

Where applicable, fill the identification number entered with zeroes to create a 6-digit string. Refer to Chapter 6 of the Operating and Programming Instructions (parameters N1600 to N1664) or Section 3.4 for general notes on the programming of tool numbers.

AF 5341 Only X or Z permissible

General error

Function G62, "unilateral synchronization". With the function G62 it is also possible to determine, in a defined block, a position in the X-, Z- or Y-direction which must be reached to actuate the wait operation (synchronization). More than one axial direction was named in this case, therefore synchronization is not unambiguously defined.

Remedy

Of the max. available 3 coordinates, X, Z and Y never define more than one. Each of the other two is to be confirmed. For more details, refer to the detailed description of function G62 in Section 3.2 of the Operating and Programming Instructions.

AF 5342 Recursive subprogram call not permitted

General error

No program is permitted to contain itself as a subprogram, e.g. the program %280455 is called as a subprogram in the main program %280455.

Remedy

Check and modify program structure.

BF 5343 Interpreter stop in cycle contour or SRK

Operator error

All G-functions from the 900th range except G908 and G915 generate an interpreter stop. No function with interpreter stop is to be called within a cycle contour or SRK. In addition, it is not permissible to program a G14, T, G92 T or external variable input (i.e. V{?=...} ; V{D...}; V{T...}; V{E...}; V{M...}, since these functions also trigger an interpreter stop.

Remedy

Remove corresponding NC values from the cycle contour.

AF 5344 C-axis 2 already swivelled

General error

C-axis 2 is allocated to slide 1 and an attempt is being made to allocate it to slide 2 as well, or it is already allocated to slide 2 and an attempt is being made to allocate it to slide 1 as well.

Remedy

It is only possible to transfer the C-axis from one slide to the other. Refer to the description of functions G986 and G926 in Section 3.2 of the Operating and Programming Instructions.

AF 5345 C-axis 1 already swivelled

General error

C-axis 1 is allocated to slide 1 and an attempt is being made to allocate C-axis 2 to slide 1 as well, or C-axis 1 is already allocated to slide 2 and an attempt is being made to allocate C-axis 2 to slide 1 as well.

Remedy

It is only possible to transfer the C-axis from one slide to the other. Refer to the description of functions G985, G986 or G925, G926 in Section 3.2 of the Operating and Programming Instructions.

AF 5346 C-axis 2 already swivelled

General error

C-axis 2 is allocated to slide 1 and an attempt is being made to allocate C-axis 1 to slide 1 as well, or C-axis 2 is already allocated to slide 2 and an attempt is being made to allocate C-axis 1 to slide 2 as well.

Remedy

It is only possible to transfer the C-axis from one slide to the other. Refer to the description of functions G985, G986 or G925, G926 in Section 3.2 of the Operating and Programming Instructions.

AF 5347 Check slide deactivation

General error

Case 1.   Operating mode: milling with C-axis 1
C-axis 1 is allocated to slide 1 (2) and an attempt is made to allocate it to slide 2 (1) as well.

Case 2.   Operating mode: milling with C-axis 1
C-axis 1 is allocated to slide 1 (2) and an attempt is made to allocate C-axis 2 to slide 1 (2) as well.

Case 3.   Operating mode: milling with C-axis 2
C-axis 2 is allocated to slide 1 (2) and an attempt is made to allocate it to slide 2 (1) as well.

Case 4.   Operating mode: milling with C-axis 2
C-axis 1 is allocated to slide 1 (2) and an attempt is made to
allocate C-axis 2 to slide 1 (2) as well.

Remedy

It is only possible to transfer the C-axis from one slide to the other. Refer to the description of functions G985, G986 or G925, G926 in Section 3.2 of the Operating and Programming Instructions.

AF 5348   X or Z not programmed

General error

Function G15, Approach starting point. No starting point coordinate has been defined for an existing axis.

Remedy

Program a starting point coordinate in the parts program under the appropriate address for all existing axis slides (including the slides switched off). Refer to Section 3.2 of the Operating and Programming Instructions for a detailed description of function G15.

AF 5349   Q-value invalid

General error

Function G15, Approach starting point. The Q-value must be max. smaller than or equal to 2. If the value is higher or a Q-value has been programmed for only one existing axis slide, this results in an error message.

Remedy

Program either diagonal traverse path (Q = 0), traverse path first in X-direction (Q = 1) or traverse path first in Z-direction (Q = 2). Refer to Section 3.2 of the Operating and Programming Instructions for a detailed description of function G15.

AF 5350   Slide cannot be moved

General error

No axes have been projected for this slide, i.e. the full scope of the control system cannot be utilized due to lack of appropriate machine hardware.

Remedy

Remove all program parts from the parts program which actuate machine equipment not currently included in this machine configuration.

AF 5351   Coordinate not in selected plane

General error

By programming the function G17, G18 (active at program start) or G19,
a plane is selected which does not contain the coordinate in question.
The G17 plane (X/Y plane) for example, has no Z-coordinate.

Remedy

Change address parameters. Refer to the detailed description of
functions G17 to G19 in the Operating and Programming Instructions.

AF 5352   3-D interpolation not possible

General error

In this configuration the control system does not contain 3-D interpolation for feed
movements with G0 or G1 following plane deselection with G16.

Remedy

Divide infeed movement into a number of program steps.

AF 5353   Cycle only permitted for plane of rotation (G18)

General error

Before the cycle begins, a plane (G17 or G19) is selected in which the
cycle cannot be machined, or the machining plane is deselected with G16.

Remedy

Program the function G18 to change over to the plane of rotation
before calling the cycle.

AF 5354   No plane active with G80

General error

G80, cycle end, has been programmed. The selected machining plane must
be active before introducing cycle end with G80.

AF 5355   Interpreter stop forbidden

General error

A G900 to G999 function which triggers an interpreter stop has been
programmed together with G41 or G42 (SRK, FRK).

Remedy

As a basic rule, no interpreter stop is permitted with the functions milling cutter or tool nose radius compensation. Change program.

AF 5356   G41 or G42 only permitted after plane selection

General error

If programming was carried out with G41 or G42, no G16 plane deselection is to be active.

Remedy

Select the necessary machining plane by programming the function G17, G18 or G19.

AF 5357   Calculation of points without dimensions only after plane selection

General error

Although no machining plane is selected (function G16 is active), an attempt was made to work with simplified geometry programming. Only two-dimensional problems can be calculated with simplified geometry programming.

Remedy

The appropriate plane has to be selected with the functions G17, G18 and G19 in order to calculate points without dimensions (see Section 3.2 of the Operating and Programming Instructions).

AF 5358   Circle only permitted after plane selection

General error

It is not permissible to program a G12 or G13 function, circle with absolute centre indication, together with G16, plane deselection.

Remedy

Select plane G17, G18, G19.

AF 5360   Expression larger than 9

General error

Jump function G61, jump distributor. Valid only with transport axes. The variable expression H{?} must no larger than 9. If the variable is calculated, the expression is rounded off.

Remedy

> Refer to function G61 in Section 3.1 of the Operating and Programming
> Instructions for the transport axis system.

SWF 5361 More than 9 jump targets

Software error

> Normally there can be no more than 9 jump targets.

AF 5362   Cut-off control without traverse command (G0, G1)

General error

> The function G991R... has been programmed in a block with no traverse
> command (G0, G1) in the direction of the X- or Z-axis.

Remedy

> Insert a traverse command into the block in which G991 is called.

AF 5363   No speed difference for cut-off control

General error

> No value has been defined for speed difference in parameter N490. The
> control system needs this value as a decision criterion in carrying
> out the cut-off control.

Remedy

> Use the function G992 to define a value for the speed difference.
> Refer to the detailed description of functions G991 and G992 in
> Section 3.2 and the description of parameter N490 in Chapter 6 of the
> Operating and Programming Instructions.

AF 5364   No monitoring time for cut-off control

General error

> No value for the monitoring time has been defined in parameter N491.
> The control system requires this value as a decision criterion when
> carrying out the cut-off control.

Remedy

> Use the function G992 to define a value for the monitoring time. Refer
> to the detailed description of functions G991 and G992 in Section 3.2
> and the description of parameter N491 in Chapter 6 of the Operating
> and Programming Instructions.

AF 5365 Plane change while FRK/SRK active

General error

A machining plane change cannot be effected with active SRK or FRK (G41, G42). It is possible, however, to carry out a plane change in the block in which G41 and G42 are called.

Remedy

Use G40 to deselect SRK/FRK, then change the machining plane and switch tool nose radius compensation (SRK) back on (refer to Section 3.2, "Tool nose radius compensation" or the functions G17, G18 and G19 in the Operating and Programming Instructions).

AF 5366 G40 in block with vertical infeed

General error

Deselection of SRK/FRK is not permissible during vertical infeed.

Remedy

Deselect SRK/FRK before or after the infeed block (refer to Section 3.2, "Tool nose radius compensation" in the Operating and Programming Instructions).

AF 5367 Change planes in the contour cycle

General error

It is not possible to conduct a plane change (G17, G18, G19) during a contour cycle.

Remedy

Check the program (refer to Section 3.2, "Tool nose radius compensation" in the Operating and Programming Instructions).

AF 5368 D-correction is allocated to incorrect slide

General error

It is possible to allocate the 80 available D-corrections to the individual slides. This is done on the basis of the $-allocation. If no allocation is made, the D-correction is permanently allocated to the slide which accesses it first in the program. This error message appears if the slide with fixed allocation is again allocated in the program.

Remedy

Check tool calls with regard to the allocation of the D-correction to the slide and change the program to avoid possible double allocations.

AF 5371   Too many special functions in succession (FIFO too small)

General error

Affects functions G600 to G99. The memory (FIFO) has been constantly
occupied with these special functions over a fixed, specified period
and has therefore not had the opportunity to write in new functions.

Remedy

Reduce the number of special functions from the range G600 to G699
which occur in direct succession.

Note
It is not possible to definitely name a maximum permissible number of
successive functions from this range, since this depends on the
individual structure of the program.

AF 5372   Cycle G862 only permitted for slides 3 and 4

Cause of error

Due to physical arrangement in the working space, slides 1.2 and 5 are not desig-
ned for the machining of cycle G682 (and thus of G864) in the case of GAC
machines.

Correction of error

Process cycle G862 (G864) with slides 3 or 4.

AF 5373   Function can not be processed

Cause of error

The function G64 (intermittend feed) is valid and a C-axis processing has been
programmed. C-axis processing with intermittent feed is not allowed.

Correction of error

Switch off intermittent feed before C-axis processing (programming the function
G64 without the addresses E and F).

AF 5501   Too many programs active

General error

The user list is full. Too many programs/subprograms have been
assigned to a single slide. No more than 6 programs can be opened per slide.

AF 5502 Program missing

General error

> User does not exist. The program sought is not on the user list
> (directory of all programs in the memory).

Remedy

> Check user list (parts program memory); perhaps an incorrect program
> number was used.

AF 5503 Not enough storage space

General error

> The NC parts program memory is full. When very large programs are
> involved, all the storage capacity may be in use before the max.
> number of programs is reached. This error message is output when a new
> program is supposed to be opened, but hardly any storage space is left
> (less than 17 bytes)

Remedy

> Transfer programs to external data media or retrofit a memory expansion.

SWF 5504 Program access blocked

Software error

> Reading access to write program. A program in the editor is called by
> a main program, for example.

SWF 5505 Program access blocked

Software error

> Multiple writing is not possible. This module is already being
> accessed from another station.

AF 5506   Program directory full

General error

The maximum number of opened programs permitted is 200. When very large programs are involved, however, all the storage capacity may be in used before the maximum number of programs (200) is reached. The error message is then "Not enough storage capacity".

Remedy

Transfer programs to external data media or retrofit a memory expansion.

SWF 5507 Program not opened

Software error

The program is not ready to be written to.

AF 5508   Program missing

General error

The program number was not found in the parts program memory, e.g. mode delete editor/program. A non-existent program number has been entered and an attempt made to delete it, or a non-existent subprogram is being called by a main program.

Remedy

Check the program memory. Perhaps an incorrect program number has been named.

AF 5509   Block missing

General error

Operating mode automatic/single block/block selection. The selected block number was not found in this program, a non-existent subprogram is called by a main program, the end of the program is missing or the jump target is missing (G61 function).

Remedy

Review and change the program on the basis of these criteria.

AF 5510 Program access blocked

General error

> Write access to read program. For example, if a program is being
> executed in the automatic mode and an attempt is simultaneously made
> in the editor to edit the program, this error message appears.

Remedy

> Editing can be performed in this program after it has been completely
> executed.

SWF 5513 Invalid characters in the program memory

Software error

> Faulty block structure. The string of block number is not the valid
> length.

AF 5514 Not enough storage capacity

General error

> Insertion is not possible. Not enough storage capacity remains to edit
> a program or to add anything.

Remedy

> If the memory capacity is 32 kBytes, it can expanded to 64 kBytes. If
> it is already 64 kBytes, storage on external data media is advisable.

AF 5515 No block number free

General error

> All available 9999 NC blocks are in use.
> The program organization can manage max. 300 NC blocks. This error message also
> appears if a program with more than 300 blocks is to be restructured.

Remedy

> If applicable, change the program structure.

SWF 5516 Invalid characters in the program memory

Software error

> The test sum is faulty. The stored program has been damaged. The test
> sum comparison is reporting an error.

Possible action

Selecting the program again will generate a re-interpretation.

AF 5527 Block number too large

General error

If a program is created at a programming station or with DNC and is read in via the serial interface, no syntax check is conducted during the transmission. It is therefore possible that block numbers without the required format (e.g. 5 digits) are also transmitted. During the first program call the interpreter detects the format error and generates this error message.

Remedy

Change block number structure so that there are only block numbers with max. 4 digits.

AF 5570 Start block search forbidden with C-axis swivelled

General error

The function start block search has been triggered in a program for C-axis machining although use of this function is not permitted for programs with C-axis machining.

Remedy

Start program from the beginning.

AF 5599 Variable programming option missing

General error

The current control system configuration does not include this option.

Remedy

If this option was purchased by the customer and, due to an error, it was not delivered (released), inform Gildemeister Service. This option can generally be purchased subsequently.

AF 5601 Transition same line / same circle

General error

Case 1:
The same straight line is involved which was programmed in two
traverse paths and therefore in two NC blocks. Since simplified
geometry programming has been utilized, the control system is not
capable of defining starting points or end points of the individual
lines, e.g.:

N1 G1 X? Z? A30
N2 G1 X? Z? A30

Case 2:
The programmed circle elements involve the very same circle, since
the centre points of the circle elements, the radii and the direction
of rotation of circle element 1 ($\pm$ I and $\pm$ K) and circle element 2 ($\pm$
I and $\pm$ K) are identical. The programmed circle elements are therefore
identical and an infinitely number of possible points of intersection
exist for the control system.

Remedy

Review and change the program on the basis of these criteria.

AF 5602 Transition line – line: no point of intersection

General error

No point of intersection can be calculated with the values programmed
under the addresses X, Z and A, for example the connecting coordinates
for the second straight line are not correct or the two lines are
parallel.

Remedy

Check programmed address parameters for mathematic validity and change
them.

AF 5603 Transition line – circle: no point of intersection

General error

The control system cannot calculate a point of intersection using the
programmed address parameters for straight line and circle and/or the
data is contradictory.

Remedy

Check data with regard to mathematic facts.

AF 5604   Transition circle – line: no point of intersection

General error

The control system cannot calculate a point of intersection using the programmed address parameters for circle and straight line and/or the data is contradictory.

Remedy

Check data with regard to mathematic facts.

AF 5605   Transition circle – circle: no point of intersection

General error

The control system cannot calculate a point of intersection using the programmed address parameters for the circular arcs or the data is contradictory.

Remedy

Check data with regard to mathematic facts.

AF 5606   Curvature cannot be defined

General error

The start, centre or end of the curvature cannot be defined.

Remedy

Check the entries under the address parameters with regard to mathematic correctness and logic.

AF 5607   Chamfer only allowed between straight lines

General error

| Case 1 | The path element before and after the desired chamfer has to be a straight line. |
| Case 2 | The chamfer width indicated under address B is longer than the neighbouring path elements. |

Remedy

| Case 1 | Change program. If applicable, program a curvature instead of the chamfer. |
| Case 2 | Correct the chamfer width with regard to the length of the neighbouring path elements. |

AF 5608   SRK not switched off before end of program

General error

The tool nose radius compensation is switched off with the function
G40 when it is no longer required. Note that the control system takes
the SRK into consideration up to the NC block before G40. Only a feed
movement in a straight line with G0 or G1 is to be programmed in the
block with G40.

AF 5610   Radius is zero or negative

General error

The radius of a circle (circular arc) must be positive and not equal
to zero.

Remedy

Correct address parameters for the radius accordingly. Take into
consideration that the programmed circle (circular arc) is not
permitted to be smaller than or equal to the tool nose radius.

AF 5611   Chamfer or curvature must be followed by line or circle

General error

A circular arc was programmed as the element following a straight line
with a chamfer (address B-).

Remedy

A chamfer as a transitional element must always be followed by a
straight line; a straight line or a circular arc must be the element
programmed to follow a curvature.

AF 5612   No difference in X

AF 5613   No difference in Z

General error

There is no difference between starting point and cycle start with the
functions G81 and G82.

Remedy

Modify the starting point slightly.

AF 5620   Unknown G-function

General error

If a program is created at a programming station or with DNC and is
read in via the serial interface, no syntax check is conducted during
the transmission. It is therefore possible that unknown G-functions
may be transmitted to the control system. During the first program
call the interpreter detects the format error and generates this error
message. Unknown G-functions are recognized as incorrect in the editor
and are rejected.

Remedy

The code list indicates the scope of function of the control system
and the meanings of the individual G-functions. Change program
accordingly.

AF 5625   Circles too far apart, no intersection

General error

The starting point and the centres or radii of the circular arcs have
been indicated while programming a contour with two successive
circular arcs. No point of intersection between the two circular arcs
can be calculated with the values defined under the corresponding
addresses since the distance to the centres is further than the sum of
the radii. The circles (centres of circles) are too far apart.

Remedy

Check data with regard to centre of circle and radius and make corrections.

AF 5626   Small circle inside large circle, no intersection

General error

The starting point and the centres or radii of the circular arcs have
been indicated while programming a contour with two successive
circular arcs. No point of intersection between the two circular arcs
can be calculated with the values defined under the corresponding
addresses although the distance to the centres is shorter than the sum
of the radii. Therefore the circles must be inside one another.

Remedy

Check data with regard to centre of circle and radius and make corrections.

AF 5627 Three-circle problem unsolvable

General error

> In the case of a contour with three circular arcs, the control system
> is unable to calculate a connection to a contiguous contour from the
> entries under the address parameters for radii, centres of circles,
> starting points and end points of the individual contour sections.

Remedy

> Check the entries under the address parameters with regard to
> mathematic correctness and logic.

AF 5629 Tangential transition impossible

General error

> Circular arc and straight line are supposed to merge tangentially. The control system
> cannot calculate a tangential transition on the basis of the values for circular arc and
> straight line programmed under the address. Either the indicated address parameters
> are too inaccurate (outside the tolerance range) or they are meaningless.

Remedy

> Precisely calculate the point of intersection and/or check that programmed address
> parameters are realistic, changing them if appropriate.

AF 5630 Contour inadequately defined; supply further values

General error

> The indicated points are insufficient to calculate the programmed
> contour. The end point of the circular arc cannot be ascertained.

Remedy

> Provide the control system in the parts program with more information to calculate
> the contour. If programmed address parameters are not contradictory, excessive data
> on the contour will not result in an error message.

AF 5631 Contour inadequately defined; supply further values

General error

> Only the radius of the circle is known, not its centre.

Remedy

> Specify the coordinates for the centre of the circle under addresses I and K.

AF 5635   Geometry could not calculate radius

General error

On the basis of the data the control system knows only that a circular
arc is involved. Neither radius nor centre of the circle can be
calculated, for example.

Remedy

Use the control system's stock of addresses, where known or
calculable, in the parts program when programming the circular
function. Excessive data will not result in an error message.

AF 5641   Contour has too many elements

General error

Only a specific number of NC blocks is permitted between cycle call
and cycle end, 80 or 50, depending on the type of control system.

Remedy

If the contour to be programmed is too complex to be programmed within
this limited number of NC blocks, it is advisable to call the
identical cycle twice in succession, i.e. the contour to be machined
is divided into two parts.

Note: Insofar as the number of NC blocks is concerned, remember that
additional NC blocks are generated in the control system by the SRK
and when a change is made from one quadrant to the other.

AF 5642   Contour has fewer than 2 points

General error

There is no traverse path between cycle call and end of cycle. After
the cycle call the control system waits for the specification of a
contour to be machined with the aid of the cycle, if a contour cycle
is involved. Otherwise the starting point of the cycle is identical
with the end point of the cycle.

Remedy

A contour table has to be programmed (possibly as a subprogram) in the
case of contour cycles. Pay attention to the selection of the starting
and end point where other cycles are involved.

AF 5645   Too many blocks without traverse path within the contour

General error

The internal memory can only process 20 NC blocks without traverse path within one contour. This error message is generated by a user information text with a stock of more than 20 NC blocks has been inserted into the program while creating it via an external programming unit.

Remedy

Remove extensive user data from the program.

AF 5646   Too many infeeds perpendicular to the selected plane

General error

It is not permissible to program more that nine vertical infeeds to the selected plane.

Remedy

Describe infeed movement with fewer NC blocks.

AF 5647   Groove width narrower than tool nose width + allowance

General error

The tool is wider than the groove less the stock allowance.

Remedy

It is necessary to select a tool whose geometry is suitable for the geometry of the contour to be machine. If a suitable tool has been selected, it may be necessary to change the parameter addresses in the corresponding tool file (N1001 to N1064).

AF 5650   Incorrect position of starting point

General error

In the case of contour cycles G817, G818, G819, G827, G828, G829, G83 and G836, the target point of the first G-command has to be the starting point of the contour, since the first path element in the cycle has been interpreted by the control system as the approach to the contour.

or

Starting point S of cycle G861/863 and end point E of the contour described by this cycle must have the same X-coordinate. In the Z-direction the starting point must be at least 0.1 mm plus a programmable allowance to the right of the contour end.

or

Starting point S of cycle G862/864 and end point E of the contour
described by this cycle must have the same Z-coordinate. In the X-
direction the starting point must be at least 0.1 mm plus twice the
programmed allowance above (internal: below) the end of the contour.

Remedy

Correct the starting point: refer to "Cycles" in Section 3.2 of the
Operating and Programming Instructions.

AF 5652 Infeed missing or too small

General error

More than 30000 infeeds wold be necessary with this infeed dimension.

Correct of error

Check and correct the entry for infeed.

SWF 5654 Cut does not reach contour in the expected direction

Software error

The calculated points of intersection are less than 1.

AF 5655 Starting point for incremental unknown

General error

The control system cannot convert the incremental value into an
absolute value.

Remedy

Starting points of a contour are to be only in absolute dimensions.
Describe the programmed contour with different dimensioning.

AF 5656 Starting point in X or Z unknown

General error

Functions G31, G32, G35, G74, G77, G78, G79, G86, G87 and G88 require
a starting point, but none has been programmed.

Remedy

> Enter starting point as described under the description of functions
> in Section 3.2 of the Operating and Programming Instructions.

AF 5657 Unexpected end of cycle

General error

> A G80 "Cycle end" has been programmed without have first activated a
> cycle in the parts program by programming a corresponding function.

Remedy

> Correct the parts program.

AF 5678 Length of element is zero

General error

> A straight line has been programmed with a known angle but a length of zero.

Remedy

> Check and change coordinates X and Z of the linear movement.

AF 5679 G-function forbidden when SRK is active

General error

> It is not permissible to write the G-function programmed here when SRK
> G41/42 is active.

Remedy

> Change parts program or deselect SRK (G40).

AF 5680 Allowance (G58) too large

General error

> The radius of the internal circle to be traversed is smaller than or
> equal to the total of tool nose radius plus allowance.

Remedy

> Select a smaller stock allowance.

AF 5683   Radius of tool nose larger than or equal to circle radius

General error

The radius of the internal curvature to be traversed is smaller than or equal to the total of tool nose radius plus allowance.

A further cause for this error message may be that G41 has been confused with G42.

Generally, in the case of contours with internal circular arcs, care must be exercised to ensure that the tool nose radius is min. 2 m smaller than the radius of the circular arc, otherwise the contour cannot be calculated on the basis of the tool geometry.

Remedy

Select a tool with a smaller tool nose radius and/or check the SRK function.

AF 5684   The values do not produce a circle

General error

The entries under the address parameters and/or the resultant values do not produce a circular arc.

Remedy

Check and correct address parameters with regard to mathematic criteria.

AF 5685   Straight line: angle not suitable

General error

If the contour runs from the starting points at the indicated angle (address A), the programmed end point is not reached.

Remedy

The addresses X, Z and A have to be programmed so that the end point lies on the straight line described by angle A.

AF 5686   Length of element is zero

General error

Traverse paths with a length of zero are not permitted in simplified geometry programming (VGP) or active SRK (tool nose radius compensation). Two contour elements with identical coverage have probably been programmed.

Remedy

Search in parts program for traverse paths which have been programmed
twice by mistake; make correction.

AF 5687 Position of calculated point is incorrect

General error

The geometry was able to calculate a point, however this is far away
from the preceding and following points; values may be incorrect
(decimal point position incorrect).

Remedy

Check and correct the programmed contour.

AF 5688 Starting point situated outside of circle

General error

The starting point of the circle calculated from the indicated values
is outside the programmed circular arc.

Remedy

Check and correct the address parameters.

AF 5689 End point situated outside of circle

General error

The end point of the circle calculated from the given values is
outside the programmed circular arc.

Remedy

Check and correct the address parameters.

AF 5697 Invalid tool data

General error

The I- and K-values for the tool nose dimensions of the tool in question
are impermissibly large. The max. input value is 50 mm for I and K.

Remedy

Check and correct parameter addresses I and K in the tool files (N1001 to N1064).

AF 5721   Value for P is forbidden

General error

Function G77, G78; P-indexing of a spindle. The address P does not contain the value for the main spindle (0) or the value for one of the auxiliary spindles 1-3.

Remedy

Refer to address parameter P in the description of the function G77/G78 in Section 3.2 of the operating and programming manual.

AF 5722   Incorrect speed or feed

General error

Function G86, "Cycle grooving". If this cycle is to be executed by two compound slides, a dwell time has to be programmed under E or, in the NC block with G86, a speed with G96 or G97 as well.

AF 5723   Cycle forbidden on this slide

General error

This cycle is allocated to an unauthorized slide or a slide which is not on the machine.

Remedy

Cycles can only be machined by compound slides on the machine equipment.

AF 5724   Drill too short

General error

A longer drill has to be used to reach the programmed bore hole coordinates, or the specified coordinates are incorrect.

Remedy

Change drill or change pertinent parameter address in the associated tool file (N1001 to N1064).

AF 5725   No safety clearance or safety clearance too short

General error

Function G74. The safety clearance R of the function is the distance before the beginning of drilling at which the infeed switches from rapid traverse to feed rate. Either no value was entered or the value is too small.

Remedy

Program a positive value under address R as a safety clearance. If applicable, increase the safety clearance.

AF 5726 Speed too low

General error

Function G31, G35. The programmed spindle speed is too low for the exact machining a thread. Furthermore, there must be assurance that the programmed speed is valid for slide or slides which execute the thread-cutting. It may be necessary to program a new speed.

Remedy

Increase the speed or program a new speed.

AF 5727 Incorrect speed or feed rate

General error

Function G35. The control system has calculated the infeed rate with the values of thread pitch F and speed S. The infeed rate is not in the valid range. The parts program contains either no feed rate or no speed or the value for the feed rate or speed was determined on the basis of a variable which is zero or negative.

Remedy

Check and correct feed rate and speed value, particular when assigning values on the basis of a variable.

AF 5728 Value for Q is too large or too small

General error

Function G85, Undercut, or G77/78, PCD.

Case 1 for G85:

The permissible value for Q is between 0 and 4. It indicates the location of the undercut. IF Q = 0, the position of the undercut must be obtainable from tool data I and K in the tool file or from the corresponding addresses of function G92.

Case 2 for G77, G78:

Q is the number of holes. If no hole has been programmed or one hole and just the end angle but no starting angle, an error message is issued.

Remedy

Refer to the descriptions of functions G85 and/or G77/78 in Section 3.2 of the Operating and Programming Instructions.

AF 5729   Value for I missing

General error

Curvature radius or chamfer width is unknown. In the case of functions G79, "milling keyways" (infeed in X-direction), G87, "cycle radius" (radius I) or G88, "cycle chamfer" (chamfer width I), no values have been programmed under the addresses named.

Remedy

Refer to the descriptions of functions G79, G87 and G88 in Section 3.2 of the Operating and Programming Instructions.

AF 5731   Diameter not in table

General error

The table contains only standard diameters. Therefore the selected diameter does not correspond to the stored values.

Remedy

Select a nominal diameter which is listed in the Operating and Programming Instructions in the table after function G35.

AF 5732   Tool nose in I or K equal to zero

General error

The control system requires I and K of the tool to recognize the direction or the tool nose radius.

Remedy

Check that the tool nose values in parameters N1001 to N1064 are correct and make any necessary corrections.

AF 5733   Incorrect tool type

General error

Affects grooving, milling keyways and deep-hole drilling cycles. A tool unsuitable for the work to be performed has been programmed or assigned to the tool file.

Remedy

Change the address parameter WT (tool type) in the tool files used
(N1001 to N1064) to ensure that the tool type selected is proper for
the particular work process.

AF 5735   No free M-function

General error

Functions G77/G78, "PCD cycles", G79, "milling keyways". No more than 3
M-functions can be programmed per NC block. In the case of functions
G77, G78 and G79 the control system automatically generates further M-functions.
If M-functions have been programmed in an NC block with G77, G78 or G79, the
max. permissible number of M-functions (3) per NC block may be exceeded and the
control system generates this error message.

Remedy

It is advisable not to program any M-functions in NC blocks containing the functions
G77/G78 or G79. Program the contents to be described with the M-functions in a
separate NC block.

AF 5736   Grooving depth too shallow compared to grooving width

General error

Function G85, "undercut cycle". As per DIN 76, the ratio of undercut depth to
undercut width is limited. Accordingly, the ratio indicated under addresses K and I
of function G85 is not permissible.

Remedy

Refer to the description of function G85 in Section 3.2 of the
Operating and Programming Instructions.

AF 5737   Thread depth zero or too shallow

General error

Function G31/G32, "thread cycles". Address I or K of the cycle is zero
and therefore not permissible.

Remedy

Correct address parameter I or K.

AF 5738   Grooving width narrower than tool nose width + allowance

General error

Function G86, "cycle grooving". The dimension under address K (keyway width) is smaller than the nose of the selected tool plus stock allowance.

Remedy

Either correct the value under address K or select a tool with a different geometry.

AF 5739   Chamfer depth deeper than grooving depth

General error

It is impossible for the chamfer to be deeper than the grooving.

Remedy

Correct the chamfer depth in relation of the grooving depth.

AF 5740   Incorrect position of the starting point

General error

Refers to functions G31, G32. If in X-direction there is no difference between starting point and end point of the thread, the control system is unable to decide whether to proceed an internal or an external thread.

Remedy

Correct coordinates of the starting point.

AF 5741   No infeed or infeed too small

General error

The error may be the result of:

1st function: G31, "thread cycle", for I, "infeed in X", contains no value.

2nd function: G32, "thread cycle", for K, "infeed in Z", contains no value.

3rd function: G79, "milling keyways", for I, "infeed in X", contains no value or the value is negative.

Remedy

Correct the pertinent address parameter.

AF 5742   Unknown starting point in X and Z

General error

A defined starting point must be indicated for the following functions:

G31, G32, G35	Thread cycles
G74	Deep-hole drilling cycle
G77, G78	PCD cycles
G79	Milling keyways
G86	Cycle grooving
G87	Cycle radius
G88	Cycle chamfer

Remedy

Refer to "G-functions" in Section 3.2 of the operating and programming instructions.

AF 5743   G-function forbidden when SRK is active

General error

When SRK G41/42 is active, writing the G-function programmed here is forbidden.

Remedy

Change the parts program or deselect SRK (G40).

AF 5744   Unknown direction of rotation for driven tool

General error

Functions G77, G78, G79, "PCD/milling keyways". The address J, "direction of rotation of the drive tool", has not been programmed. This results in this error message.

Remedy

If direction of rotation is to be clockwise (CW), program J = 1; if direction of rotation is to be counter-clockwise, program J = 2.

AF 5745   Location in X or Z not exactly defined

General error

Function G85, "undercut cycle". The X- or Z-coordinates for the approach to the cycle start are unknown.

Remedy

Program the start coordinates

AF 5746 Value for A is forbidden

General error

Functions G74, "deep-hole drilling cycle", G77/78, "PCD cycles", G79,
milling keyways". Depending on which function is selected, the following occurs:

G74 The value by which the cutting length is automatically reduced
from one cut to the next is defined under reduction value A. A value
less than zero has been programmed under address A.

G77/G78/G79   The drive spindle for the driven tool has to be
determined under address A.

Remedy

G74 Specify the reduction value.

G77/G78/G79   It is only permissible to program the following values:
0 = main spindle H
1 = auxiliary spindle 1
2 = auxiliary spindle 2
3 = auxiliary spindle 3

AF 5747 Value for P is too small

General error

Function G74, "deep-hole drilling cycle". A value smaller than the
minimum drilling depth W or zero has been programmed under the address
P, cutting depth for the initial drilling cut.

Remedy

A value which is reasonable considering the minimum drilling depth
must be programmed for the first drilling depth (cutting depth for the
initial drilling cut).

AF 5748 Value for P is too large

General error

Function G74, "deep-hole drilling cycle". The end coordinate
in Z is already reached with the value programmed under P (1st
drilling depth), therefore the first drilling cut would result in a
drilling depth deeper than the total drilling depth.

Remedy

Check that address parameters of the function G74 are realistic and correct and make necessary corrections.

AF 5749 Starting position is target position

General error

The starting and target coordinates in the Z-direction are identical.

Remedy

Check the starting and target coordinates; correct them so that a drilling operation is possible.

AF 5776 Chamfer or curvature follows G80

General error

The traverse path programmed in the next NC block after a G80 "Cycle end" is not to contain simplified geometry programming (VGP). Since VGP is triggered by NC blocks in which chamfers or curvatures are programmed under the address B, it is not permissible to program them after the function G80 either.

Remedy

Check and change parts program.

AF 5799 X and Z must be programmed after the preceding element

Operator error

The traverse path following a cycle in the parts program has to contain the coordinates X and Z. Calculations with simplified geometry programming (VGP) are not permitted.

Remedy

The programmer is to calculate the target coordinates of the traverse path and specify them as numerical values under the appropriate addresses.

AF 5850 Contour element too small for allowance + SRK

General error

Tool nose radius compensation. The error may occur when contour elements are so small that they fit into a circle whose radius is equal to the tool nose radius plus G58 allowance.

Remedy

Reduce the size of G58 general allowance, select tool with smaller
tool nose radius if applicable, or change the contour.

AF 5859   Incorrect location of starting point or end point

General error

In the case of longitudinal cycles in the Z-direction the cycle starting point has to
be situated before the entire contour or equal to the contour start; correspondingly
in the X-direction in the case of transverse cycles.

Remedy

Appropriately correct the starting point coordinates (Refer to "Cycles" in Section 3.2
of the Operating and Programming Instructions).

AF 5876   Peak element not parallel to axis

General error

During a grooving cycle (G861, G862, G683 and G684) the contour peak
is not parallel to the axis.

Remedy

When programming these functions take care that the contour elements before and
after the grooving contour are parallel to the axis. There must be a connecting element
parallel to the axis (peak element) where a number of groove valleys are involved.
Correct the programmed contour in accordance with these criteria.

AF 5877   No base elements found

General error

The programmed contour during a grooving cycle (G861, G862, G683 and G684) is
a straight line. Consequently no grooving contour is available and the control system
cannot find any base elements.

Remedy

Check the programmed groove contour and correct it so that an actual groove
exists (refer to Section 3.2 of the Operating and Programming Instructions).

AF 5878   No peak elements found

General error

During a grooving cycle (G861, G862, G683 and G684) a convex contour has been
programmed instead of a groove contour. Consequently no groove contour (concave
contour) exists and the control system cannot find any peak elements.

Remedy

> Check the programmed groove contour and correct it so that an actual groove
> exists (refer to Section 3.2 of the Operating and Programming Instructions).

AF 5879   Too many base elements

General error

> A maximum of 9 contour valleys can be programmed per cycle call.

Remedy

> Where applicable, distribute the contour to be programmed to two cycle calls.

AF 5880   Too many peak elements

General error

> A maximum of ten contour hills can be programmed per cycle call.

Remedy

> Where applicable, distribute the contour to be programmed to two cycle calls.

AF 5881   Synchronous cycle only permitted for 2 slides

General error

> A synchronous cycle has to be processed by two compound slides.
> Possible incorrect slide code, or no slide code where more than two
> slides before the NC block are involved.

Remedy

> If the control system contains more than two compound slides, the
> synchronous cycle call has to be assigned a $-flag for the two slides
> in question.

AF 5883   Synchronous cycle requires several cuts

Operator code

> At least two infeeds have to be possible with synchronous cycles so
> that each of the slides can execute one cut.

Remedy

> Call the cycle which was programmed as the synchronous cycle as the
> standard cycle for only one slide or check whether the contour is realistic.

AF 5885   Amount of infeed missing

General error

No value was programmed under the address I and K while programming a certain cycle. The control system requires these values during this cycle in order to calculate the infeeds required for machining.

Remedy

Enter the required values under address I and K in accordance with the description of the individual address parameters (refer to Section 3.2 of the Operating and Programming Instructions).

AF 5886   G80 missing at end of contour

General error

A cycle which has been called has to be concluded with a G80, "Cycle end".

Remedy

Program the function G80 in a separate NC block after the cycle contour.

AF 5887   Starting point at incorrect location

General error

In the case of contour cycles G817, G818, G819, G827, G828, G829, G83 and G836 the target point of the first G-command must be the starting point of the contour, since the first path element in the cycle is interpreted by the control system as the approach to the contour.

The starting point S of cycle G861/G863 and the end point E of the contour described by this contour must have the same X-coordinate. In the Z-direction the starting point must be at least the minimal amount of 0.1 mm plus a programmable stock allowance to the right of the end of the contour.

The starting point S of cycle G862/G864 and the end point E of the contour described by this contour must have the same Z-coordinate. In the X-direction the starting point must be at least the minimal amount of 0.1 mm plus two times the programmed stock allowance above (internal: below) the end of the contour

Remedy

Correct the starting point: Refer to "Cycles" in Section 3.2 of the Operating and Programming Instructions.

AF 5888   Contour of cycle contains impermissible G-function

General error

The contour contains a G-function which is not included in the sectors
SRK, rapid traverse, straight line, special thread, CW circle or CCW circle.

Remedy

Remove all special functions and M-functions. Only the geometric
contour and the approach to the control are to be shown in the contour table.

AF 5889   Cycle requires contour with SRK

General error

Tool nose radius compensation. The approach to the contour has to be
programmed in the first NC block after the cycle is called, since only
one approach path can be programmed. Furthermore, this first NC block
after the cycle call has to contain the SRK.

Remedy

Depending on the contour, insert the function G41 or G42 in the first
NC block after the cycle is called.

AF 5890   Groove narrower than tool nose width + allowance

General error

Function G86, "grooving cycle". The dimension under the address K,
"keyway width" is smaller than that of the nose of the selected tool
plus the stock allowance.

Remedy

Check the value for keyway width K; where applicable, select a tool
with a narrower cutting tool.

AF 5891   Tool nose in I and K equal to zero

General error

The control system needs I and K of the tool to recognize the
direction or the tool nose radius.

Remedy

Check whether the tool edge values in parameters N1001 to N1064 are
correct. If the tool was called with G92 ... T..., the address
parameters I and K of this function have to be programmed accordingly.

AF 5893 Groove narrower than tool nose width + allowance

General error

Function G86, "grooving cycle". The dimension under address K, "keyway width", is smaller than the dimension of the nose of the selected tool plus stock allowance.

Remedy

Check the value for keyway width K. Where applicable, select a tool with a narrower cutting tool.

AF 5984 Groove does not reach the lowest point (change starting point)

General error

Functions G861/G862. The lowest point of the contour (base point) has not been reached.

Remedy

Change starting point or select narrower cutting tool.

AF 5895 Infeed movement strikes contour unexpectedly

General error

Since the resolution accuracy is 1 m, the cutting tool would be below the contour line after the next infeed.

Remedy

Modify starting point or infeed depth.

AF 5896 Residual material not machined due to tool geometry

General error

The programmed contour cannot be completely machined with the existing tool geometry.

Remedy

Use a tool better suited in terms of its geometry.

AF 5897 Incorrect tool position (I, K of tool nose)

General error

Longitudinal and transverse cycles. I and K of tool nose do not correspond to the selected working direction.

Remedy

Correct tool parameters I and K.

AF 5898 Incorrect tool type

General error

The tool selected is not a button, special, roughing or finishing tool. It is not capable of executing the programmed machining. Tool type allocation under parameter address WT in tool files N1001 to N1064 may be incorrect.

Remedy

Select a tool appropriate for the machining operation under the address WT, with tool input dialogue if applicable (refer to "Tool input dialogue" in Chapter 6 of the Operating and Programming Instructions).

AF 5899 Cutting angle incorrect

General error

Cutting angle A/B not entered or values incorrect or unrealistic.

Remedy

Check and correct addresses A and B of parameters N1001 to N1064.

AF 5900 Start block in contour of G770 is not permitted

Cause of error

The function start block search was used within the contour description of the angle circle cycle G770 in the automatic mode to select a start block. In addition, the function G770 is beeing used in conjunction with work with the C-axis. Under these circumstances a start block within the contour description is not permissible.

Correction of error

Move the start block to an NC block prior to the cycle call and select the NC block containing the cycle call as the start block, or define a suitable start block after the end of cycle (G80)

AF 5951   Feed missing or is zero

General error

Neither "rotational feed" nor "feed per minute" was defined in the
parts program for the machining process or the value programmed under
the address F is zero. In this case the control system can only
execute rapid traverse movements at the feed rate for rapid traverse
defined in the parameter.

Remedy

Depending on the type of feed desired, program a positive value other
than zero under the address F which will be interpreted by the control
system as [mm/min] in the case of "feed per minute" (G94) and as
[mm/rev] in the case of "rotational feed" (G95).

AF 5952   Value for S is zero or negative

General error

In the parts program, a constant cutting speed of the main spindle
(G96) has not been defined for the machining process nor has a speed
for the main spindle (G97), or the value zero has been programmed
under the address S.

Remedy

Depending on the type of machining desired, program a positive value
other than zero under the address S which will be interpreted by the
control system as the constant speed of the main spindle in [mm/min]
in the case of G96 and as the speed of the main spindle [mm/rev] in
the case of G97.

AF 5953   Total time not calculated yet

General error

Involves the "graphic representation option" (utilization graphics).
The relative time share of the slides in the traverse paths is shown in a diagram, etc.
The total time of both slides has to be calculated beforehand. It has no effect on the
machining process in the automatic mode if the parts program is correctly
programmed.

AF 5956   Drill diameter zero or negative

General error

An invalid value was defined for the diameter of the drilling tool under the address
D in parameter blocks N1001 to N1064 or an invalid value was programmed under
address E with the function G92... T... in the parts program.

Remedy

Correct the corresponding tool parameter (N1001 to N1064) or correct function G92 (address parameter E) in the parts program.

AF 5958   Cutting depth negative

General error

Function G25. Performance calculation in connection with the utilization graphics. The value under address B, "G25, cutting depth in mm", is less than zero. When the value is zero, only a warning is issued. It has no effect on the machining process in the automatic mode if the parts program is correctly programmed.

AF 5959   Specific cutting force is zero or negative

General error

The values for the cutting force from parameters N1301 to N1364 are needed for the utilization graphics. The value for the main value of specific cutting force is to be found under address KC; the value for the slope of the cutting force is under Z. The value for KC is zero or negative. It has no effect on the machining process in the automaticmode if the parts program is correctly programmed.

AF 5960   Slope value not between zero and 0.5

General error

The values for the cutting force from parameters (N1301 to N1364) are needed for the utilization graphics. The value for the main value of specific cutting force is to be found under address K; the value for the slope of the cutting force is under Z. The value for KC has to be between zero and 0.5. It has no effect on the machining process in the automatic mode if the parts program is correctly programmed.

AF 5961   Maximum drive power is zero or negative

General error

The main drive power is required to calculate the utilization graphics. The value in parameter N044 is zero or negative. It has no effect on the machining process in the automatic mode if the parts program is correctly programmed.

AF 5962   Efficiency of gear 1 not between zero and one

AF 5963   Efficiency of gear 2 not between zero and one

AF 5964   Efficiency of gear 3 not between zero and one

AF 5965   Efficiency of gear 4 not between zero and one

   General error

   The efficiency of the gears, the ratio between output mechanic power at the chuck
   to the input electric power at the main spindle, is required to calculate the utilization
   graphics; parameters N040 to N043. The value in the parameters is less than zero or
   greater than one. It has no effect on the machining process in the automatic mode
   if the parts program is correctly programmed.

AF 5966   Cutting angle greater than 90° or smaller than -90°

General error

   The cutting angle is required for the utilization graphics. It is entered under the address
   C in parameters N1001 to N1064. The value must be between -90° and +90°. It has
   no effect on the machining process in the automatic mode if the parts program is
   correctly programmed.

AF 5967   Entering angle greater than 180° or smaller than 0°

General error

   Parameters N1001 to N1064. The value under addresses A and/or B must
   be between 0° and 180°.

Remedy

   Check and correct the corresponding parameter block.

AF 5968   Cutting depth zero

General error

   Function G25. Performance calculation in connection with the utilization graphics.
   The value zero has been programmed under address B, "G25, cutting depth in mm".
   In this case, only a warning is issued. It has no effect on the machining process in
   the automatic mode if the parts program is correctly programmed.

AF 5969   Tool number smaller than 1 or greater than 24

General error

   A maximum of 24 materials can be stored in the parameters (N1301 to N1324).

AF 5970 Slide power greater than 100%

General error

The maximum nominal power of the drive is exceeded. The feed rate programmed is too high or the tool selected not suitable.

Remedy

Reduce the feed rate, by means of feed override where applicable, or select a better-suited tool for this special machining operation.

AF 5971 Calculated total time more than 3 weeks

General error

At too low a feed rate the calculated total time per piece would amount to over three weeks.

Remedy

Where applicable, increase the feed rate and/or check and correct the parts program.

AF 5973 Turning centre impermissibly exceeded

General error

The program contains X-coordinates which cause the turning centre to be exceeded. This results in the undershooting of the X-limit, below which the speed can no longer be increased at constant cutting speed. The error message therefore warns the operator that the control system's time calculation unit cannot calculate correct values.

Remedy

If the turning centre is to be purposely exceeded with cutting edge of the tool (caution: danger of collision in case of machines with more than one slide), no constant cutting speed (G96) should be active if the time calculation is to be correct.

AF 5974 Incorrect maximum speed in the parameter memory

General error

The value in parameter N501, "max. permissible speed" is smaller than or equal to zero.

Remedy

Change parameter address MD in parameter block N501. Follow the machine manufacturer's instructions.

AF 5975   Incorrect gear speed limit (parameter)

General error

The value in parameter N502, "max. speed of gear 1-4" is smaller than or equal to zero.

Remedy

Define speed for the individual gears according to machine manufacturer's data so that the maximum speed will not be exceeded.

AF 5976   Incorrect speed limit programmed (in G26)

General error

Function G26, "speed limit". A value less than or equal to zero is under address S. If the speed programmed with G26 is larger than the speed defined in parameter N501, the speed limit from N501 has priority.

Remedy

Program a realistic value under address S of function G26.

AF 5977   Rapid traverse rate of feed smaller than or equal to zero

General error

This error is triggered by the time-per-part calculation of the workpiece time if no value (i.e. the value zero) was entered under the parameter N62, "rapid traverse rate of feed for automatic mode" or if a value smaller than or equal to zero was entered as an external parameter input.

Remedy

Correct parameter N062 accordingly.

AF 5978   Programmed speed less than or equal to zero

General error

The function G97, "constant spindle speed", has been programmed. The value for speed under the address S is zero or was not programmed.

Remedy

Check and correct address parameter S of function G97 in the parts program.

AF 5979   Programmed cutting speed lower than or equal to zero

General error

The function G96, "constant cutting speed", has been programmed. The value for cutting speed under the address S is zero or has not been programmed.

Remedy

Check and correct address parameter S of function G96 in the parts program. The cutting speed V constant can be calculated from the product of speed, the constant Pi and the diameter.

AF 5984   Leave protective zone in +X-direction

AF 5985   Leave protective zone in -X-direction

AF 5986   Leave protective zone in +Z-direction

AF 5987   Leave protective zone in -Z-direction

General error

If the cutting edge of the tool is moved out of the area which is defined by the protective zones, an error message is issued and the slide stops automatically (collision protection for the tool).

Remedy

If the protective zone values programmed under parameter N201 are correct, the traverse paths in the parts program have to be corrected appropriately to permit the cutting edge of the tool to be moved without colliding. If unrealistic protective zone values or no values at all have been specified in parameter N201, change parameter N201 accordingly.

AF 5988   Wear factor not between 1 and 2

General error

A value between one and two can be entered in parameter N13. This value is supposed to represent the degree of wear of the tools.

1 = the tool is new
2 = the tool is unusable

These values are used if the power calculation of the utilizationgraphics is missing. They have no effect on the machining process in the automatic mode if the parts program has been programmed correctly and the tool is in operating condition.

AF 5990 Cutting angle entered for threading tool missing or incorrect

General error

> The cutting angle of the tool (parameters N1001 to N1064, address B)
> is not between 0° and 180°.

AF 5991 Tool nose in I or K equal to zero

General error

> The control system needs the addresses I and K of the tool (location of the centre of
> the tool nose) to recognize the direction or the tool nose radius.

Remedy

> Program addresses I and K accordingly (either parameters N1001 to
> N1064 or the function G92 in the parts program).

AF 5992 Chip thickness is zero

General error

> The control system needs the value of programmed feed (rotational feed) and the
> entering angle to calculate the chip thickness (in conjunction with the utilization
> graphics).

Remedy

> Place appropriate values at the disposal of the control system.

AF 5993 Cross section of chip is zero

General error

> The control system needs the value of programmed feed (rotational
> feed) and the value of one coordinate in transverse direction to
> calculate the chip cross section (using the utilization graphics).

Remedy

> Place appropriate values at the disposal of the control system.

SWF 5994 Reinitialize the graphic representation

   Software error

      Error in the graphic representation area.

AF 6000   Positive limit switch X overtravelled

AF 6001   Negative limit switch X overtravelled

AF 6002   Positive limit switch Z overtravelled

AF 6003   Negative limit switch Z overtravelled

   General error

      While executing a program in the automatic mode the axis slide in question would overtravel the software limit switch in +X, -X,, +Z or -Z direction (parameter N200).

   Remedy

      If the software limit switches are set correctly, the traverse paths in the parts programs have to be checked and corrected. If the parameter address in parameter N200 were not programmed correctly, check and change these appropriately.

AF 6004   No reference point approached

   General error

      The IL1 "interpolation board" needs the reference point as a basis in order to calculate the traverse paths. If a program is to be executed in the automatic operating mode (individual block), the reference points of the individual slides have to be approached in the reference operating mode beforehand, so that the control system can establish a reference between measuring system and slide position.

   Remedy

      In the reference mode, move all existing slides to their individual reference points (refer to information on set-up/manual operation, reference mode).

      Note: If the synchronization of the slides with the measuring system is to be conducted by "setting the zero point" in the manual control mode, set parameter N3 to 1; this deselects the reference run procedure.

AF 6005 Axis X is on reference cam

AF 6006 Axis Z is on reference cam

> ### General error
>
>> The axis in question is positioned on its reference cam and no
>> automatic moving clear procedure is projected.
>
> ### Remedy
>
>> The axis slides have to be moved clear manually. To do so, proceed as follows:
>>
>> 1. Press the delete button
>> 2. Select the manual control mode
>> 3. Use the manual direction key to move the slide in question away
>>   from the reference cam
>> 4. Repeat the reference run.

AF 6008 Feed rate is zero

> ### General error
>
>> The feed rate for the manual direction keys with spindle stationary is defined under
>> the address F in parameter N60. This parameter has not been assigned a value or no
>> feed rate was specified during manual control with spindle moving.
>
> ### Remedy
>
>> Define an appropriate feed rate under address F in parameter N60. If the slide is to
>> be moved in the manual control mode with spindle moving, use the softkey menu to
>> specify a feed rate beforehand.

AF 6009 Exiting protective zone in +X direction

AF 6010 Exiting protective zone in -X direction

AF 6011 Exiting protective zone in +Z direction

AF 6012 Exiting protective zone in -Z direction

> ### General error
>
>> A traverse movement of the tool's cutting edge, which is outside the protective zone
>> defined in parameter N201, is to be executed in an NC block of the program started.
>>
>> The tool would run an elevated risk of collision upon moving outside
>> the protective zone.

Remedy

Check the program and make changes where applicable. If the protective zones are not set correctly under parameter N201, these can be changed when the situation has been carefully analysed.

AF 6013 Feed rate too high

General error

The selected feed rate (in mm/rev) is too high in relation to the speed.

Remedy

Check and correct feed rate and speed.

AF 6014 Reference run forbidden

General error

Parameter N3 is at 1. The zero point of the NC coordinate system is defined by "scratching" in manual control. If the parameter is at 0, the reference run is permitted and has to be executed.

Remedy

Either define the zero point of the NC coordinate system by "scratching", as described for the manual control mode in Chapter 2 of the Operating and Programming Instructions or change the parameter N3 (N3 = 0).

SWF 6015 Interpolation sector exceeded

Software area

Error in the control system (interpolator and position regulator)

AF 6016 Incorrect position correction values

General error

e.g.: - difference between K1 and K2 (reversing clearance) is greater than 1 mm
       - measured position values entered incorrectly
       - position grid too small (must be more than 5 mm)

Remedy

Refer to parameters operating mode (N298 to N380) in Chapter 6 of the Operating and Programming Instructions. Normally these parameters only have to be set once, when commissioning the machine, and have been programmed by the machine manufacturer. Should any problems arise, it is advisable to inform the machine manufacturer.

AF 6017   Lag error overflow on X-axis

AF 6018   Lag error overflow on Z-axis

General error

The axis was not able to traverse the programmed path. The resultant lag error accumulated until overflow occurred (lag overflow limit = 32 mm) and the control system generated the EMERGENCY OFF state. Possible reasons that the axis cannot traverse include mechanical obstruction or an error in transmitting of axis sensor signals.

Remedy

Determine and eliminate the cause of the problem. Afterwards continue the machining operation as described in "Restart after EMERGENCY OFF" in Chapter 5.

AF 6019   Short circuit or broken cable in measuring system, X-axis

AF 6020   Short circuit or broken cable in measuring system, Z-axis

AF 6021   Short circuit or broken cable in measuring system, main drive

AF 6022   Measuring system for X-axis not working properly

AF 6023   Measuring system for Z-axis not working properly

AF 6024   Measuring system for main drive not working properly

General error

This error stems from the IL alarm routine in which the alarm reports from the measuring systems and the sensor monitor are handled. Interrupts can be caused by a transient fault. The control system generates an EMERGENCY OFF if the fault signal persists for a longer time (static disturbances).

The dynamic disturbances, which last a very short time, do not generate an EMERGENCY OFF. A counter was installed for this purpose. It is interrogated for its limiting value (4); if it is over 4, this error message is issued.

The sensor signals arriving in the control system are disrupted or faulty. The reason may be:
>   broken cable,
>   intermittent short circuit,
>   supply voltage of sensors faulty,
>   the sensor itself
>   the plug-in system for the sensor line.

Remedy

Check the above causes and make corrections as necessary. If the EMERGENCY OFF state exists, proceed in the machining operation as described in Chapter 5, "Restart after EMERGENCY OFF".

7-117

SWF 6025 ILG missing or faulty

    Software error

        Hardware initialization is faulty.

AF 6026   Slide cannot be traversed in X-direction

    General error

        There is no X-axis.

    Remedy

        Program only movements for this slide which do not contain any X-coordinates.

AF 6027   Slide cannot be traversed in Z-direction

    General error

        There is no Z-axis.

    Remedy

        Program only movements for this slide which do not contain any Z-coordinates.

AF 6028   Invalid slide selected

    General error

        The slide selected is not installed or not projected.

BF 6029   Slide switchover key currently inoperative

    Operator error

        The slide switchover key will not operate during the reference run.

    Remedy

        Switch over to the desired slide after the reference run.

BF 6030   Reference cam already tripped

    Operator error

        The reference cam for the axis in question has already been tripped.

AF 6031   AP board missing or faulty (1)

AF 6032   AP board missing or faulty (2)

AF 6033   AP board missing or faulty (3)

AF 6034   AP board missing or faulty (4)
General error

> Control system is polled during run-up to determine whether the board is present and, if so, whether this is permitted and/or whether the board reported appropriately.

Remedy

> Where applicable, check whether the AP board is correctly inserted in the rack. If so, please contact Gildemeister Service.

AF 6036   Machine dimension is not within range

General error

> The machine dimension stored in parameters N751 to N759 would trigger a traverse movement which would exceed one of the limiting values defined in parameter N200.

Remedy

> Check and correct the parameter address of parameter blocks N751 to N759.

AF 6037   Positive limit switch Y overtravelled

AF 6038   Negative limit switch Y overtravelled

General error

> During the execution of a program in the automatic mode the axis slide in question would travel past the software limit switch in +Y or -Y direction (parameter N200).

Remedy

> With the software limit switches set correctly, check and correct the traverse paths in the parts program. If the parameters have not been programmed correctly in parameter N200, check them and make appropriate changes.

AF 6039   Exiting protective zone in +Y direction

AF 6040   Exiting protective zone in -Y direction

General error

> A traverse movement of the tool's cutting edge, which is outside the protective zone defined in parameter N201, is to be executed in an NC block of the program started.

Elevated risk of collision would exist if the tool were moved outside the protective zone.

Remedy

Check the program and make changes where applicable. If the protective zones are not set correctly under parameter N201, these can be changed if the situation is carefully analysed.

SWF 6042 Slide cannot be traversed in Y-direction

Software fault

Projection data for machine incorrect; the Y-axis is not projected.

SWF 6043 Incorrect Kv factor

Software error

Incorrect projection data; the Kv factors exceed the min. or max. values.

AF 6044 Too many position corrections

General error

The number of values stored under address X of parameter N298 must be less than or equal to 80.

Remedy

Refer to parameter N298 in Chapter 6 of the Operating and Programming Instructions.

AF 6048 Incorrect slope calculation

General error

When an FMS system exists; the parameters N3000 to N3011 for position regulation are incorrect.

Remedy

> For information on the control of parameters N300 to N3011, refer to Section 2.1, "Transport axis system", in the Operating and Programming Instructions.

AF 6049 D/A converter value not within range

General error

> The digital/analog converter for nominal value output (parameter N164) is being assigned a value which is too high or too low. The value has to be higher than zero and lower than ten. If the error occurs in connection with a transport axis system, the error message refers to parameter N3012.

Remedy

> Check and correct the parameter in question.

AF 6050 Lag error limit not within the range

General error

> The value of the parameter N165 for the lag error limit is too high or is zero. The value specified here has to be higher than zero as the control system would otherwise be unable to execute any traverse paths in Y-direction, however it can be no more than 32 mm. If the error occurs in connection with a transport axis system, the error message refers to parameter N3013.

Remedy

> Check and correct the parameter in question.

SWF 6051 AP board cannot be started

Software error

> AP186 board cannot be actuated via software or hardware.

SWF 6052 Feed override is zero

Software error

> The selected feed override either has a value of zero or is higher than the permissible maximum.

AF 6053   Incorrect AP parameters

General error

Affects the transport axis system; the parameter values entered for the Kv factor or lag error are outside the permissible range.

Remedy

Check and correct parameters N3000 and following. Refer to Section 2.1 of the Operating and Programming Instructions for the transport axis system.

AP 6054   AP fixed stop, parameters too small

General error

Case 1: Lathe

The scope of the tolerance range for comparing two successive actual values for path monitoring when travelling the fixed stop are defined under parameter N722, i.e. if two successive actual values for location are within this range, the axis drives are stopped. The range extends from 1 μm to 99 μm.

Case 2: Transport axes

Text and parameters as above, except that the range extends from 1 μm to 999 μm.

Remedy

Increase the parameter value if applicable (refer to Chapter 6 of the Operating and Programming Instructions).

AF 6055   Impossible to set reference point

General error

This error message affects the transport system

Case 1: The axis does not exist.

Case 2: A reference point can only be set manually for the axes which do not interpolate with one another, i.e. the slides here can only move in one axial direction.

Remedy

Refer to Section 2.1 of the Operating and Programming Instructions of the transport axis system.

AF 6056   Target point cannot be reached

General error

Due to a disturbance (e.g. mechanical obstruction) the slide was unable to reach the position programmed in the parts program, i.e. a static lag error exists which is below the lag error limit of 32 mm (from this point onward the control system generates EMERGENCY OFF), but can damage to the drive motors due to thermal stress generated in the attempt to reach the nominal value of the control system.

If a static lag error of min. 0.3 mm exists at the X- or Z-axis in the automatic or individual block modes for a period of 10 seconds and more following a path-finished message in the control system, the control system disconnects the drives from the mains by generating the EMERGENCY OFF state.

Remedy

Determine the cause of the disturbance and eliminate it. Afterwards, continue the machining operation as described in "Restart after EMERGENCY OFF".

AF 6057   Overtraval negative limit switch E

AF 6058   Overtravel positive limit switch E

General error

While executing a program in the automatic mode, the axis slide in question would overtravel the software limit switch in +E or -E (parameter N200).

Remedy

With the software limit switches set correctly, check and correct the traverse paths in the parts program. If the parameters have not been programmed correctly in parameter N200, check them and make appropriate changes.

AF 6061   Exit protective zone in +E-direction

AF 6062   Exit protective zone in -E-direction

General error

A traverse movement of the gripping attachment is to be executed in an NC block of the started program and this movement is outside the protective zone defined in the parameter N201.

If the protective zone were to be exceeded, the gripping attachment would be subject to elevated risk of collision.

Remedy

Check the program and make changes if necessary. If the protective zones are not correctly set under parameter N201, they can be changed while carefully checking the situation.

AF 6313 Switched-off slide not at starting point

General error

The parameter N9 is at 1. The selected slide is switched off in the automatic mode. If the control system, after the softkey automatic mode has been selected, determines that one or more slides are switched off and they are not at the starting points programmed in the parts program with G15, this error message is issued. If a switched-off slide is already at the coordinates programmed in the NC block with G15, no further error message is issued.

Remedy

Either exit the automatic mode or start the program after pressing "Cycle ON". In this case the switched-off slides are moved to their respective starting points (see function G15 in Section 3.2 of the operating and programming instructions).

AF 6314 No G15 programmed

General error

The parameter N17 is at 1, "start at starting point", but no starting point was defined at the start of the parts program with G15.

Remedy

A starting point has to be given with G15 as the first command in every parts program for every slide in every existing axial direction. If there is no G15 at the beginning of the program or the start coordinate for one axis is missing, this error message is issued. The automatic sequence can only be started if the starting point has been programmed for every slide axis, including the slides switched off with parameter N9.

AF 6318 Exact stop not achieved

General error

It was not possible to approach the programmed exact stop position even after an internally set idle time elapsed, since a mechanic problem (e.g. sticking slide) blocks the approach to the position.

Remedy

Determine and eliminate any mechanical disturbance.

AF 6401   Mathematic term too complicate

General error

The mathematic term (expression) exceeds the capacity of the main
memory or the number of planes.

Remedy

Simplify the term (expression) if possible.

AF 6402   Brackets not in pairs

General error

There is not an equal number of bracket-open and bracket-closed characters.

Remedy

If round brackets are used in mathematic terms and informational texts
or square brackets where expanded NC functions are involved, there
must be an equal number of left brackets and right brackets.

SWF 6403 Right curly bracket missing

Software error

Error in the control system

AF 6404   Equal sign missing

General error

The control system expects an equal sign at this point.

Remedy

Check the mathematic situation and makes corrections, e.g. in variable allocation, etc.

AF 6405   Function unknown

General error

The function transferred to the control system could not be identified, i.e. an incorrect
or unknown arithmetic function has been called.

Remedy

The only permitted arithmetic functions are PLUS, MINUS, ABS, TAN, ARCTAN, COS, ARCCOS, EXP, INT, LN, ROUND, SGN, SIN, ARCSIN and SQRT.

AF 6406   Square root of negative number

General error

The program contains a negative value for which a root is to be found. The control system cannot calculate a square root of a negative number.

Remedy

Check that the pertinent mathematic operations are correct.

AF 6407   Argument too small

General error

The natural logarithm LN is to be formed from a value which is smaller than 0.0001.

Remedy

Check and correct the mathematic term which is supposed to calculate the argument from which the natural logarithm is to be formed. The natural logarithm is not defined for arguments which are zero or negative.

AF 6408   Division by zero

General error

A division by zero is to be executed in a mathematic term. A division by zero cannot be performed.

Remedy

Check and correct mathematic term.

AF 6409   Exponent too large

General error

The largest exponent to be processed must be smaller than or equal to 20.

Remedy

Check and correct mathematic term.

AF 6410 "=" forbidden

General error

The equal sign is forbidden at this point of a variable allocation.

Remedy

Refer to rules for allocation of variables in Section 8.1 of the Programming Instructions.

AF 6411 Function too long

General error

A symbolic function name is not to be over seven characters long.

AF 6412 Variable value too large

General error

The variable value has to be smaller than 1019.

Remedy

Check and correct variable allocation.

AF 6413 Variable number too large

General error

The range of variable numbers extends from 1 to 999.

Remedy

The variable V1 to V299 are at the user's free disposal. The other variables have a meaning which is permanently specified by the control system manufacturer (refer to variable programming, assignment of variables in Section 8.1 of the Operating and Programming Instructions).

AF 6414 NC syntactical error

General error

Error in construction of NC block; the required forms have not been used.

Remedy

Check and correct the NC block in question.

AF 6415   Variable not yet defined

General error

A variable which has not yet been assigned a value is to be accessed during the program, e.g. V11 = V21 + 10.

The variable V21 was not previously assigned a variable, therefore the control system generates this error message.

Remedy

Make certain that all variables accessed during the program are assigned a value first (refer to variable programming, assignment of variables in Section 8.1 of the Operating and Programming Instructions).

AF 6416   Empty operand

General error

No value has been assigned to the operand, e.g.; V11 = (3 + ...).

Remedy

Check terms used for mathematic logic and make changes.

AF 6417   Option: variable programming missing

General error

The variable programming option has not been released in the software.

Remedy

If the option was purchased by the customer and mistakenly not released, inform Gildemeister SERVICE. Option can also be purchased subsequently.

AF 6419 The value of the tangent, sine, cosine is no longer within the permissible range

General error

An attempt has been made to calculate the arc sine or the arc cosine from a value smaller than -1 or larger than +1. Neither function is defined for these values, therefore the control system generates this error message.

An attempt was made to calculate the tangent for one of its poles, e.g. tangent of 90°.

Remedy

When forming mathematic terms, comply with the defined range of the mathematic functions used.

AF 6423 Invalid number for external event

General error

According to function G91, "conditional branch", only the event E90 [1] can be interrogated at the moment.

Remedy

The function implemented until now is:

E90 [1]   = 1 Control system is in "start block search" state
           = 0 Control system is in normal block processing state

AF 6424 Text or variable input too long

General error

Per line, max. 48 characters are at the operator's disposal.

AF 6425 No variable transfer can be executed

General error

The variable transfer between SPS, FMS and NC is disrupted. The NC is attempting unsuccessfully to transfer data. The cause of this error message may be a faulty board, for example, or the variables have not been released due to an incorrect transfer mode in the projection data.

Remedy

Check the board and consult Gildemeister Service if necessary.

AF 6426   Serial interface currently busy

General error

> The selected interface is currently busy due to another output (e.g. printer/punched-tape reader, data transmission from a programming station, etc.).

Remedy

> Wait until the interface is free again.

AF 6428   Transfer variables blocked

General error

> Transfer variables are variables which communicate with the SPS, FMS and NC. This error message appears if the transfer variables in the projection data in the range of the variables V701 to V719 are not released but are called in the program nevertheless.

Remedy

> The cause of the error may be the transport system or the SPS.

AF 6901   Angle change > 180 degrees during frontface machining

General error

> Note that moving the tool beyond the turning centre is not permitted.

Remedy

> Movements which exceed the turning centre in the X-direction therefore have to be stopped in the centre and continued in the opposite direction following a 180° rotation of the work spindle. Check the program and make changes (refer to frontface machining in Section 3.2 of the Operating and Programming Instructions).

AF 6902   The values do not produce a circle

General error

> The radius calculated from I and J or K do not correspond with the one from the interval of the programmed circular arc end point, calculated from starting point, I and J or K of the centre of the circle. The calculated deviation is not to be more than the product of 0.05 times the radius calculated from I and J or K.

Remedy

Refer to frontface or circumferential machining in Section 3.2 of the Operating and Programming Instructions (G102, G103, G112, G113).

AF 6903 Lag error in C-axis

General error

A lag error of the C-axis may be caused by the C-axis drive not achieving the programmed angular speed or because incorrect programming, improper or incorrectly defined tool result in such a high load that the C-axis motor can no longer reach the required position.

Remedy

Check and make any necessary corrections in parts program and utilized tools or C-axis drive.

AF 6906 Feed rate is zero

General error

This error message is issued when the function G101 is called but no feed rate has been programmed beforehand.

Remedy

Use the function G94 to define a feed rate for milling.

AF 6950 Lag error overrun in Y-axis

General error

The Y-axis has accumulated an error which is larger than the lag error window defined in parameter N165 and the control system generates the EMERGENCY OFF state. This is caused by mechanical problems (e.g. obstructions, sluggishness, etc.) such as an improperly functioning drive motor, amplifier or pulse generator which prevent the prevent the position from being reached.

Remedy

Determine and eliminate the cause of the disturbance. Afterwards continue in the machining operation as described in Chapter 5, "Restart after EMERGENCY OFF".

AF 6951   Impermissible Y-axis controller parameters

Operator error

Incorrect Kv factors, parameters N161, N162, N163, or the parameter N165, "lag error window", is zero. A value greater than zero has to be entered, otherwise the control system cannot travel any traverse paths in the Y-direction. The parameter N164, "DAC limit", may be a value greater than zero and smaller than ten.

Remedy

Enter new parameter values (not zero), then switch off the control system and let it run up again (refer to the parameter description in Chapter 6 of the Operating and Programming Instructions).

AF 6952   Y-axis cannot be moved during rotational feed (G95)

General error

The Y-axis can only be traversed with G94, "feed per minute" [mm/min]. However a rotation-based feed with G95 [mm/rev.] is active in the program.

Remedy

The Y-axis can only be moved at feed per minute. Program the function G94 before machining involving the Y-axis.

AF 6953   Contour element too small for feed

General error

When machining circular arcs in conjunction with the Y- or C-axis, the control system breaks the programmed arcs down into a number of linear movements (straight lines) to approximate a circular movement. The accuracy of the approximation depends on the feed rate and decreases in accuracy as the feed rate increases, particularly when circular arcs with a small radius are concerned. The limiting case (fault) occurs when it is no longer possible to approximately machine a circular arc with a small radius.

Remedy

Reduce the feed rate.

SWF 7000 Unknown error. Error number not in range of error numbers

Software error

For example the board has not been initialized, the error number is beyond 9999 or offset contact is missing.

AF 7033   Test of misalignment not successful

General error

Function G906, "Misalignment". With angular synchronous run switched on and the chuck closed, an angular difference greater than the value stored in the control system has formed between the two spindles within a predetermined time.

Remedy

The error may be produced by a cutting chuck which does not close with the required clamping pressure for example. Where applicable, check the cutting chuck.

AF 7034   No angular synchronous run selected

General error

The function G906, "measure the misalignment between guiding and guided spindle", has been programmed without having first programmed the M-function M803, "angular synchronous run".

Remedy

Use the function M803 S... to activate the angular synchronous run (refer to the description of the function M803 in Section 3.6 of the Operating and Programming Instructions).

AF 7103   Target speed not reached

General error

The programmed target speed was not reached within the defined time window. The cause may be:

1.   faulty drive actuator/motor
2.   demand for drive performance too high
3.   nominal speed of drive is lower than target speed
4.   the time window selected is too small.

Remedy

Check and correct the tolerance value of spindle speed in parameter N506.

AF 7111   No spindle synchronization for cut-off control

General error

The function G991 (cut-off control) can only be programmed in conjunction with the spindle synchronization. During the cut-off operation this function utilizes the rotational speed or angular velocity of the two synchronous spindles to check whether a workpiece has been completely separated from the blank.

Remedy

Activate spindle synchronization (M801 to M803) (refer to Section 3.6 of the Operating and Programming Instructions).

AF 7105   Warning - SPS not operating correctly

Operator error

The NC cyclically monitors for proper operation of the SPS (machine adjustment). If the SPS does not report to the NC within a defined time, this warning is output.

Remedy

The operator has no possibility for control. Proper functioning of the SPS is no longer ensured. Notify GILDEMEISTER Service if the warning is repeated.

AF 7106   EMERGENCY OFF, failure of SPS

General error

The NC cyclically monitors for proper operation of the SPS (machine adjustment). If the SPS does not report to the NC within a defined time, this warning is output.

If the warning is output several times in direct succession, reliability of operation is no longer ensured and EMERGENCY OFF is triggered.

Remedy

The operator has no possibility for control. Proper functioning of the SPS is no longer ensured: Notify GILDEMEISTER Service.

AF 7125 DI64 board is missing or faulty

General error

The digital input board (DI64) for tool fracture recognition cannot be operated via software. The board is missing, faulty or not correctly inserted in the rack.

Remedy

Check that board is seated correctly; correct if necessary.

BF 7126 Tool fracture reported

Operator error

The sensor and the monitoring unit have detected a tool break.

Remedy

The tool in question has to be exchanged.

SWF 7127 Unknown DI64 interrupt

Software error

The sixth DI64 board is used in connection with tool fracture recognition and has interrupt capability. This error number encompasses all interrupts not generated by the tool fracture bit.

AF 7129   Remaining path deleted by SPS

Cause of error

For safety reasons, the machine interface (SPS) has requested the deletion of the remaining traversing path. One possible reason for this may be a collision, the breaking of a tool or the overloading of a servo-motor. It is not possible to continue the NC program processing by pressing "CYCLE ON".

Correction of error

Exit the AUTOMATIC operating mode. Pin-point the cause responsible for the reaction of the SPS and correct the fault at the machine. The NC program can then be restarted.

BF 7130   Swinging in the C-axis prohibited when spindle moving

Cause of error

An attempt is being made in the MANUAL CONTROL mode (checking device set-up mode) to swing the C-axis in while the spindle is moving.

Correction of error

First stop the spindle and then swing in the C-axis.

SWF 7200 No answer from SPS during initialization

Software error

The SPS board does not receive a check-back signal indicating whether the initialization was successful. The board is faulty or not installed.

AF 7201   No acknowledgement of M- or T-function from SPS

General error

The NC software passes the programmed M- or T-functions to the SPS (machine adjustment); the SPS acknowledges receipt. If this confirmation of receipt has not been received after a certain time, the execution of the program is interrupted.

AF 7206 Swivelling of C-axis forbidden while spindle is moving

General error

An attempt is being made to swivel the C-axis while main drive is running.

Remedy

Swivel C-axis only while spindle is stationary, i.e. spindle stop has to be programmed before swivelling the C-axis.

AF 7207 C-axis 1 already swivelled

General error

C-axis 1 is allocated to slide 1 and an attempt is being made to allocate C-axis 2 to slide 1 as well, or C-axis 1 is already allocated to slide 2 and an attempt is being made to allocate C-axis 2 to slide 1 as well.

Remedy

It is only possible to transfer the C-axis from one slide to the other. Refer to the description of the functions G985, G986 or G925, 926 in Section 3.2 of the Operating and Programming Instructions.

AF 7208   C-axis 2 already swivelled

General error

> C-axis 2 is allocated to slide 1 and an attempt is being made to allocate C-axis 1 to slide 1 as well, or C-axis 2 is already allocated to slide 2 and an attempt is being made to allocate C-axis 1 to slide 2 as well.

Remedy

> It is only possible to transfer the C-axis from one slide to the other. Refer to the description of the functions G985, G986 or G925, G926 in Section 3.2 of the Operating and Programming Instructions.

AF 7209   C-axis 1 already swivelled

Remedy

> C-axis 1 is allocated to slide 1 and an attempt is being made to allocate it to slide 1 again, or it is allocated to slide 2 and it is to be allocated to slide 2 again.

Remedy

> Remove second allocation of the C-axis from the program.

AF 7210   C-axis 2 already swivelled

Remedy

> C-axis 1 is allocated to slide 1 and an attempt is being made to allocate it to slide 1 again, or it is allocated to slide 2 and it is to be allocated to slide 2 again.

Remedy

> Remove second allocation of the C-axis from the program.

AF 7211   Spindle position undefined

General error

> The spindle has been stopped with "Cycle OFF" during the reference run in automatic mode and is now at an undefined location.

Remedy

> Restart program from the beginning.

AF 7213 Number not within range

General error

> This error message appears if the tool number is greater than 64 where chain changers are involved.

Remedy

> Refer to tool data/tool change for chain changers in Section 3.4 of the Operating and Programming Instructions.

AF 7220 Transfer buffer in the NC is not free

General error

> Transfer variables are variables which communicate with the SPS, FMS and NC. If the transfer variables in the range of variables V701 to V719 are not released in the projection data but are called in the program, this error message appears. The error may be in the connection between transport system and NC.

Remedy

> Check the connection.

AF 7300 Fixed limit exceeded

General error

> Post-process measurement. A fixed tolerance limit between +1 and −1 mm is programmed in the control system symmetrically to the nominal dimension. When this fixed limit is exceeded, the measuring cycle is interrupted.

Remedy

> Re-set the measuring device or check the tool and exchange it if necessary. Check parts program or programmed measurement cycle (refer to post-process measurement in Chapter 6 of the Operating and Programming Instructions).

AF 7301 Reject: Tolerance limit C has been exceeded

General error

> A value of B ranging to max. 0.999 mm can be defined under the tolerance limit C as the deviation from the nominal dimension. If the value measured is between the tolerance limits B and C, the workpiece is classified as scrap and, depending on the programming of parameter N700, a tool exchange is initiated. The measured value is processed by the control system to calculate the correction value for this measuring point.

Remedy

No rework is possible when the tolerance limit C is exceeded, since the measured value is below the nominal dimension in the case of external machining, above it in the case of internal machining.

AF 7302 Scrap: Tolerance limit B has been exceeded

General error

A value of A ranging to max. 0.999 mm can be defined under the tolerance limit B as the deviation from the nominal dimension. If the value measured is between the tolerance limits AB and BC, the workpiece is classified as ACCEPTABLE at this measuring point. The measured value is processed by the control system for the calculation of the correction value for this measuring point.

Remedy

If tolerance limit B is exceeded, the workpiece is to be designated as scrap, since the depth of cut required would be too shallow for a rework operation.

AF 7303 Rework: Tolerance limit C has been exceeded

General error

A value of B ranging to max. 0.999 mm can be defined under the tolerance limit C as the deviation from the nominal dimension. If the value measured is between the tolerance limits B and C, the workpiece is classified as scrap and, depending on the programming of parameter N700, a tool exchange is initiated. The measured value is processed by the control system to calculate the correction value for this measuring point.

Remedy

No rework is possible when tolerance limit C is exceeded, since the measured value is above the nominal dimension in the case of external machining, below it in the case of internal machining.

AF 7304 Tool change: Workpiece is scrap

General error

The value is less than the measuring limit B and rework is not possible. Depending on the programming of parameter N700, the correction value – the corresponding measuring point M1 to M16 which is written in the variables V941 to V956 – is increased by one.

Remedy

To adhere to the required limits, use C61 to program a tool change after interrogating the variables V941 to V956 in the parts program (refer to parameters N700 in Chapter 6 of the Operating and Programming Instructions).

AF 7307 (Post-process) measurement SET not initialized

General error

The post-process measuring board was not initialized during the run-up
of the control system. The board is faulty or missing.

Remedy

Contact Gildemeister Service.

AF 7311 Interruption of communication

General error

Communication between the post-process board and the measuring device has been
interrupted, is faulty or the connection to the measuring device is faulty.

Remedy

Check the lines and also check whether the measuring device is
switched on and is functioning properly.

AF 7312 Overflow in measuring range

General error

The measured value has exceeded the measuring range of 0.999 mm which
is to be transmitted.

Remedy

Readjust the measuring device or check the tool and exchange it if appropriate (refer
to post-process measurement in Chapter 6 of the Operating and Programming
Instructions).

AF 7313 Disturbance in data traffic of SET measuring device

General error

The structure of the data telegram is incorrect. The measuring device has not been
adjusted to match; it is operating with an incorrect data protocol.

Remedy

Check the measuring device and, where applicable, parameter N699,
"changing the prefix of measured value".

AF 7314   Measuring device not initialized

General error

The post-process measurement option has been released via software, but the measuring device has not been installed or is faulty.

Remedy

Check the measuring device.

AF 7315   (Post-process) measurement SET faulty

General error

The CHECK-UP of the post-process measurement board reports a fault inside the board, i.e. a read error due to a faulty post-process board or a faulty module has developed during the CHECK-UP of the memory.

Remedy

The board or module has to be exchanged. Contact Gildemeister Service.

AF 7350   Too many measurements

General error

EMS = Electronic Measuring System (option). The EMS counter V970 has been incorrectly programmed by a parts program. Only one value is permitted between zero and nine.

Remedy

Check parts program (refer to EMS in Section 8.4 of the Operating and Programming Instructions).

SWF 7360   Unknown error of SET

Software error

EMS = Electronic Measuring System (option). Hardware fault or software error in data traffic with the SET.

AF 7361   Incorrect parity

General error

EMS = Electronic Measuring System (option). The EMS operates with a different parity than the control system expects and permits.

Remedy

Check the EMS and set an even parity.

AF 7362 Interruption in the data telegram

General error

EMS = Electronic Measuring System (option). The characters to be
transmitted have to follow each other in a set time sequence.
Disharmony in time results in the generation of this error message.

Remedy

Check that cable connecting EMS and control system is correctly seated and correct
any loose connection. Perform further checks to ensure that the device operates
properly.

AF 7363 Number of transmitted characters > 32

General error

EMS = Electronic Measuring System (option). If the string of characters (max. 31) in
the data telegram is longer, the data buffer is exceeded.

Remedy

Check that EMS functions properly. EMS faulty?

SWF 7364 CNC has not picked up measured value

Software error

EMS = Electronic Measuring System (option). When the data transmission is finished
the BFI board applies the data to the dual port interface and sets a strobe. The ZST
(central control unit) picks up the values and simultaneously cancels the strobe. If
values are to be transferred again but the strobe is still set, the ZST has either not
picked up the old values or has not cancelled the strobe. The BFI therefore generates
this error message.

AF 7365 Decimal point missing

General error

There has to be a decimal point in the data telegram to separate the
digits to be before and after the point.

Remedy

Check that EMS functions properly. EMS faulty?

AF 7366   Data telegram contains errors

General error

A character which was not expected has been detected in the data. For example, if digits are expected, it is not permissible to send letters.

Remedy

Check that EMS functions properly. EMS faulty? In addition, check that the device used is a measuring system compatible with the control system software.

AF 7367   Too many characters after the decimal point

General error

The permissible number of digits to the right of the decimal point and before actuation of the confirmation key is two or three, depending on the model. All other values are incorrect.

Remedy

Make certain that EMS functions properly. EMS faulty? In addition, make certain that the device used is a measuring system compatible with the control system software.

AF 7500   In-process measurement: Option not available

General error

A measuring circuit has been programmed although the option in-process measurement has not been released.

Remedy

If the option is not available, parameter N690 has to be at zero. If the option was purchased by the customer but mistakenly not released, notify Gildemeister Service.

AF 7501   In-process measurement: hardware faulty or not available

General error

The measuring circuit cannot be normalized when the reference point is reached. The measuring circuit is faulty or not available.

Remedy

If faulty, call Gildemeister Service.

AF 7502 In-process measurement: measuring circuit incorrect (parameter)

General error

The measuring circuit is less than zero and greater than the measuring circuit maximum.

Case 1:   Parameter N690 contains a value less than zero (external parameter input) or greater than two, however the software is only configured for two compound slides.

Case 2:   The measuring circuit in parameter N690 is 0 and therefore undefined.

Remedy

Check parameter N690 based on the above error causes and make corrections (refer to in-process measurement N690 in Chapter 6 of the Operating and Programming Instructions).

SWF 7503 In-process measurement: sensor monitor not active

Software error

The sensor monitor is not set in the projection data. Please call Gildemeister Service.

AF 7504 In-process measurement: measurement end but probe not deflected

General error

The measurement operation has been ended with G913 without retracting the probe beforehand.

Remedy

Correct parts program. Retract the probe with G0 and G1. Then program end of measuring with G913 (refer to "Measurement", Section 3.2 in the Operating and Programming Instructions).

AF 7505 In-process measurement was already activated

General error

Case 1:   The function G910, "measurement of workpiece in the parts program", has been programmed although this function has already been activated.

Case 2:   Measurement is not to be activated until after end of function G916, "traversing to fixed stop".

Case 3:   The function G911, "tool measuring in the parts program", has been programmed although this function has already been activated.

Remedy

Check and correct the parts program with regard to these criteria.

AF 7506 In-process measurement: type of measurement unknown

General error

A "detect actual value" G912 has programmed without activating a probe input (G910, G911) beforehand.

Remedy

Check and correct the parts program.

AF 7507 In-process measurement: Probe not deflected

General error

The measurement process has been interrupted by the command "measurement OFF" without the probe having been deflected.

AF 7508 In-process measurement: Probe was deflected too late

General error

The programmed path selected is too short.

Remedy

In accordance with the G-function description in Section 3.2 of the Operating and Programming Instructions, the measurement path has to be selected to ensure adequate deflection of the probe.

AF 7510 In-process measurement: Collision!

General error

The G-functions G910 and G911 (workpiece/tool measuring) activate the probe's collision monitor. The probe was deflected during preliminary positioning.

Remedy

Review and correct the parts program (positioning operation before the actual measurement).

AF 7511 Path end not at fixed stop

General error

Two conditions may have been responsible for the error:

1.   The operating mode is changed during traversing to fixed stop G916.

2.   The axis has not struck the fixed stop G916 by the time an internal period has elapsed.

Remedy

G916 has to be programmed in a separate NC block; this switches the monitor on. Program the next traverse path (this is the only one monitored) so its end position in the traversing direction is behind the stop point, therefore ensuring a movement against the fixed stop. Otherwise this error message will be generated.

AF 7612 No position regulation during synchronous speed run

General error

Error in programming. When the synchronous speed run (M801, M803, M804) is switched on, the guiding spindle is in position regulation (M19). The synchronous speed run is already switched on and afterwards the position regulation (M19) is requested by the guiding spindle.

Remedy

Request "Spindle stop" (M05) before activating the synchronous speed run. Do not request position regulation during synchronous speed or angular synchronous run.

AF 7613 No guiding/guided change during spindle synchronization

General error

Error in programming. While the spindle synchronous run (M801, M803, M804) is switched on a further synchronization function has been requested which would necessarily be followed by a change of the guiding and/or guided spindle. This is not allowed. This applies for the manual control and automatic modes.

Remedy

Delete incorrect synchronization function or, if change is necessary, switch off the synchronization (M800) beforehand.

AF 7614 No reference run performed with spindle

General error

Position regulation (M19 or M319) or angular synchronous run M801 S..., M803 S...,
M804 S...) has been requested but the spindle in question has not executed a single
rotation since the machine was switched on.

Remedy

The spindle must execute a rotation, i.e. with spindle jogging in manual control or
in automatic after cycle stop and retracted spindle release "door OPEN"; manually
if necessary.

AF 7615 Speed regulation required for guided spindle

General error

Error in programming. Synchronous run (M801, M803, M804) has been
requested although the guided spindle is being operated with speed
control, i.e. the AP286 is not exercising speed regulation.

Remedy

Select the correct synchronization function, i.e. the guided spindle
is the one subject to speed regulation from the AP186.

AF 7621 No task for guided spindle during synchronization

General error

Error in programming or operation. A task has been given to the guided spindle while
synchronization (M801, M803, M804) is switched on, e.g. M-function, G-function.
spindle keys, speed override with the handwheel.

Remedy

Do not program task for guided wheel, or select guiding spindle with
"spindle selection" during manual control.

AF 7622 No gear-shifting while synchronization is switched on

General error

Error in programming. Gear-shifting has been requested (M41, M42, M43,
M44) while spindle synchronization (M801, M803, M804) is switched on.

Remedy

Program gear-shifting only when synchronization is switched off.

AF 7623 No gear-shifting during speed regulation

General error

> Error in programming. It is not permissible to shift gears (M41, M42, M43, M44) while speed regulation (G96, G97) is active.

Remedy

> Do not request this function at this point in the parts program.

AF 7641 Error in spindle sensor circuit

General error

> The AP spindle drive reported a fault (cable break) to the sensor monitor.

Remedy

> Check spindle sensor and/or the connection between sensor and AP186 board.

AF 7643 Spindle sensor uncoupled mechanically

General error

> The control internal examination of the produced target values of the spindle and the real values of the spindle whcih are read in, show as the difference between the old and the new values for the turning moment and the speed, the difference between in- and output. This is interpreted by the control as a break of the mechanic connection between speed sensor and spindle (tachobreakage) and causes the sopt of the spindle and the axis of the corresponding spindle channel. All orders except spindle-stop are not executed and effects the renewed output of the error message.

Remedy

> Switch off control and machine. Therafter examine mechanic in the section of speed sensor/spindle and if necessary correct the error.

AF 7646 Software index exceeded; refer to NC error list

General error

> The basic software for board and the operating system are not compatible.

Remedy

> Completely exchange the software.

BF 7674   Fixed stop not reached

Operator error

When AP186 board used: The path programmed in the block after G916, "traversing to fixed stop", is too short to reach the fixed stop.

Remedy

The traverse path to be programmed has to be situated behind the fixed stop (refer to G916 in Section 3.2 of the Operating and Programming Instructions).

SWF 8010   Unknown function code for graphic representation operation

Software error

Error in the control system in the area of graphic representation.

BF 8021   Handwheel already allocated

Operator error

When the handwheel is assigned, for feed override for example, another allocation, for the magnifying lens function for example, is no longer possible.

Remedy

To activate a new handwheel allocation, switch off the existing allocation.

BF 8101   Number not in range

Operator error

During the text test the max. permissible text number in the diagnosis mode is currently 2400. If no text appears for a lower number, that number is not allocated.

BF 8102   Number not in range

Operator error

During the image test the max. permissible image number in the diagnosis mode is 100. If no screen changes occurs for a lower image number, that number is not allocated.

AF 8103   Memory error during writing

General error

In the diagnosis mode. Memory input: Delete program memory, parameter memory,

main memory or set memory. An internal error (software error) has developed during
the specified work operations or impermissible entries have been made during input.

Remedy

Notify Gildemeister Service if error message is repeated.

BF 8104 Value is larger than 255

Operator error

A value greater than 255 has been used in the diagnosis/memory input mode.

Remedy

Enter only values smaller than or equal to 255 in the memory.

BF 8105 Incorrect password

Operator error

The password entered in the diagnosis mode (access to protected
parameters) was not correct.

Remedy

Enter correct password. If the password is not known, it is not possible to access
protected parameters (refer to Section 7.1 of the operating and programming
instructions).

BF 8106 Invalid module number entered

Operator error

This error occurs in the diagnosis mode. The module number entered is
outside the permissible range.

Remedy

This range is currently max. 54.

BF 8107 Incorrect date

Operator error

This error occurs in the diagnosis mode. The date is not in the proper format.

Remedy

Enter date correctly:

| Day | from 1 to max. 31 |
| Month | from 1 to max. 12 |

       Hour   from 0 to max. 23
       Minute   from 0 to max. 59

BF 8108   Number not in range

Operator error

In the diagnosis/memory input operating mode.
A segment address small than or equal to zero has been specified.

BF 8109   New password invalid

Operator error

The verification given when entering the new password was incorrect.

Remedy

As a control, after a new password is entered it has to be entered again (refer to diagnosis/new password in Section 7.1 of the Operating and Programming Instructions).

BF 8110   Number not in range

Operator error

In the diagnosis mode. The SPS input and output numbers are only defined in the range between 1 and 16, the variable output numbers only in the ranges 1 to 299, 601 to 610 and 800 to 999.

AF 8111   Machine has to be restarted

General error

The machine has to be restarted after erasing the main memory.

BF 8112   Key forbidden here

Operator error

Modification of variables (e.g. undefined setting of variables) not allowed when diagnosis mode used as secondary operating mode.

Remedy

Select diagnosis mode directly as the main operating mode.

BF 8113 Option missing

Operator error

The elapsed-time counter option is missing. The individual machine
adjustment SPS does not support this option.

Remedy

This option was purchased by the customer and mistakenly not released.
Inform Gildemeister Service.

AF 8114   Double output configured, no input possible

General error

The I/O configuration has been defined as 1/1 = output/output. With
this configuration it is therefore impossible to select an input.

Remedy

Exit the diagnosis mode. Select the diagnosis mode again, press the softkey Input or
Output and select the desired configuration in response to the ensuing query. It is
irrelevant whether the input or output number is entered first. Positioning is carried
out automatically in accordance with the configuration.

AF 8115   Double input configured, no output possible

General error

The I/O configuration has been defined as 2/2 = input/input. With this configuration
it is therefore impossible to select an output.

Remedy

Exit the diagnosis mode. Select the diagnosis mode again, press the softkey Input or
Output and select the desired configuration in response to the ensuing query. It is
irrelevant whether the input or output number is entered first. Positioning is carried
out automatically in accordance with the configuration.

AF 8116 Configuration missing

General error

Input or Output (I/O) has been selected in the diagnosis mode without
having specified the configuration of lines 1 and 2 beforehand.

Remedy

A configuration has to be selected before selecting an I/O Input or
Output in the diagnosis mode for the first time. This is done by
pressing the softkey Input/Output. The configuration selected remains

AF 8304   Program access blocked

General error

The concerned diagnosis program was called up in the operating mode editor. Thereafter in this operating mode the diagnosis mode was called up as auxiliary operating mode and then a new access to the diagnosis program was effected. A double program access is not possible.

Remedy

Leave operating mode EDITOR and then access to the desired diagnosis progam by operating mode DIAGNOSIS.

BF 8308   Program not existing

General error

Selected diagnosis program ist not existing in the parts program memory.

Rmedy

Determine the program numbers of the available diagnosis programs in the operating mode EDITOR by means of the function "program list".
Renew program selection with correct program number input in the operating mode DIAGNOSIS.

BF 8309   Bloc not existing

General error

Each diagnosis program is to be finished by function M30 (program termination). In this program the function M30 is missing.

Remedy

Complete diagnosis program. accordingly.

AF 8330   NC Syntax

General error

The program contains a syntax error in the program bloc on which the cursor is positioned.

Remedy

Correct program corresponding to the syntax description of the diagnosis program (s. chap. 7, operating mode DIAGNOSIS).

AF 8331   Variables input blocked

General error

The data in the third bracket of a diagnosis program block define the access right to the variable resp. to the event by means of the diagnosis program; if the third bracket is missing, the content of the variables/the event is only shown and no inputs can be made. Contains the bracket a H, inputs in the operating modes DIAGNOSIS and MANUAL CONTROL/AUXILIARY MODE can be made. Inputs in the operating mode AUTOMATIC/AUXILIARY MODE DIAGNOSIS can be made when the bracket contains the character A. The third bracket must contain a Z for inputs during execution of a parts program (CYCLE ON).

Fehlerbehebung

Select the operating modes corresponding to the defined access rights and then make the desired inputs.

SF 9901   **BFC has initialized**

System error

Hardware error on the BFC board.

SF 9902   **BFC has detected processor error**

System error

Hardware error on the BFC board.

SF 9903   **BFC has detected memory error**

System error

Hardware error on the BFC board.

SF 9904   **Machine key FIFO full**

System error

Error in the hardware BFC, BFI or ZST.

SF 9905   **No test characters from BFC**

System error

Error in the hardware BFC, BFI or ZST.

SF 9911   Unexpected restart 1

            System error

            Error in the hardware BFI.

SF 9912   Unexpexted restart 2

            System error

            Error in the hardware BFI.

SF 9913   Unexpected restart 3

            System error

            Error in the hardware BFI.

SF 9914   Unexpexted restart 4

            System error

            Error in the hardware BFI.

SF 9915   Unexpected restart 5

            System error

            Error in the hardwae BFI.

SF 9916   Unexpected restart 6

            System error

            Error in the hardware BFI.

SF 9917   Unexpected restart 7

            System error

            Error in the hardware BFI.

SF 9953   Unexpected interrupt occured

            System error

            Hardware error in the central system.

SF 9956   **TRAP-Interrupt (INT 01, Single-Step)**

System error

Hardware error on the ZST.

SF 9965   **Numeric processor error   (ABBRUCH INT 16)**

System error

Hardware error on the ZST.

SF 9980   **Powerfail**

System error

Main voltage incorrect.

SF 9981   **INTER-cancel on hidden system (HH <--> NC)**

System error

Hardware/software error on ZST resp. TPS.

SF 9982   **Geoslave memory faults**

System error

Error in the hardware Geo-AP186..

SF 9983   **INTERR-cancel on   Geo-Slave**

System error

Error in the hardware Geo-AP186.

SF 9985   **Vector latch on geometry board is not free**

System error

Error in the hardware Geo-AP186.

AF 9986   **Faulty hardware configuration of geometrty board (AT)**

System error

Error in the hardware Geo-AP186.

AF 9987   **Faulty hardware configuration of geometry board (Busmode)**

System error

Error in the hardware Geo-AP186.

SF 9988   **Watchdog half time expired**

System error

Hardware/Software error on the ZST.

SF 9989   **Timeout monitoring**

System error

Hardware/Software error on the ZST.

SF 9990   **Operating interface BFI does not answer any more**

System error

Error in the hardware BFC resp. BFI.

SF 9991   **BFC was newly initialized**

System error

Error in the hardware BFC resp. BFI.

SF 9992   **BFC has detected processor error**

System error

Error in the hardware BFC resp. BFI.

SF 9993   **BFC has detected memory error**

System error

Error in the hardware BFC resp. BFI.

SF 9994   **Machine operating keys could not be performed**

System error

Error in the hardware BFC resp. BFI.

SF 9995   **BFC transmits no more test characters**

System error

Error in the hardware BFC resp. BFI.

SF 9996   **Free BFI error**

System error

Error in the hardware BFC resp. BFI.

SF 9998   **MPST-Interrupt 0FFH at unallowed moment**

System error

Hardware error in the central system.

SF 9999   **Unknown MPST-Interrupt**

System error

Hardware error in the central system.

401 F: LUBRICANT: PERMANENT PRESSURE

Cause: Maintenance is necessary:
Lubricant pressure switch S4711 signals
permanent pressure

Overcome by mechanical maintenace
personnel:
Check lubricant pressure lines and/or pressure
switch

402 F: LUBRICANT: NO PRESSURE

Cause: Maintenance is necessary:
Lubricant pressure switch S4711 signals no
pressure

Overcome by mechanical maintenance
personnel:
Check lubricant container,
check pressure switch

403 W: LUBRICANT: PERMANENT PRESSURE

Cause: Maintenance is necessary:
Lubricant pressure switch S4711 signals
permanent pressure

Overcome by mechanical maintenace
personnel:
Check lubricant pressure lines and/or pressure
switch

404 W: LUBRICANT: NO PRESSURE

Cause: Maintenance is necessary:
Lubricant pressure switch S4711 signals no
pressure

Overcome by mechanical maintenance
personnel:
Check lubricant container,
check pressure switch

405 F: FEED DRIVES: NOT OPERATIONAL

Cause: possible defect:
Feed drive servo chopper fault

Overcome by electrical maintenance
personnel:
Refer to eletric technical documentation;
chopper, circuit diagrams

406 F: PARAMETER MACHINING ON REAR SPINDLE SYNCHRONI-
SATION; INADMISSIBLE VALUE

Cause: incorrect setting parameter:
an inadmissible value is entered in the
"rear machining clamping type" parameter

Overcome by technologist, setter:
Correct parameter 104

407 F: MACHINE NOT SWITCHED ON

Cause: inadmissible condition in machine
status:
Drives not switched on on operation panel

Overcome by operator:
Press "Drive on" button,
the lamp in the button must be illuminated

408 F: MACHINE. EMERGENCY STOP

Cause: inadmissible operation in machine
status:
EMERGENCY STOP button pressed

Overcome by operator:
Release emergency stop button

409 F: NC: NOT OPERATIONAL

Cause: possible defect:
Contouring error overflow; cross slide

Overcome by electrical maintenance per-
sonnel:
check feed drives,
check cross slide encoder

410 F: TOOL HOLDER PARAMETER: INADMISSIBLE TYPE

Cause: incorrect setting, parameter:
incorrect tool holder in parameter 105

Overcome by technologist, setter:
Correct parameter 105
(refer to programming instructions)

411 F: MAIN CHUCK PARAMETER: INADMISSIBLE TYPE

Cause: incorrect setting, parameter:
incorrect main chuck type in parameter 106

Overcome by technologist, setter:
Correct parameter 106

412 F: MAIN CHUCK PARAMETER: INADMISSIBLE CHUCKING METHOD

Cause: incorrect setting parameter:
incorrect value parameter 101

Overcome by technologist, setter:
Correct parameter 101 together with
parameter 106

413 F: INPUT: M FUNCTION DUE TO INADMISSIBLE MACHINE STATUS

Cause: inadmissible operation in machine
status:
M function due to inadmissible machine status

Overcome by operator:
Acknowledge malfuncion

414 F: REAR MACHINE CLAMP PARAMETER: INADMISSIBLE VALUE

Cause: incorrect setting parameter:
an inadmissible value is entered in the
"rear machine clamping type" parameter

Overcome by technologist, setter:
Correct parameter 104

415 F: SETTING OPERATION: INADMISSIBLE

Cause: inadmissible operation in machine
status:
Protection hood is closed or MANUAL
operation mode is not selected

Overcome by operator:
open protection hood an select
MANUAL operating mode

416 F: SETTING OPERATION: SELECTION INADMISSIBLE

Cause: inadmissible operation in machine
status:
Protection hood is closed or MANUAL
operation mode is not selected

Overcome by operator:
Open protection hood and select
MANUAL operating mode

417 F: INPUNT: M FUNCTION NOT DEFINED

Cause: inadmissible operation in machine
status:
M function not defined for this machine
equipment

Overcome by operator:
acknowledge error

418 F: MAIN CHUCK PARAMETER: INADMISSIBLE WORKING
         DIRECTION

Cause: incorrect setting, parameter:
incorrect value parameter 106

Overcome by technologist, setter:
Correct parameter 106 together with
parameter 101

420 F: TOOL HOLDER: NOT LOCKED IN POSITION

Cause: inadmissible operation in machine
status:
Tool holder in incorrect position
or not locked

Overcome by operator:
Swivel tool holder with T command

421 F: TOOL HOLDER: INCORRECT PREINEXING

Cause: inadmissible operation in machine
status:
Preindexing was found to be faulty when
testing the tool holder

Overcome by operator:
Swivel tool holder with T command

424 F: TOOL HOLDER: INCORRECT PARITY CHECK

Cause: possible defect:
Tool holder position encoder has a parity fault

Overcome by operator:
Swivel tool holder (Tx) or
overcome by electrical maintenance
personnel:
position encoder, check cable

425 F: TOOL HOLDER DOES NOT LOCK

Cause: possible defect:
Initiator A4911-S8 "Check locking" does not
switch

Overcome by technologis, setter:
check initiator, if necessary, readjust and test

426 F: TOOL HOLDER: POSITION NOT FOUND

Cause: incorrect setting, parmeter:
value in parameter 105 incorrect or
Cause: possible defect:
Position generator fault

Overcome by technologist, setter:
correct parameter 105 or
Overcome by mechanical maintenance
Check position encoder

427 F: TOOL HOLDER: T VALUE COULD NOT BE TAKEN OVER

Cause: inadmissible operation in machine
status:
operating mode incorrect or
driven tools switched on

Overcome by operator:
select permitted operated mode or
if necessary, switch off driven tools

428 F: TOOL HOLDER: T VALUE TOO LARGE FOR TOOL HOLDER

Cause: inadmissible operation in machine
status:
Entered T value too large or
incorrect parameter setting:
parameter 105 too small

Overcome by operator:
enter correct t value or
overcome by technologist, setter:
correct parameter 105

429 F: TOOL HOLDER PARAMETER: INADMISSIBLE NUMBER OF
          TOOLS

Cause: incorrect setting, parameter:
Parameter 105 incorrect

Overcome by technologist, setter:
correct parameter 105

430 M: MAIN CHUCK: STROKE LIMIT CLAMPING REACHED

Cause: incorrect setting, parameter: Overcome by technologist, setter:
Main chuck has reached end position without
pressure

431 F: MAIN CHUCK: INCORRECT INPUT SIGNALS

Cause: incorrect setting parameter: Overcome by technologist, setter
incorrect signals occured when chucking   check chucking pressure and useful
or releasing   stroke (pressure monitor, proximity detectors)

432 F: MAIN CHUCK: FAULTY FRONT STROKE LIMITATION

Cause: possible defect: Overcome by electrical maintenance
Proximity detector test proved proximity   personnel:
detector S5411 to be faulty   change proximity detector or check signall
                                               path in accordance with circuit diagram

433 F: MAIN CHUCK: FAULTY REAR STROKE LIMITATION

Cause: possible defect: Overcome by electrical maintenance
Proximity detector test proved proximity   personnel:
detector S5421 to be faulty   change proximity detector or check signal path
                                               in accordance with circuit

434 F: MAIN CHUCK: FAULTY MONITORING

Cause: incorrect setting, parameter: Overcome by technologist, setter,
"Chucked" or "Released" end position left   check chucking pressure and operating
without reason   stroke

435 F: MAIN CHUCK: PRESSURE DROP

Cause: incorrect setting, parameter:
Pressure switches F5341, F5351 signal
pressure drop

Overcome by technologist, setter:
Check appropriate pressure switch in
accordance with the chucking device

436 M: MAIN CHUCK: POSITION NOT DEFINED

Cause: inadmissible operation in machine
status:
Main chuck is neither clamped nor fully open

Overcome by operator:
Give "Clamp" or "Released" command with
M command or food switch

437 F: INADMISSIBLE COMMAND

Cause: inadmissible operation in machine
status:
Chuck command inadmissible whilst spindle
turning

Overcome by operator:
Firstly stop spindle

438 M: MAIN CHUCK: DISENGAGEMENT STROKE SELECTED

Cause: inadmissible operation in machine
status:
disengagement stroke selected in parameter 101

Overcome by operator:
correct parameter 101

439 M: MAIN CHUCK: SPINDLE RUNNING TIME WITHOUT CLAMPING
            PRESSURE ECEEDED

Cause: inadmissible operation in machine
status:
Spindle runs for more than 90 seconds with
chuck open

Overcome by operator:
Acknowledge error,
if necessary, correct part program

440 F: STEADY REST: INADMISSIBLE COMMAND

Cause: inadmissible operation in machine
status:
Spindle is switched on or
inadmissible centrre sleeve position

Overcome by operator:
Stop spindle or clamp centre sleeve

441 F: STEADY REST 1: FAULTY MONITORING

Cause: incorrect setting, parameter:
Chucked condition was left or
opened status was left

Overcome by technologist, setter:
Check pressure switch F6051 or
check pressure switch F6041

442 M: STEADY REST 1: POSITION NOT DEFINED

Cause: inadmissible operation in machine
status:
Steady rest 1 is neither clamped nor fully open

Overcome by operator:
Give "chucking" or "release" command with
M command or food switch

443 F: STEADY REST 2: FAULTY MONITORING

Cause: incorrect setting, parameter:
Chucked condition was left or
opened status was left

Overcome by technologist, setter:
Check pressure switch F6351 or
check pressure switch F6341

444 M: STEADY REST 2: POSITION NOT DEFINED

Cause: inadmissible operation in machine
status:
Steady rest 2 is neither clamped nor fully open

Overcome by operator:
Give "chucking" or "release" command with
M command or foot switch

449 F: NUMBER OF STEADY RESTS PARAMETER: VALUE TOO LARGE

Cause: incorrect setting, parameter:
Value in parameter 108 is too large

Overcome by technologist, setter:
correct parameter 108 in accordance with the
machine equipment

451 F: CENTRE SLEEVE PARAMETER: INADMISSIBLE TYPE

Cause: incorrect setting, parameter:
Incorrect centre sleeve type in parameter 108

Overcome by technologist, setter:
Correct parameter 108

452 M: CENTRE SLEEVE: FRONT END POSITION REACHED

Cause: incorrect setting, parameter:
Front end position reached when chucking
without pressure

Overcome by technologist, setter:
Centre sleeve stroke not sufficient,
Traverse tailstock

453 F: CENTRE SLEEVE: FAULTY INPUT SIGNALS

Cause: incorrect setting, parameter:
Incorrect signals encountered when traversing
the centre sleeve

Ovecome by technologist, setter:
Check centre sleeve setting (pressure switch
proximity detectors)

454 F: CENTRE SLEEVE: FAULTY MONITORING

Cause: incorrect setting, parameter:
Clamped or retracted condition left without
reason

Overcome by technologist, setter:
Check centre sleeve setting (pressure switch,
proximity detectors)

455 F: CENTRE SLEEVE: FAULTY MONITORING

Cause: incorrect setting, parameter:
Centre slleve clamped and pressure drop

Overcome by technologist, setter:
Check pressure setting switch F5751

456 M: CENTRE SLEEVE: POSITION NOT FOUND

Cause: inadmissible operation in machine
status:
Centre sleeve is neither clamped nor fully
opened

Overcome by operator:
Command "Forward" or "Return" with M
command or foot switch

457 F: SLEEVE COMMAND INADMISSIBLE

Cause: inadmissible operation in machine
status:
Spindle is switched on or the steady rest not
clamped

Overcome by operator:
Stop spindle or
clamp steady rest

460 F: DRIVEN TOOLS: FAULTY COUPLING CYCLE

Cause: incorrect setting, parameter:
faulty coupling cycle at M53/M54

Overcome by technologist, setter:
Proximity detectors engaged/disengaged,
check A4911-S1011

461 F: DRIVEN TOOLS: COUPLING MONITORING FAULTY

Cause: incorrect setting, parameter:
The selected status has been left without
reason

Overcome by technologist, setter:
Proximity detectors engaged/disengaged
check A4911-S1011

462 F: DRIVEN TOOLS: INADMISSIBLE COMMAND

Cause: inadmissible operation in machine
status:
Tool holder not in position for driven tools

Overcome by operator:
Swivel tool holder into a permissible
position

463 F: MAIN SPINDLE CLAMPING: FAULTY EXECUTION

Cause: incorrect setting, parameter:
proximity detector of the spindle clamping
have incorrect signal

Overcome by technologist, setter:
Check proximity detector S7211-S7241

464 F: MAIN SPINDLE CLAMPING: FAULTY MONITORING

Cause: incorrect setting, parameter:
The selected status was left without reason

Overcome by technologist, setter:
Check proximity detectos S7211-S7241

465 F: MAIN SPINDLE CLAMPING: INADMISSIBLE COMMAND

Cause: inadmissible operation in machine
status:
main spindle clamping is not in position
window

Overcome by operator:
Execute spindle positioning M19 in advance

466 F: PICK-OFF DEVICE: INADMISSIBLE COMMAND

Currently not occupied

Currently not occupied

467 F: SWIVEL PICK-OFF DEVICE: FAULTY MONITORING

Cause: incorrect setting, parameter:
The selected status was left without reason

Overcome by technologist, setter:
Check proximity detectors S8041 and S8051

468 M: SWIVEL PICK-OFF DEVICE: POSITION NOT DEFINED

Cause: Inadmissible operation in machine
status:
Swivel arm is in neither of the two end
positions

Overcome by operator:
Command "Forward" or "Return" with M
command or key

469 F: PICK-OFF DEVICE GRIPPING: FAULTY MONITORING

Cause: incorrect setting, parameter:
The selected status was left without reason

Overcome by technologist, setter:
Check proximity detectors S8241 and S8251

470 M: PICK-OFF DEVICE GRIPPING: POSITION NOT DEFINED

Cause: inadmissible operation in machine
status:
Swivel arm is in neither of the end positions

Overcome by operator:
Command "Grip" or "Release" with M
command or key

471 F: PICK-OFF DEVICE FLAP: FAULTY MONITORING

Cause: incorrect setting, parameter:
The selected status was left without reason

Overcome by technologist, setter:
Check proximity detectors S8441 and S8451

472 M: PICK-OFF DEVICE FLAP: POSITION NOT FOUND

Cause: inadmissible operation in machine:
status:
Swivel arm is in neither of the end positions

Overcome by operator:
Command "Open" or "Clos" with M command
or key

473 F: PICK-OFF DEVICE PARAMETER: INADMISSIBLE VALUE

Cause: incorrect setting, parameter:
Parameter 107 incorrect

Overcome by technologist, setter:
Correct parameter 107

474 F: INPUT MODULES: CONTROL VOLTAGE FAILURE

Cause: possible defect:
Automatic fuse F551 triggered

Overcome by electrical maintenance
personnel:
Overcome cause of overloading

475 F: OUTPUT MODULES: CONTROL VOLTAGE FAILURE

Cause: possible defect:
Automatic fuse F711 triggered

Overcome by electrical maintenance
personnel:
Overcome cause of overloading

476 F: VALVE; CONTROL VOLTAGE FAILURE

Cause: possible defect:
Automatic fuse F731 triggered

Overcome by electrical maintenance
personnel:
Overcome cause of overloading

477 F: MACHINE WARNING: REFER TO FAULT LIST

Cause: possible defect: Overcome by electrical maintenance
Only with certain options,   personnel:
refer to circuit diagram page 20   Overcome fault

478 W: MACHINE WARNING: REFER TO FAULT LIST

Cause: possible defect: Overcome by electrical maintenance
Only with certain options,   personnel:
refer to circuit diagram page 20   Overcome fault

479 F: FEED DRIVES: MOTOR OVER-TEMPERATURE

Cause: possible defect or incorrect setting,   Overcome by electrical maintenance
parameter: personnel:
One of the feed drives signals overtemperature   allow motor to cool down

480 F: FEED DRIVES: MOTOR OVER-TEMPERATURE

Cause: possible defect: Overcome by electrical maintenance
Motor protection switch Q3921 triggered   personnel:
                                                 Check current setting Q3921

482 F: LUBRICATION: MOTOR OVERCURRENT

Cause: possible defect: Overcome by electrical maintenance
Motor protection switch Q4711 triggered   personnel:
                                                 Check current setting Q4711

485 F: CHIP CONVEYOR: MOTOR OVER CURRENT

Cause: inadmissible operation in machine   Overcome by operator:
status: Run chip conveyor empty or
Motor protection switch Q5211 triggered   Overcome by electrical maintenance
                                                 personnel:
                                                 check current setting Q5211

487 F: ROTATIONAL SPEED MONITORING: UNIT INTERRUPTED OR TRIGGERED

Cause: inadmissible operation in machine
status:
Spindle turns at more than 50 rpm in setting
operation or
Cause: possible defect:
fault A2611

Overcome by operator:
Reduce totational speed to less than 50 rpm or
Overcome by electrical maintenance
personnel:
Check unit A2611

488 W: LUBRICATION: MOTOR OVERCURRENT

Cause: possible defect:
Motor protection switch Q4711 triggered

Overcome by eletrical maintenance
personnel:
Check current setting Q4711

490 F: MACHINE FAULT: SEE FAULT LIST

Cause: possible defect:
Only with certain opetions, refer to circuit
diagram page 20

Overcome by electrical maintenance
personnel:
overcome fault

493 F: CHIP CONVEYOR: MOTOR OVERCURRENT

Cause: inadmissible operation in machine
status:
Motor protection switch Q5211 triggerd

Overcome by operator:
Run chip conveyor empty or
Ovecome by electrical maintenance
personnel:
Check current setting Q5211

495 F: MAIN SPINDLE DRIVE: MOTOR OVERTEMPERATURE

Cause: inadmissible operation in machine
status:
Motor overloaded

Overcome by electrical maintenance
personnel:
Check chopper and allow motor to cool down

496 W: MAIN SPINDLE DRIVE: MOTOR OVERTEMPERATURE

Cause: inadmissible operation in machine
status:
Motor overloaded

Overcome by electrical maintenance
personnel:
Check chopper and allow motor to cool down

497 F: MAIN SPINDLE DRIVE: NOT OPERATIONAL

Cause: possible defect:
Main spindle drive chopper damaged

Overcome by electrical maintenance
Overcome fault, refer to technical
documentation

498 F: MAIN SPINDLE DRIVE: AUXILARY VENTILATOR FAULT

Cause: possible defect:
Motor protection switche Q3121 triggered

Overcome by electrical maintenance
personnel:
Check current setting Q3121

499 W: MAIN SPINDLE DRIVE: AUXILARY VENTILATOR FAULT

Cause: possible defect:
Motor protection switche Q3121 triggered

Overcome by electrical maintenance
personnel:
Check current setting Q3121

500 F: MAIN CHUCK: MAX. ROTATIONAL SPEED EXCEEDED

Cause: incorrectly set parameters:
Rotational speed was higher than the set
chuck speed

Overcome by technologist, setter:
Check set rotational speed, if necessary
correct part program

501 M: MAIN CHUCK: IN CHANGE POSITION

Cuase: inadmissible operation in machine
status:
Chuck key is not inserted in operator panel

Overcome by operator:
Insert chuck key in release switch on
operator panel

502 M: CROSS SLIDE: SLIDE IS AT REFERENCE POINT

Cause: inadmissible operation in machine
status:
X and/or Z axis is on reference dog

Overcome by operator:
Move slide left down in MANUAL operating
mode

503 F: OPERATING MODES: MOVE FREE SELECTED

Cause: inadmissible operation in machine
status:
Move free selected, without the slide being
on the end limit

Overcome by operator:
Deselected operating mode, 'move free' with
the key switch

504 F: PARAMETER MATERIAL SUPLLY TYPE: VALUE INADMISSIBLE

Cause: incorrect setting, parameter:
value in parameter 107 incorrect

Overcome by technologist, setting
Correct parameter 107

505 F: BAR OPERATION: INADMISSIBLE COMMAND

Cause: inadmissible operation in machine
status:
Command M94 triggered, without bar
operation running

Overcome by operator:
Switch on and select bar operation

506 F: BAR OPERATION: NOT OPERATIONAL

Cause: inadmissible operation in machine
status:
Bar operation selected, but not operational

Overcome by electrical maintenance
personnel:
Check bar unit

507 M: BAR OPERATION: CHANGE BAR

Cause: inadmissible operation in machine
status:
Bar is fully machined

Overcome by operator:
Insert new bar, as no bar feeding magazin

510 F: TOOL HOLDER: MOTOR OVERCURRENT

Cause: possible defect:
Motor protection switch Q4911 triggered

Overcome by electrical maintenance
personnel:
Check current setting Q4911

511 W: TOOL HOLDER: MOTOR OVERCURRENT

Cause: possible defect:
Motor protection switch Q4911 triggered

Overcome by electrical maintenance
personnel:
Check current setting Q4911

512 F: TOOL HOLDER: MOTOR OVERTEMPERATURE

Cause: inadmissble operation in machine
status:
Motor overtemperature monitoring F4951
triggered

Overcome by eletrical maintenance
personnel:
motor must cool down

513 W: TOOL HOLDER: MOTOR OVERTEMPERATURE

Cause: inadmissble operation in machine
status:
Motor overtemperature monitoring F4951
triggered

Overcome by eletrical maintenance
personnel:
motor must cool down

514 F: HYDRAULIC: FILTER BLOCKED

Cause: maintenance is necessary:
Differential pressure prior to/after the
hydraulic filter is too large

Overcome by mechanical maintenance
personnel:
Change hydraulic filter

515 W: HYDRAULIC: FILTER BLOCKED

Cause: maintenance is necessary:
Differential pressure prior to/after the
hydraulic filter is too large

Overcome by mechanical maintenance
personnel:
Change hydraulic filter

516 F: SWITCH CABINET COLLING: OVERCURRENT

Cause: possible defect:
Motor protection switch Q321 triggered

Overcome by electrical maintenance
personnel:
Check current setting Q321

517 W: SWITCH CABINET COLLING: OVERCURRENT

Cause: possible defect:
Motor protection switch Q321 triggered

Overcome by electrical maintenance
personnel:
Check current setting Q321

518 F: SWITCH CABINET COOLING: FILTER BLOCKED

Cause: Maintenance is necessary:
Differential pressure prior to/after the filter
mat is too large

Overcome by mechanical maintenance
personnel:
Clean or replace filter

519 W: SWITCH CABINET COOLING: FILTER BLOCKED

Cause: Maintenance is necessary:
Differential pressure prior to/after the filter
mat is too large

Overcome by mechanical maintenance
personnel:
Clean or replace filter

520 F: SWITCH CABINET COOLING: OVERTEMPERATURE

Cause: possible defect:
Overtemperature in switch cabinet

Overcome by mechanical maintenance
personnel:
Check cooling unit,
Ambient temperature too high ?

521 F: COOLANT: MOTOR OVERTEMPERATURE

Cause: possible defect:
Motor protection switch Q4811 or Q4831
has been triggered

Overcome by electrical maintenance
personnel:
Check current setting Q4811 and Q4831

522 F: HYDRAULIC GROUP: MOTOR OVERCURRENT

Cause: possible defect:
Motor protection switche Q4611 triggered

Overcome by electrical maintenence
personnel:
check current setting Q4611

525 F: MAIN SPINDLE C AXIS: FAULTY MONITORING

Cause: incorrectly set parameter:
Pressure and limit switch incorrectly set
Cause: possible defect:
C axis incorrectly set or defective

Overcome by electrical maintenance
personnel:
Check pressure and limit switches, if
necessary replace

530 M: PROTECTION HOOD: NOT OPENED AND NOT CLOSED

Cause: inadmissible position in machine
status:
Protection hood not locked and not open

Overcome by operator:
Open and than close protection hood

531 F: PROTECTION HOOD: FAULTY IN SAFETY CIRCUIT

Cause: possible defect:
Saftey circuir of the protection hood has an
incorrect signal sequence

Overcome by electrical maintenance
personnel:
see circuit diagram page 27

532 F: PROTECTION HOOD: UNAUTHORIZED OPENING

Cause: inadmissible operation in machine
status:
Protection hood was opened with the
emergency release

Overcome by operator:
Open and than close protection hood

533 M: PROTECTION HOOD: OPENING NOT PERMITTED AS BUTTON IS PRESSED

Cause: inadmissible operation in machine status:
A button is pressed when opening the protection hood

Overcome by operator:
Release all buttons and open protection hood

534 M: PROTECTION HOOD: SAFETY RAIL HAS RESPONDED

Cause: inadmissible operation in machine status:
Closing of automatic protection hood interrupted

Overcome by operator:
Acknowledge malfunction, if necessary check saftey rail

535 M: PROTECTION HOOD: MOTOR OVERCURRENT

Cause: incorrectly set parameter, possible defect:
motor protection switch Q10211 trigerred

Overcome by electrical maintenance personnel:
Check Q10211 current setting

540 F: TAILSTOCK TYPE PARAMETER: INVALID VALUE

Cause: incorrectly set parameter:
Incorrect value in parameter

Overcome by technologist, setter:
Correct parameter 108

541 M: TAILSTOCK: POSITION UNDEFINED

Cause: inadmissible operation in machine status:
The traversable tailstock is not in one of the end positions

Overcome by operator:
Move using the M command or the "FORWARD" or REVERSE" button

542 F: TAILSTOCK: FAULTY MONITORING

Cause: possible defect:
Limit switch of traversable tailstock

Overcome by electrical maintenance
personnel:
Check and, if necessary, replace limit switch

543 F: TAILSTOCK: INADMISSIBLE COMMAND

Cause: inadmissible operation in machine
status:
centre sleeve is extended or main spindle
turning

Overcome by operator:
acknowledge fault, if necessary part
program

550 F: PARAMETER HANDLING TYPE: INADMISSBLE VALUE

Cause: incorrect parameter setting,
possible defect:
value in parameter 104 incorrect

Overcome by technologist, setter:
correct parameter 104

551 F: PORTAL: INADMISSBLE COMMAND

Cause: inadmissble operation in machine
status:
The portal condition is incorrect

Overcome by technologist, setter
acknowledge error, switch portal to the
correct condition

552 F: PORTAL: NOT OPERATIONAL

Cause: possible defect:
portal not switched on or not operational

Overcome by operator:
check portal

553 F: LOADING FLAP: COMMAND INADMISSBLE

Cause: possible defect:
Protection hood is open or gripper is in
working space

Overcome by operator:
Close protection hood and take gripper
out of the working space

554 F: LOADING FLAP: FAULTY MONITORING

Cause: inadmissible operation in machine
status:
The selected status is left without reason

Overcome by technologist, setter:
Check the loading flap limit switch

555 M: LOADING FLAP: POSITION UNDEFINED

Cause: inadmissible operation in machine
status:
Loading flap is neither closed or open

Overcome by operator:
trigger "open " or "close" command

560 F: GEARING TYPE MAIN DRIVE PARAMETER: INADMISSBLE VALUE

Cause: incorrectly set parameters:
Parameter 104 incorrect

Overcome by technologist, setter:
Correct parameter 104

561 M: MAIN DRIVE GEARING: POSITION NOT DEFINED

Cause: inadmissble operation in machine
status:
The gearing is not in any of the admissble
positions

Overcome by operator:
Trigger a command for an inadmissible
position

562 F: MAIN DRIVE GEARING: FAULTY COUPLING CYCLE

Cause: possible defect:
The required position is not reached

Overcome by operator, setter:
Repeat command, check gearing control

563 F: MAIN DRIVE GEARING: FAULTY MONITORING

Cause: possible defect:
The required position is not reached

Overcome by operator, setter:
Repeat command, check gearing control

564 F: MAIN DRIVE GEARING: INADMISSIBLE COMMAND:

Cause: incorrect operation in machine status
The command is an inadmissble in this machine
status

Overcome by operator, setter:
Change machine condition, repeat command

653 F: PARAMETER: TAKE OVER ONLY WHEN ACCELERATING THE MACHINE

Cause: incorrectly set parameter:
If parameters 106, 107, 108 are changed,
switch off machine

Overcome by technologist, setter
Change parameters, switch off machine and
switch on again

654 F: HYDRAULIC GROUP: NO SYSTEM PRESSURE

Cause: maintenance necessary:
Hydraulic pressure loss or pressure drop

Overcome by mechanical maintenance
personnel:
Check oil level, hydraulic leaks