B&R Revision Information

ARNC0

23-May-2019
Contents

B&R Revision Information (23.05.2019) ARNC0

Requests and problems by product/component ........................................................................1
TA4000.02 (TL Motion Components) ......................................................................................1
NC Software - ARNC0 Important Informations .................................................................1
NC Software - ARNC0 V5.06.1 ............................................................................................2
NC Software - ARNC0 V5.06.0 ............................................................................................2
NC Software - ARNC0 V5.05.5 ............................................................................................2
NC Software - ARNC0 V5.04.1 ............................................................................................2
NC Software - ARNC0 V5.04.0 ............................................................................................2
NC Software - ARNC0 V5.03.1 ............................................................................................3
NC Software - ARNC0 V5.03.0 ............................................................................................3
NC Software - ARNC0 V5.02.1 ............................................................................................4
NC Software - ARNC0 V5.02.0 ............................................................................................4
NC Software - ARNC0 V5.01.1 ............................................................................................4
NC Software - ARNC0 V5.01.0 ............................................................................................5
NC Software - ARNC0 V4.00.0 ............................................................................................5
NC Software - ARNC0 V3.18.0 ............................................................................................6
NC Software - ARNC0 V3.18.1 ............................................................................................6
NC Software - ARNC0 V3.17.0 ............................................................................................7
NC Software - ARNC0 V3.16.2 ............................................................................................8
NC Software - ARNC0 V3.16.1 ............................................................................................8
NC Software - ARNC0 V3.16.0 ............................................................................................9
NC Software - ARNC0 V3.15.1 ...........................................................................................9
NC Software - ARNC0 V3.15.0 ..........................................................................................10
NC Software - ARNC0 V3.14.6 .........................................................................................10
NC Software - ARNC0 V3.14.5 .........................................................................................10
NC Software - ARNC0 V3.14.4 .........................................................................................10
NC Software - ARNC0 V3.14.3 .........................................................................................10
NC Software - ARNC0 V3.14.2 .........................................................................................11
NC Software - ARNC0 V3.14.1 .........................................................................................11
NC Software - ARNC0 V3.14.0 .........................................................................................11
NC Software - ARNC0 V3.13.0 .........................................................................................13
NC Software - ARNC0 V3.12.2 .........................................................................................13
NC Software - ARNC0 V3.12.1 .........................................................................................14
NC Software - ARNC0 V3.12.0 .........................................................................................14
NC Software - ARNC0 V3.11.1 .........................................................................................14
NC Software - ARNC0 V3.11.0 .........................................................................................15
NC Software - ARNC0 V3.10.2 .........................................................................................15
NC Software - ARNC0 V3.10.1 .........................................................................................15
NC Software - ARNC0 V3.10.0 .........................................................................................16
NC Software - ARNC0 V2.52.2 .........................................................................................17
NC Software - ARNC0 V2.52.1 .........................................................................................17
NC Software - ARNC0 V2.52.0 .........................................................................................18
NC Software - ARNC0 V2.49.4 .........................................................................................18
NC Software - ARNC0 V2.49.3 .........................................................................................18
NC Software - ARNC0 V2.49.2 .........................................................................................19
NC Software - ARNC0 V2.49.1 .........................................................................................19
NC Software - ARNC0 V2.49.0 .........................................................................................19
NC Software - ARNC0 V2.48.2 .........................................................................................20
NC Software - ARNC0 V2.48.1 .........................................................................................20
NC Software - ARNC0 V2.48.0 .........................................................................................20
NC Software - ARNC0 V2.47.4 .........................................................................................21
NC Software - ARNC0 V2.47.3 .........................................................................................21
NC Software - ARNC0 V2.47.2 .........................................................................................21
NC Software - ARNC0 V2.47.1 .........................................................................................27
NC Software - ARNC0 V2.47.0 .........................................................................................27
NC Software - ARNC0 V2.46.2 .........................................................................................28
NC Software - ARNC0 V2.46.1 .........................................................................................28
NC Software - ARNC0 V1.46.0 .........................................................................................29
NC Software - ARNC0 V1.45.3 .........................................................................................30
NC Software - ARNC0 V1.45.2 .........................................................................................30
NC Software - ARNC0 V1.45.1 .........................................................................................31
NC Software - ARNC0 V1.45.0 .........................................................................................31
NC Software - ARNC0 V1.44.5 .........................................................................................32
NC Software - ARNC0 V1.44.4 .........................................................................................32
NC Software - ARNC0 V1.44.3 .........................................................................................33
NC Software - ARNC0 V1.43.2 .........................................................................................33
NC Software - ARNC0 V1.43.1 .........................................................................................33
NC Software - ARNC0 V1.43.0 .........................................................................................34
NC Software - ARNC0 V1.42.1 .........................................................................................36
NC Software - ARNC0 V1.42.0 .........................................................................................36
NC Software - ARNC0 V1.41.2 .........................................................................................38
NC Software - ARNC0 V1.41.1 .........................................................................................38
NC Software - ARNC0 V1.41.0 .........................................................................................38
NC Software - ARNC0 V1.40.1 .........................................................................................39
NC Software - ARNC0 V1.40.0 .........................................................................................40
Contents
<table>
<thead>
<tr>
<th>B&amp;R Revision Information (23.05.2019) ARNC0</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>NC Software - ARNC0 V0.31.2</td>
<td>108</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.30.0</td>
<td>108</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.29.2</td>
<td>108</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.29.1</td>
<td>109</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.29.0</td>
<td>109</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.28.0</td>
<td>109</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.27.3</td>
<td>110</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.27.2</td>
<td>110</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.27.1</td>
<td>110</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.27.0</td>
<td>111</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.26.0</td>
<td>111</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.25.0</td>
<td>111</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.24.3</td>
<td>112</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.24.2</td>
<td>112</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.24.1</td>
<td>112</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.24.0</td>
<td>113</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.23.5</td>
<td>114</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.23.4</td>
<td>114</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.23.3</td>
<td>114</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.23.2</td>
<td>115</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.23.1</td>
<td>115</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.3</td>
<td>115</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.2</td>
<td>115</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.1</td>
<td>115</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.0</td>
<td>116</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.5</td>
<td>116</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.4</td>
<td>116</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.3</td>
<td>117</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.2</td>
<td>117</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.1</td>
<td>117</td>
</tr>
<tr>
<td>NC Software - ARNC0 V0.22.0</td>
<td>117</td>
</tr>
</tbody>
</table>
Requests and problems by product/component

1A4000.02 (1.1 Motion Components)

NC Software - ARNC0 Important Informations

ID#551295 : Important Information

ARNC0 versions V5.00.0 or higher can be used for SG4 ARM target systems

The following libraries are available for ARM target systems:
- Arnc0man
- Arnc0ext
- Arnc0mx

Note:
For ARM target systems with a Cortex A8 core (e.g. C30), these libraries can not be used.

ID#511345 : Important Information

AS Upgrade for Arnc0 Libraries

From ARNC0 version V3.16.0, the following AS Library upgrades are no longer generated:
- AS3.0_MC_Arnc0_V3.xx.x_EXT.exe
- AS4_MC_Arnc0_V3.xx.x_EXT.exe

They are replaced by:
- AS3.0_MC_Arnc0ext_V3.xx.x.exe
- AS4_MC_Arnc0ext_V3.xx.x.exe

The new upgrades contain only the Arnc0ext Library.

ID#457530 : Important Information

ARNC0 versions V2.51.3 up to V2.99.9

POWERLINK or SDC interfaces defined in the ARNC0 configuration are no longer supported starting with ARNC0 V3.00.0.

ARNC0 V2.51.3 to V2.99.9 contains expansions / error corrections based on ARNC0 V2.51.2. Expansions / Error corrections listed for ARNC0 V2.51.3 to V2.99.9 do not apply to V3.00.0 or later.

ID#451410 : Important Information

Interfaces not longer supported by ARNC0

With ARNC0 V3.10.0 or higher, ACP10 and SDC axes can be used only with PLCopen (set "Use PLCopen" in the Wizard to insert axes).

If POWERLINK or SDC interface are defined in the ARNC0 configuration, the ARNC0 startup will be aborted.

ARNC0 supports only virtual interfaces (PV name="VIRTUAL" Mode="0").

From ARNC0 V1.37.0, drives with CAN interface will not be supported anymore.

ID#450105 : Important Information

Minimum Requirements for ARNC0

From ARNC0 V3.19.0:
- Automation Studio 3.0.90.32 up to 3.1.0.0 or
- Automation Studio 4.1.7.61 or higher

From ARNC0 V3.10.0:
- Automation Studio 3.0.90.32 up to 3.1.0.0 or
- Automation Runtime V3.00 or higher
- Automation Runtime B3.01 or higher if ACP10 axes via PLCopen IF are used

From ARNC0 V1.20.0:
- Automation Studio 3.0.80.25 or higher
- Automation Runtime V3.00 or higher

From ARNC0 V1.00.0:
- Automation Studio 3.0.80 or higher
- Automation Runtime 02.95 or higher

From Automation Runtime E3.00:
- ARNC0 V1.25.0 or higher

ID#407820 : Important Information

Included drive operating systems
The ARNC0 requires the Library ACP10man. Starting with ARNC0 V2.45.0 versions of both libraries, except for the last digit of the version number, must be identical.
The drive operating system with the version of Library ACP10man is used.

ID#396830 : Important Information
From ARNC0 V2.45.0 and Automation Runtime V4.08 a valid license (Technology Guarding) is required.

ID#257892 : Important Information
Stack Requirements of the NC Manager Task Class
Starting with Automation Runtime B3.01 and ARNC0 V1.05.2, ARNC0 uses one of the cyclic task classes for calculating the setpoint (TC#1 or the respective NC Manager task class with V1.25.1 and later). Since the stack of respective task class is being used, it must be increased in some circumstances (particularly when using coordinate system transformations).

NC Software - ARNC0 V5.06.1
ID#400280914 : solved problem, solved since V5.06.1
Commands in NC Test or NC Trace permanently blocked
In rare cases, the execution of commands in the NC Test or NC Trace is permanently blocked. If this problem occurs, Automation Studio displays the following message:
Message from '<NcObject>':
The status of the NC object '<NcObject>' is '0'
Would you like to cancel the operation?

NC Software - ARNC0 V5.06.0
ID#400219911 : solved problem, solved since V5.06.0
Backward move on the path when G17Q/G18Q/G19Q or G217/G218/G219 is active
If Wrapping (G17Q/G18Q/G19Q) or Axis Mapping (G217/G218/G219) is active, then driving back on the path is allowed again.
Note: Not allowed since Arnc0 V5.00.1. In this case the program was aborted with error 7252.

ID#519280 : solved problem, solved since V5.06.0
Acceleration too low with G114
In circular blocks, G114 may cause the acceleration values to be too small.

NC Software - ARNC0 V5.05.0
ID#618925 : solved problem, solved since V5.05.0
Program abort immediately after program start, abort is not completed.
If a CNC program is aborted immediately after starting it (some CNC cycles), the program may not be aborted and a deadlock may occur.

NC Software - ARNC0 V5.04.1
ID# 400255357, 400259495 : solved problem, solved since V5.04.1
Wrong positions on axes with a PLCopen period unequal to zero
If an axis in the CNC system uses a PLCopen period other than zero, the axis position may be permanently offset and an unexpected movement may be made over several periods at the beginning of the program.

ID#400244923 : solved problem, solved since V5.04.1
Jump of TCP orientation
A stop on path (e.g. due to dwell time G04) in a non-optimized section with active G818 could lead to a jump in TCP orientation (not on joint axes) for robots.

ID# 400124830, 400133526 : solved problem, solved since V5.04.1
Not all parameters applied when initializing axes
If the axes used in the CNC system are not defined in the init parameter module, but only at runtime (ARNC0CNC_type), the following parameters are not applied during initialization (ncAXES, ncINIT):
- cnc_obj.axis.axis[i].ipl_mode
- cnc_obj.axis.axis[i].drive_axfilter
- cnc_obj.axis.axis[i].add_par_id

NC Software - ARNC0 V5.04.0
ID#400257069 : solved problem, solved since V5.04.0
Automatic tangential axis (G141) in combination with rounding edges (G126)
If an automatic tangential axis (G141) is used together with rounding edges (G126), a programmed angular offset is not taken in consideration and a position jump on the tangential axis may occur.
ID#596335 : solved problem, solved since V5.03.1
Restart status information is not updated
The restart command does not delete the status flag "cnc_obj->restart.status.data_saved".

ID#400250642 : new function since V5.03.1
New system variable $G102_MODE
System variable $G102_MODE can be used to configure the behavior of G102.
The circle is not clearly defined if the starting point, endpoint and programmed circle point lie on a straight line or if the programmed circle point coincides with the starting point or endpoint of the circle.

$G102_MODE not programmed or $G102_MODE = 0:
In this case, the CNC program is aborted with error 10461: "Points of circle lie on a straight line".

$G102_MODE = 1:
In this case, the affected circular block is converted into a linear block and warning 10536: "G102, points of circle on a straight line, converted into a linear block" is output.

ID#400250642 : solved problem, solved since V5.03.1
CNC program with G102 aborted
A CNC program with G102 is aborted with error 10461: "Circle points in straight line" although starting point, endpoint and programmed circle point do not lie on a line.

ID#400214030 : solved problem, solved since V5.03.1
Single-step mode
In single-step mode, certain commands ncAction(ncMOVE, ncCONTINUE) must be issued multiple times in order to continue the CNC program.
Reason: In Arnc0, some commands are divided into multiple steps (e.g. G102 or MoveC), and in single-step mode execution is suspended after each one.
Now, with the ncSBL_CONTROL mode, only one command is required for continuing such blocks.

NC Software - ARNC0 V5.03.0
ID#586955 : solved problem, solved since V5.03.0
Contour violation with active CDC
If a contour violation occurs in a path section immediately before CDC deselection due to a very large tool radius, this is not detected. ARNC0 always behaves as if G239 is active. In other words, regardless of G39 or G239, a contour violation is always permitted, the program is not aborted or does not attempt to minimize the contour violation.

ID#586255 : solved problem, solved since V5.03.0
Warning 8154 after blocks with G05 or G06
If a line with a non-moving block (e.g. G04, M function) follows immediately after a block with G05 or G06, warning 8154: "NC block with move distance equal to 0.0" is output.

ID#580045 : solved problem, solved since V5.03.0
Page fault in ARNC0
In very rare cases, a page fault may occur when executing a command or CNC program.

ID#400223311 : solved problem, solved since V5.03.0
G05/G06 with coordinate system rotation (G92, G192, G292, etc.)
With G05 or G06, error 10452: "Polar coordinate machine - programmed rotation/shift plane not allowed" occurs if the coordinate system has been rotated and then rotated back again.
Note:
G05 and G06 are only permitted if the coordinate system is not rotated (10385: "Polaraxis must not be programmed (Rotation is active)").

ID#509160 : solved problem, solved since V5.03.0
Deadlock when restarting CNC program
A deadlock occurs when a CNC program is restarted and an axis error is active at the same time.
Single step mode ncLINENUMBER in combination with ncFILE_XL

In mode ncFILE_XL the single step mode ncLINENUMBER doesn't stop at lines with not executable code at the beginning of a program. Only the last line before a line with an executable code is considered for stepping, all other lines at the beginning are skipped.

Page fault when starting program in mode ncFILE_XL

G-Code program that contains no executable code at the first line can cause a page fault in mode ncFILE_XL.

Wrong variable initialization

An explicitly initialized variable could be still zero. (The problem occurred especially on ARM targets.)

Deadlock in ncFILE_XL / ncDNC mode

Missing empty line at end of program causes a deadlock (deadlock occurs at the end of the program).

Incorrect axis positions for axes of type ncROTARY

For axes of type ncROTARY, it can occur that the target position is offset by one period.

Incorrect axis positions in CNC monitor

It can occur that the axis positions are updated too late in the CNC monitor.

For axes of type ncROTARY or ncTANGENT, it can occur that a position offset by one period may be displayed.

Note:

Only the display in the CNC monitor is affected, not the actual axis positions!

Page fault in ncDNC

Page fault occurred when the DNC buffer was empty (since ARNC0 V1.45.1).

Error 40144 at program abort

If a CNC program is aborted immediately after a G201 (within the residual path), error "40144: Internal Error - Job ID already acknowledged" may arise.

Syntax error on position latch after trigger (G200/G201)

On a position latch after trigger (G200/G201), only the trigger position of the first ten axes (e.g. L0.S0 to L0.S9) can be used. A syntax error occurs when accessing the trigger position of the eleventh to fifteenth axis (e.g. L0.S10).

Improved behaviour of Tracking (synchronization and desynchronization phase)

The behaviour in the synchronization and desynchronization phase has been improved. In these phases the entire length of the movement is used for the synchronization and desynchronization on a conveyor belt.

When starting a CNC program, a page fault may occur (from ARNC0 V3.14.0).

Trigger function (G200 or G201) does not work properly when periodic axes (PLCopen period > 0) are used.
NC Software - ARNC0 V5.01.0

ID#555140 : new function since V5.01.0
Displaying current CNC parameters in extended CNC monitor
On program start and each block with G705, the current parameters of the CNC system and axes being used are copied to the extended CNC monitor.

NC Software - ARNC0 V5.00.2

ID#400213244 : solved problem, solved since V5.00.2
Deadlock when restarting NC program
If a synchronous M-function is active at the restart point (M-function groups), then the NC program cannot be aborted immediately after the movement is continued (command ncMOVE, ncCONTINUE). The program is aborted only after the M-function is reset.

NC Software - ARNC0 V5.00.1

ID#400219911 : solved problem, solved since V5.00.1
Wrapping/Mapping: first block after activation will be ignored
Mapping/Wrapping (G17Q / G18Q / G19Q, G217 / G218 / G219): If a move block is programmed immediately after activation, and only the rotary axis carries out a move (lead in move), this block is discarded.

Note:
(1) In the first block after the activation, the rotary axis in addition to the programmed path also performs a lead in movement. If this block is a circular set, the rotary axis must first be moved to the start position of the circle. Otherwise, the path is the sum of the programmed circle and the lead in move.
(2) If Mapping or Wrapping is active, reverse move on the path is prevented (program abort with error 7252)

ID#400224767 : solved problem, solved since V5.00.1
G102 (circle in general orientation)
Under circumstances, it is not recognized if the start point, the end point and the intermediate point lie on a line and the CNC program is not aborted with an error.

NC Software - ARNC0 V5.00.0

ID#546440 : solved problem, solved since V5.00.0
Tracking - Position jump in joint axes at the beginning of desynchronisation phase in combination with G172
At the beginning of the desynchronisation phase a jump occurs in the joint axes when the command TRACKING_OFF is followed by the command G172.

ID#544470 : new function since V5.00.0
ARNC0 Software for SG4 ARM targets
ARNC0 versions V5.00.0 or higher can be used for SG4 ARM target systems.
The following libraries are available for ARM target systems:
- Arnc0man
- Arnc0ext
- Arnc0mx

Note:
For ARM target systems with a Cortex A8 core (e.g. C30), these libraries can not be used.

NC Software - ARNC0 V3.18.5

ID#400280914 : solved problem, solved since V3.18.5
Commands in NC Test or NC Trace permanently blocked
In rare cases, the execution of commands in the NC Test or NC Trace is permanently blocked. If this problem occurs, Automation Studio displays the following message:
Message from '<NcObject>':
The status of the NC object '<NcObject>' is '0'
Would you like to cancel the operation?

Note: The error is fixed in ARNC0 V3.18.5 to V4.99.9 and from V5.06.1 on.

ID#400284686 : solved problem, solved since V3.18.5
Limit value exceeded when movement is aborted and OptMot is active
When aborting a movement, the acceleration limits on the axes may be violated when OptMot is active. A parameter in the OptMot configuration can be used to stretch the deceleration ramp and thus prevent the limit value from being exceeded.
Expansion factor OPTMOT_CONFIG_ADVANCED_type::internals.reserve1[7], the length of the braking ramp is multiplied by this value.
The allowed range is 1.0 to 2.0, for all other values 1.0 is used.

Note: The error is fixed from ARNC0 V3.18.5 to V4.99.9.
NC Software - ARNC0 V3.18.4
ID#632220 : new function since V3.18.4

Tracking

Programmed feed rate is taken into account either in the machine coordinate system (MCS) or in the moving coordinate system (Moving Frame).
The parameter "feedrateMode" in the OptMotConfig structure can be used to select between MCS (=0) and Moving Frame (=1).
This functionality is available from ARNC0 V3.18.4 to V4.99.9 and in all versions starting from V5.08.0.

NC Software - ARNC0 V3.18.3
ID#588155 : solved problem, solved since V3.18.3

Contour violation with active CDC

If a contour violation occurs in a path section immediately before CDC deselection due to a very large tool radius, this is not detected.
ARNC0 always behaves as if G239 is active. In other words, regardless of G39 or G239, a contour violation is always permitted, the program is not aborted or does not attempt to minimize the contour violation.
Note: Problem is fixed only in ARNC0 V3.18.3.

NC Software - ARNC0 V3.18.2
ID#40024767 : solved problem, solved since V3.18.2
G102 (circle in general orientation)

Under circumstances, it is not recognized that the start point, the end point and the intermediate point lie on a line and the CNC program is not aborted with an error.

ID#544330 : solved problem, solved since V3.18.2

Tracking - Position jump in joint axes at the beginning of desynchronisation phase in combination with G172

At the beginning of the desynchronisation phase a jump occurs in the joint axes when the command TRACKING_OFF is followed by the command G172.

ID#40021991 : solved problem, solved since V3.18.2
Wrapping/Mapping: first block after activation will be ignored

Note: Problem is fixed only in ARNC0 V3.18.2.
Mapping/Wrapping (G17Q / G18Q / G190 / G217 / G218 / G219): If a move block is programmed immediately after activation, and only the rotary axis carries out a move (lead in move), this block is discarded.

Note:
(1) In the first block after the activation, the rotary axis in addition to the programmed path also performs a lead in movement. If this block is a circular set, the rotary axis must first be moved to the start position of the circle. Otherwise, the path is the sum of the programmed circle and the lead in move.
(2) If Mapping or Wrapping is active, reverse move on the path is prevented (program abort with erro 7252)

NC Software - ARNC0 V3.18.1
ID#40021607 : solved problem, solved since V3.18.1

Acceleration limits of the axes are exceeded at block transition

Violation may occur under the following conditions:
- a block with a very high speed follows a block with a very low speed
- the linear / rotary axes have a long distance to travel in relation to the CNC axes
- very high acceleration

NC Software - ARNC0 V3.18.0
ID#538790 : solved problem, solved since V3.18.0

Interpreter build in function 'WaitForCommand' causes a syntax error if MPlog is executed as an NC program.

ID#400207758 : new function since V3.18.0

D_FULL additive elements

A behavior of cnc.axis.transformation.d_full_add_el[] for PTP movements can be defined from NC programs via D_FULL_ADD_EL. Additive elements of D_FULL either cause a shift of TCP (i.e. no joints movement; D_FULL_ADD_EL TCP_SHIFT; the only behavior till V3.17.x; default settings since V3.18.0), or they can cause a movement of joints (i.e. TCP does not change its position; D_FULL_ADD_EL JOINTS_SHIFT).

ID#537365 : solved problem, solved since V3.18.0

Error 8141 or 8142: "SW end violated" on circular blocks

In case of circular blocks with 180° angle, error 8141: "Position on the Circle > Positive SW End" or error 8142: "Position on the Circle < Negative SW End" can occur, even if there is no violation of the SW end.
MC_BR_TrackObject in combination with ST-MX program

If a synchronized movement is started by an ST-MX program, the tracking process is not started.

Value of system variable $RADIUS_ERR not applied

If the permissible radius error during center point programming programmed with $RADIUS_ERR, then the value may not always be applied.

Checking the count of arguments for BuildInProcs / BuiltInFns in Language configuration file of the Interpreter

Argument’s attribute “MinCount” is taken in account. If not defined then value of 0 is considered (it is not a mandatory argument).

At program start, the axis limits are written into the DPR trace

Cycle time violation by task “arn0appTskE”

If function ncaccess() or ncalloc() is called cyclically, a cycle time violation in Arnc0 can occur.

Discontinuities in the path speed profile in the case of short blocks (G126)

At the end of short blocks with a subsequent stop, the path speed may be too high and the movement will be terminated immediately. The effect occurs mainly with G126 in combination with movement reversal (block transition with 180°).

No limitation of the allowed value range for v_jump or a_jump (data type REAL)

Automatic correction of radius errors for center point programming of circular blocks

If the center point of the circle and target point are programmed inaccurately, then the programmed target point is not moved to precisely. With active correction, the center point of the circle is shifted such that intervals "Starting point - Midpoint" and "Midpoint - Endpoint" are the same and the programmed target point is reached.

Correction is enabled with system variable $RADIUS_ERR_MODE. Various methods for calculating the midpoint can be selected.

Separate status for each displayed when initializing limit values, decoder (interpreter), axis settings and CNC PLC data

Status "global.init" is set to ncTRUE if the initialization of the axes, limit values, decoder (path planner) and CNC PLC data is completed successfully (either with command ncGLOBAL or ncINIT or by initializing the individual components).

NC objects are now generated with property “project-independent”

Property “project-independent” is now set for NC objects that are newly generated on the target system. As a result, they will not be deleted during a transfer if they do not exist in the Automation Studio project (if no “initial transfer” is performed).

Bezier points for G806 and G807 spline consider G818 (SMOOTH_ORIENTATION_CHANGE; quaternions) for an orientation part of TCP description.
WS_D_FULL instead of D_FULL for workspace monitoring
WS_D_FULL can be used instead of D_FULL for definition of a tool for all kinds of workspace monitoring (e.g. cyclic, inter-channel).

Additional tool point for D_FULL and WS_D_FULL
D_FULL and WS_D_FULL were extended with parameter DMP providing a definition of additional tool point for workspace monitoring. Available only for selected mechanical systems.

Movements after a collision detected at cyclic workspace monitoring
If a movement is stopped in cyclic set-position generator of ARNC0 (e.g. a collision detected by cyclic or inter-channel workspace monitoring) and jolt filters are defined for axes (cnc.axis.axis[i].t_axfilter), then new G-code G840 (alias SKIP_FILTER_AFTER_CYCLIC_ERROR) can change the filter behavior (for all axes in the respective CNC channel). Since especially for long filters, one could interpret a movement so that it continued even after the reported error. With G840, the positions buffered in the filter are skipped if there is an error, and therefore a movement is stopped immediately. G840 is valid from a line where it is used till the end of the main NC program.

Serial wire-frame models for external inter-channel collision detection
ARNC0INTERACTION_typ was extended so that the user can read and modify a data of external serial wire-frame models via CncChannelsInteraction PV.

Wait time after CNC program end or program abort
If an axis jolt filter has been parameterized on the drive, this filter time was waited after the end of a CNC program or after a program abort, even if the use of the filter on the drive was deactivated (cnc_obj-> axis.axis[i].drive_axfilter = ncOFF). With the setting ‘cnc_obj-> axis.axis[i].drive_axfilter = ncOFF’ on all axes in the CNC channel, this time is no longer waited.

Calling of subprogram in FILE_XL mode
Calling of subprogram in FILE_XL mode can cause a pagefault if the subprogram has been already loaded.

Mapping (G217/G218/G219), new mode for activation ($MAPPING_MODE = 2)
With absolute programming (G90), the position of the ncROTARY axis is rigidly coupled to the programmed position of the programmed axis (Cartesian axis). In the first block after activation of mapping, the programmed path speed is equal to the geometric sum of the speeds of the axis of rotation and all cartesian axes that are not assigned to the ncROTARY axis.

Tracking - Disable the additional position monitoring of the axes
In tracking sections, an additional monitoring of the joint axes position can be disabled by setting a bit (OPTMOT_CONFIG_ADVANCED_typ::internals.reserve2[3] != 0). It is not recommended.

External inter-channel collision detection
Collision detection between CNC channel and external wire-frame models is available via WS_CTRL_ON EXTERNAL_INTERCHANNEL_ON. External wire-frame models are configured via PV CncChannelsInteraction (ARNC0INTERACTION_typ).
ID#514900 : solved problem, solved since V3.16.1
IEC-ST function is unsafe if returns string
IEC-ST function configured to return an unbound string could cause page fault if the return value is referenced more then once.

ID#400189668 : solved problem, solved since V3.16.1
OptMot - Deadlock at the end of OptMot-Section fixed

NC Software - ARNC0 V3.16.0
ID#511265 : solved problem, solved since V3.16.0
Axis position may jump at program abort if accuracy hold mode G603 is used
ID#400203125 : solved problem, solved since V3.16.0
Reduction of the path speed in short blocks before G170, although the decoder synchronization has already taken place
ID#400201411 : solved problem, solved since V3.16.0
CNC channel with robot and CDC
A combination of CDC and blocking non-movement (e.g. G04) caused position jump on joint axes if robot was configured.
ID#400183576 : solved problem, solved since V3.16.0
ARNC0 got stuck if the abort command was sent very close to the end of NC program
The error caused that the whole system became unresponsive.

NC Software - ARNC0 V3.15.1
ID#400192293 : solved problem, solved since V3.15.1
Wrong limits are used if G107 with parameter index -1 is programmed
ID#506255 : new function since V3.15.1
Exclusive workspace monitoring is not active if no exclusive block is defined.
ID#503055 : solved problem, solved since V3.15.1
It is possible that system variable $ACC_HOLD_ADD_T_WAIT is not set to value zero at the beginning of the program
ID#502655 : new function since V3.15.1
With G601/G602/G603 the behavior at block transition can be determined for accuracy hold (G60)
In the case of the standard behavior (G601), an accuracy hold it is checked for each axis whether the programmed position is reached and the axis speed is zero.
With G602 and G603 only the corresponding times are waited (for details see ARNC0 documentation).
ID#502555 : solved problem, solved since V3.15.1
Unloading of a IEC-ST program could cause page fault if the program contains variable declaration with alias
ID#500910 : solved problem, solved since V3.15.1
Feed forward torque peak at the beginning of a program
For a mechanical system with coupling a torque peak could occur at the beginning of a program.
ID#400199142 : solved problem, solved since V3.15.1
Workspace monitoring and coupling
Workspace monitoring (cyclic) did not work properly if coupling was used at mechanical system.
ID#499455 : solved problem, solved since V3.15.1
Multiple output of M-Functions when selecting CDC
When activating CDC with G137, it is possible that programmed M-Functions are output both at the beginning as well as at the end of the lead-in movement.

NC Software - ARNC0 V3.15.0
ID#499505 : solved problem, solved since V3.15.0
Restart-Mode 'ncABORT_DATA' is allowed for ncROTARY axes even when a kinematic transformation is active
ID#495340 : solved problem, solved since V3.15.0
Pagefault at the restart of a CNC program.
The pagefault can occur if an axis position is out of tolerance during restart of program.

OptMot - the Profile generated by OptMot is not correct in ARSim
The desired velocity profile is not reached with OptMot in ARSim. Note: The behavior of the override with OptMot has changed.

NC Software - ARNC0 V3.14.6

With G601/G602/G603 the behavior at block transition can be determined for accuracy hold (G60)

In the case of the standard behavior (G601), at an accuracy hold it is checked for each axis whether the programmed position is reached and the axis speed is zero.

With G602 and G603 only the corresponding times are waited. For details see ARNC0 documentation.

Note:
Extension is not included in ARNC0 V3.15.0.

It may happen that the system variable $ACC_HOLD_ADD_T_WAIT is not set to the zero at CNC program start

Note:
Error fix is not included in ARNC0 V3.15.0.

It's not allowed to program movements with length zero in synchronous and asynchronous section of tracking. In the function to check the length of the programmed movement a bug was fixed.

NC Software - ARNC0 V3.14.5

NC Software - ARNC0 V3.14.4

Writing to a path-synchronous dynamic variable wasn't done path-synchronously

Rounding on optimized section in ST
Rounding between PTP and linear movement was skipped when optimizer was turned on.

Function return value
Function return value of type structure could contain invalid data when the same function is called more than once at the same line (or IEC/ST statement).

NC Software - ARNC0 V3.14.3

Deadlock with MC_BR_MoveProgram when combined with tracking
A deadlock can occur when using MC_BR_MoveProgram to start a synchronisation movement. The Done output of the MC_BR_MoveProgram was not set.

NC Software - ARNC0 V3.14.2

Restart of program
- path synchronous variables might not be restored with value from restart buffer before restart point
- a new format of restart parameter buffer (restart data is not compatible with previous ARNC0 version)
- All system variables #restart.playback, #restart.record has been changed to #thread.restart.playback, #thread.restart.record

Deadlock with long CNC programs
A deadlock is possible with extremely long CNC programs (accumulation of rounding errors when calculating the axis jolt filter).
The algorithm for calculating the axis jolt filter has been improved. This effect occurs very seldom. If problems persist in individual cases, G251
can be used to select an alternative calculating method.

ID#400127280 : solved problem, solved since V3.14.2
Position jumps if CDC and “Cam wrapping” (G17Q/G18Q/G19Q) or “Mapping” (G217/G218/G219) active
If contour violations occur due to short path sections while CDC is active (G39 active) and function “Cam wrapping” (G17Q/G18Q/G19Q) or “Mapping” (G217/G218/G219) is active, then position jumps on the axes are possible.

ID#489890 : new function since V3.14.2
Detection of NaN values
Interpreter checks motion packets for NaN and Inf values to prevent unexpected motions. The feature can be controlled using the attribute CheckForNaNs of XML element MPQUEUE (default value is 1).

ID#489330 : Information valid since V3.14.2
CNC program start accelerated
The time between the start command ncaction(ncPROGRAM, ncSTART) and starting the NC program has been shortened (about 5 CNC cycles).

ID#488025 : solved problem, solved since V3.14.2
Additional OptMot-monitor outputs without filters
In addition to the existing velocity outputs in the OptMot-monitor structure (monitor.v_joint) for which filters and overrides are taken into consideration, also unfiltered values are available.

ID#400181370 : solved problem, solved since V3.14.2
OptMot - asymmetrical acceleration profile
Correction of the computation of the internal sample time of OptMot and the thereby maximally reachable velocity.

NC Software - ARNC0 V3.14.1
ID#485100 : new function since V3.14.1
Information about ‘inline’ attribute change was added
Arguments of inlined AIL templates (G-Codes, built-in-procedures, etc.) are allowed to be used only once inside a template definition. In other case the attribute ‘inline’ is cleared automatically. Warning about this attribute change is newly reported (additional information contain template name, language index and argument name).

ID#400187062 : solved problem, solved since V3.14.1
Position jump if operation mode ‘cam wrapping’ (G17Q/G18Q/G19Q) or ‘mapping’ (G217/G218/G219) is deactivated
A position jump occurs if the operation mode ‘cam wrapping’ (G17Q/G18Q/G19Q) or ‘mapping’ (G217/G218/G219) is activated and deactivated immediately after the first move block.

ID#400184734 : solved problem, solved since V3.14.1
Invalid target position at positioning on the path (ncNCPR_POS, ncSTART)

ID#400125928 : solved problem, solved since V3.14.1
CAM wrapping: position jump during backward movement on the path
Backward movement on the path (e.g. with ncNCPR_POS, ncSTART) can lead to an axis position jump when a block with CAM wrapping (G17Q / G18Q/G19Q) is achieved.

NC Software - ARNC0 V3.14.0
ID#480850 : solved problem, solved since V3.14.0
OptMot - Defective jolt profile in specific situations
In special situations ARNC0 is not handling the acceleration/jolt profile computed by OptMot correctly, which leads to a not smooth dynamic profile.

ID#400186306 : solved problem, solved since V3.14.0
OptMot - Not smooth jolt profile in combination with specific limit configurations
Specific limit configurations for OptMot result in not smooth acceleration profiles.

ID#479710 : new function since V3.14.0
OptMot - Timing problems with command CONDITIONAL_STOP_OPTMOT
Because of timing problems it could happen that the identification of the command CONDITIONAL_STOP_OPTMOT was not published in the application task.

ID#479610 : solved problem, solved since V3.14.0
Calling of global subprogram with specified count of repetition was not supported

ID#479120 : new function since V3.14.0
Support additional return types in IEC-ST function
IEC-ST functions can return structure, array or string.

ID#478970 : solved problem, solved since V3.14.0
Wrong target position of an automatic tangential axis (G141)
If after a rapid move block (G00) a blocks with G170 follows, it may happen that target position of the automatic tangential axis is not correct. The same behavior can also happen with the combinations G01+G144, G02+G144 or G144+G03.

ID#400186120 : solved problem, solved since V3.14.0
Deadlock if an automatic tangential axis is active (G144) and a G172 command is programmed after a G00 block.
The same behaviour can also happen if a G170 is programmed after move block with G144. In both cases the NC program is aborted with error 8276.

ID#478445 : new function since V3.14.0
New system variable %_CNC.frame_G92 for an access to the actual G92 frame

ID#477690 : solved problem, solved since V3.14.0
Page fault after the axis configuration has been changed
In rare cases, a page fault occurs, if the axis configuration in a CNC channel has been changed.

ID#477430 : solved problem, solved since V3.14.0
NC program got stuck if it was started after restart info which returned error

ID#477140 : new function since V3.14.0
Position Display Mode in the CNC Monitor can be set in the CNC init parameter module
Position display mode in the CNC monitor can be set in the CNC init parameter module: ARNCCNC_typ.monitor.parameter.pos_mode (ncOFF / ncCSTRF / ncTOOLDAT / ncCSTRF+ncTOOLDAT), the setting is applied at system startup into the user data ARNCCNC_typ.
The position display mode is set during global init of the CNC system according the value in the user data ARNCCNC_typ: monitor.parameter.pos_mode = ncCSTRF, ncTOOLDAT oder ncCSTRF+ncTOOLDAT: display mode is switched on ( ncPOSMON, ncSWITCH_ON)

ID#476890 : new function since V3.14.0
Local variable of any type can be defined in AIL language. If no type specified LREAL will be used.

ID#476575 : solved problem, solved since V3.14.0
Restart didn't support checking of time stamp and length of global subprograms.
Checking of time stamp and length of subprogram is possible to turn off by setting restart_type = ... + ncNO_CHECK.

ID#400183106 : solved problem, solved since V3.14.0
The data type of system variables $MA_POS_LIMIT_PLUS, $MA_POS_LIMIT_MINUS and $MA_REFP_SET_POS has been changed from DINT to REAL.

ID#400183351 : solved problem, solved since V3.14.0
External function used wrong argument values
External function could use wrong argument values when the argument list of the external function contained a string argument. The argument values behind the string argument could be damaged.

ID#475175 : solved problem, solved since V3.14.0
Restart didn't support logging of string variables to restart buffer.

ID#474695 : solved problem, solved since V3.14.0
Restart from block number was possible in main program only.
ID#474370 : solved problem, solved since V3.14.0
Status for RESTART-INFO
If RESTART-INFO run is aborted by error 7159: "Restart aborted at end of NC program (restart position not found)" the restart.info.status.active flag isn't reseted.

ID#473045 : new function since V3.14.0
OptMot - Additional parameter added to be able to adjust the limitation of the minimal duration of the synchon and desynchron movement (Tracking)

ID#472350 : new function since V3.14.0
Protect NC_GLOBAL types from misuse
Once an NC_GLOBAL data type is defined it cannot be deleted.

ID# 400181092, 400181299 : solved problem, solved since V3.14.0
MX frame which represents very small rotations around the basic coordinate system could cause the numerical problems in transition between circular and linear movement

ID#471165 : solved problem, solved since V3.14.0
Passing of incompatible IN_OUT string argument to IEC-ST function could cause memory corruption

ID#400179219 : solved problem, solved since V3.14.0
G39: Contour violation caused by the combination of great tool radius and short blocks.
If after a block sequence with allowed contour violation (G39) a CDC deactivation is done, a large contour violation may occur. The algorithm for minimizing an allowed contour violation (G39) has been improved.

ID# 400180388, 400179947 : new function since V3.14.0
A name of a transformation variable (cnc.axis.transformation.name) extended to 100 characters

ID#469850 : new function since V3.14.0
Status if set positions in CNC monitor are valid
<cnc_object>.monitor.status.s_set_valid = ncTRUE: In the CNC monitor displayed set positions are valid
<cnc_object>.monitor.status.s_set_valid = ncFALSE: In the CNC monitor displayed set positions are not valid

ID#400178912 : solved problem, solved since V3.14.0
G10 or G36 may be ignored if programmed in a line with other G codes

ID#400174669 : solved problem, solved since V3.14.0
Single Step Operating Mode: Mode "ncLINENUMBER" can not be selected in NC Test

ID#400172832 : solved problem, solved since V3.14.0
Error 7743: "Position out of 'In-Position-Tolerance' at START/RESTART of CNC move" occurs sporadically at CNC program start

ID#400149412 : solved problem, solved since V3.14.0
Invalid RESTART POINT determined
Depending on the sequence of move blocks and non-move blocks in a CNC program, it can happen that an invalid RESTART POINT will be calculated. The RESTART will be aborted with error 7743.

NC Software - ARNC0 V3.13.0

ID#400178212 : solved problem, solved since V3.13.0
Deadlock when running a CNC program
A deadlock can occur when a CNC program with synchronous M functions, G170 or G172 is executed. The blocking CNC program can be aborted by the user.

ID#400179958 : solved problem, solved since V3.13.0
Automatic tangential axis could move wrongly in combination with rotation of the coordinate system.

ID#400179207 : new function since V3.13.0
Control blocks FOR and SWITCH newly supported in AIL.

NC Software - ARNC0 V3.12.2

ID#476995 : solved problem, solved since V3.12.2
Restart didn't support checking of time stamp and length of global subprograms.
Checking of time stamp and length of subprogram is possible to turn off by setting
restart_type = ... + ncNO_CHECK.
Fixed only in ARNC0 V3.12.2

ID#476120 : solved problem, solved since V3.12.2
Restart didn't support logging of string variables to restart buffer
Fixed only in ARNC0 V3.12.2

ID#476090 : solved problem, solved since V3.12.2
Status for RESTART-INFO
If RESTART-INFO run is aborted by error 7159: "Restart aborted at end of NC program (restart position not found)" the
restart.info.status.active flag isn't reseted.
Fixed only in ARNC0 V3.12.2

ID#476085 : solved problem, solved since V3.12.2
Restart from block number was possible in main program only
Fixed only in ARNC0 V3.12.2

NC Software - ARNC0 V3.12.1

ID#467660 : solved problem, solved since V3.12.1
Target coordinate programmed to value with flag 'undef' was not ignored.
Arguments may be marked as 'undef' (e.g. function argument is missing). When using argument with flag 'undef' for positioning, the argument
is then ignored.

ID#400176110 : solved problem, solved since V3.12.1
Unexpected error 15863
This error might occur when some enumeration data type is redefined, and DeclareEnumLiteralsAsConstants="Yes" is set in Interpreter config
file gmcipiec (its default value is "No").

ID#400176112 : solved problem, solved since V3.12.1
The IEC-ST statement CYCLIC_BL or interpreter synchronous blocking FB could cause unexpected error during high CPU load.

ID#466890 : solved problem, solved since V3.12.1
Function block declaration followed by its implementation caused array initialization problems.

ID#465830 : solved problem, solved since V3.12.1
Waiting time too short for exact stop
For ACP10 axes in a CNC channel (CNC_enabled="1"), the waiting time until the next command is too short for an exact hold (e.g. G60).
Correct behavior: As soon as the path section end has passed the path jolt filter (t_jolt) and all ARNC0 axis jolt filters (t_axfilter), the system
waits the length of the settling time (t_in_pos) and total delay time (t_total) of the axes.
With the new system variable $ACC_HOLD_ADD_T_WAIT, an additional waiting time can be programmed for an exact stop.

NC Software - ARNC0 V3.12.0

ID#462545 : solved problem, solved since V3.12.0
Positions in extended CNC monitor (ARNC0NCMON_EXT_typ) are not displayed with full accuracy

ID#461940 : solved problem, solved since V3.12.0
Systemvariable "$MAPPING_MODE" causes a syntax error
Selection of operating mode for cam wrapping (G17Q/G18Q/G19Q) or mapping (G217/G218/G219) by the system variable
"$MAPPING_MODE" or "MAPPING_MODE". So far "$MAPPING_MODE" causes a syntax error.

ID#459585 : new function since V3.12.0
Extension of restart info data
When determining the RESTART-INFO data additionally the line number is displayed.
New restart type 'restart.info.parameter.restart_type = ncABORT_DATA'.
Restart date stored at program abort will be shown (data are only displayed no preparation for a possible restart is done).

NC Software - ARNC0 V3.11.1

ID#473040 : new function since V3.11.1
OptMot - Additional parameter added to be able to adjust the limitation of the minimal duration of the synchron and desynchron movement
(Tracking). Extension only in ARNC0 V3.11.1
**NC Software - ARNC0 V3.11.0**

ID#458720 : solved problem, solved since V3.11.0

OptMot - Correction of a defect in the automatic computation of the internal sample time

ID#458075 : solved problem, solved since V3.11.0

MX command GetTool returned invalid tool values

The MX command GetTool returned invalid tool values if such was called before any tool was set.

ID#458040 : new function since V3.11.0

Interpreter single step halts program execution of local subprogram at line following program label.

ID#457540 : new function since V3.11.0

DNC interface: Data type declaration for DNC buffer header (ARNC0DNCHEADER_typ) in the ARNC0 library

ID#456555 : new function since V3.11.0

New AIL built-in function GetVarValue

The function takes name of variable (string) and writes its value on specified address. The variable is searched first in local scope, then IP-global and finally in NC/PLC global scope.

**NC Software - ARNC0 V3.10.2**

ID#400176112 : solved problem, solved since V3.10.2

The IEC-ST statement CYCLIC_BL or interpreter synchronous blocking FB could cause unexpected error during high CPU load.

**NC Software - ARNC0 V3.10.1**

ID#400167447 : solved problem, solved since V3.10.1

Data in CNC monitor deleted when starting program

Monitoring data is reset when starting a CNC program (cnc_object.move.status.cnt_ncprog = 1).

cnc_object.monitor.line_ncprog = 0

cnc_object.monitor.block_ncprog = 0

cnc_object.monitor.nr_ncprog = 0

cnc_object.monitor.name_ncprog = ""

As soon as the data is available, the current values are displayed.

ID#400169151 : solved problem, solved since V3.10.1

Random additional information for error 9249, 9250, 9270 and 9271 (CDC activation / deactivation)

ID#453900 : new function since V3.10.1

Single step can now be switched on/off while program runs after restart

ID#453785 : solved problem, solved since V3.10.1

Error when a new program is started immediately after the end of a CNC program or a global init of the CNC system is performed

At the end of the program, the variable move.status.cnt_nc_prog is set to zero although in the ARNC0 not all operations are finished. If e.g. a new program will be started immediately, so this will be aborted.

ID#453535 : solved problem, solved since V3.10.1

Deadlock by error 40112: “Timeout at processing an ARNC0 command”

Error 40112 may lead to a deadlock. Error 40112 may occur when a ncaction () is called, and this is not allowed in the current state. For example, Error 7146 occurs when ncGLOBAL, ncINIT is called during an active CNC program, and this leads to the subsequent error 40112.

ID#453290 : solved problem, solved since V3.10.1

Bug in built-in function EXEC_RPC

Repositioning to an empty line in IEC/ST program caused page fault.

ID#453275 : solved problem, solved since V3.10.1

OptMot - Position error at tool change with D_FULL and OptMot

ID#452235 : solved problem, solved since V3.10.1

Dead lock when using ParID Trace for ARNC0 axis

ParID Trace is not exited and cannot be aborted when started for an ARNC0 axis (axis on a virtual ARNC0 interface). The start of the trace is prevented by error 40265.

ID#452120 : new function since V3.10.1
Calculation of feed forward torque with increased accuracy

ID#400164007 : solved problem, solved since V3.10.1

Deadlock when starting a CNC program

When a CNC program with an automatic tangential axis (G140) is aborted by an error, it can happen that the next program start is blocked by a deadlock.

The lock can be resolved only by rebooting the controller.

The error can also occur if G17Q / G18Q / G19Q or G217 / G218 / G219 is used in the aborted program.

NC Software - ARNC0 V3.10.0

ID#451415 : Information valid since V3.10.0

Interfaces not longer supported in ARNC0

Not longer supported interfaces (ARNC0 configuration):
- POWERLINK interface
- virtual Interface with the name "SDC:<interface name>"
- virtual interface in mode "1"
- virtual interface in mode "3"
- virtual interface in mode "8"

If one of these interfaces is defined in the ARNC0 configuration, the ARNC0 startup will be aborted (entry in the logger).

ID#450360 : solved problem, solved since V3.10.0

The IEC/ST statement CYCLIC_BL didn't block Interpreter execution in case of interpreter single step or break point.

ID#450070 : new function since V3.10.0

New built-in functions EXEC_RPC introduced

Statement to start programs in IEC-ST. Unlike EXEC, the start position can be freely defined and can also be in subroutines.

ID#400165648 : solved problem, solved since V3.10.0

Signal functions (G220, G221, G222) in combination with CDC

When using signal functions (G220, G221, G222) in sections with active CDC, it may happen that the signal is output at a wrong position or at a wrong time or not at all. Possibly warning 7179 may arise.

ID#400164244 : solved problem, solved since V3.10.0

Page fault when calling a path synchronous function and restart is switched on

ID#400164219 : solved problem, solved since V3.10.0

Stack corruption when using more then 32 local variables inside of an AIL template.

ID#400162047 : solved problem, solved since V3.10.0

Deadlock when using GmcGrpAPI FUB's with G126

When using GmcGrpAPI FBK's with GmcArncGrp implementation may lead to a Deadlock when function G126 is still active at the end of CNC program.

ID#47235 : new function since V3.10.0

Error 7185 is plitted into 7185 and 7198

Error 7185; "Position modulo out of 'In-Position-Tolerance' at RESTART of CNC move" may be generated in the ARNC0 at two different locations, so it is output either as error 7185 or as error 7198 (this step serves to simplify the SW development).

ID#400163820 : solved problem, solved since V3.10.0

Cycle time violation at CNC program restart

A cycle time violation may occur at program abort if a restart buffer with a large number of trigger events is defined.

ID#446975 : solved problem, solved since V3.10.0

Wrong restart position if a ncROTARY axis is defined and restart mode is ncPATHDISTANCE

Abort position is always used for restart, if at least one ncROTARY axis is defined in the CNC channel and restart mode ncPATHDISTANCE is used.

ID#400159890 : new function since V3.10.0

Better detection of compatible types in IEC/ST

ID#400154713 : solved problem, solved since V3.10.0

Undesired movement of the tangential axis used in the transformation
The tangential axis in connection with the transformation could cause an undesired movement of the real axis in case of stopping the movement (e.g. end of program, G04).

ID# 400148704, 400153629 : solved problem, solved since V3.10.0
Retsart of ncROTARY axes
The restart of a CNC program is denied with error 7199 when a kinematic transformation (also RTCP) is active and axes of the type ncROTARY exist in the project.

ID#400151152 : solved problem, solved since V3.10.0
Deadlock when using GmcGrpAPI FUB's and CDC
When using GmcGrpAPI FBK's with GmcArcGrp implementation may lead to a Dedlock when CDC is still active at the end of CNC program.

NC Software - ARNC0 V2.52.5
ID#587235 : solved problem, solved since V2.52.5
Contour violation with active CDC
If a contour violation occurs in a path section immediately before CDC deselection due to a very large tool radius, this is not detected. ARNC0 always behaves as if G239 is active. In other words, regardless of G39 or G239, a contour violation is always permitted, the program is not aborted or does not attempt to minimize the contour violation.

ID#400223311 : solved problem, solved since V2.52.5
G05/G06 with coordinate system rotation (G92, G192, G292, etc.)
With G05 or G06, error 10452: "Polar coordinate machine - programmed rotation/shift plane not allowed" occurs if the coordinate system has been rotated and then rotated back again.
Note:
G05 and G06 are only permitted if the coordinate system is not rotated (10385: "Polaraxis must not be programmed (Rotation is active)").

ID#587220 : solved problem, solved since V2.52.5
Warning 8154 after blocks with G05 or G06
If a line with a non-moving block (e.g. G04, M function) follows immediately after a block with G05 or G06, warning 8154: "NC block with move distance equal to 0.0" is output.

ID#400238736 : solved problem, solved since V2.52.5
Error 40144 at program abort
If a CNC program is aborted immediately after a G201, error 40144 (Internal Error - Job ID already acknowledged) may arise.

NC Software - ARNC0 V2.52.4
ID#400209483 : solved problem, solved since V2.52.4
Discontinuities in the path speed profile in the case of short blocks (G126)
At the end of short blocks with a subsequent stop, the path speed may be too high and the movement will be terminated immediately. The effect occurs mainly with G126 in combination with movement reversal (block transition with 180°).

ID#400216072 : solved problem, solved since V2.52.4
Acceleration limits of the axes are exceeded at block transition
Violation may occur under the following conditions:
- a block with a very high speed follows a block with a very low speed
- the linear / rotary axes have a long distance to travel in relation to the CNC axes
- very high acceleration

NC Software - ARNC0 V2.52.3
ID#535300 : solved problem, solved since V2.52.3
Value of system variable $RADIUS_ERR not applied
If the permissible radius error during center point programming programmed with $RADIUS_ERR, then the value may not always be applied.

ID#400200645 : solved problem, solved since V2.52.3
Automatic correction of radius errors for center point programming of circular blocks
If the center point of the circle and target point are programmed inaccurately, then the programmed target point is not moved to precisely.
With active correction, the center point of the circle is shifted such that intervals "Startling point - Midpoint" and "Midpoint - Endpoint" are the same and the programmed target point is reached.
Correction is enabled with system variable $RADIUS_ERR_MODE. Various methods for calculating the midpoint can be selected.
Wait time after CNC program end or program abort

If an axis jolt filter has been parameterized on the drive, this filter time is waited after the end of a CNC program or after a program abort, even if the use of the filter on the drive is deactivated (cnc_obj->axis.axis[].drive_axisfilter = ncOFF).

With the setting ‘cnc_obj->axis.axis[].drive_axisfilter = ncOFF’ (on all axes in the CNC channel), this time is no longer waited.

CAUTION: From Arnc0 V2.52.2 the Library ACP10_MC V2.52.3 or higher is required.

Multiple output of M-Functions when selecting CDC

When activating CDC with G137, it is possible that programmed M-Functions are output both at the beginning as well as at the end of the lead-in movement.

Pagefault at the restart of a CNC program.

The pagefault can occur if an axis position is out of tolerance during restart of program.

Deadlock with long CNC programs

A deadlock is possible with extremely long CNC programs (accumulation of rounding errors when calculating the axis jolt filter).

The algorithm for calculating the axis jolt filter has been improved. If problems persist in individual cases, G251 can be used to select an alternative calculating method.

Correction of the computation of the internal sample time of OptMot and the thereby maximally reachable velocity.

In addition to the existing velocity outputs in the OptMot-monitor structure (monitor.v_joint) for which filters and overrides are taken into consideration, also unfiltered values should be available.

If contour violations occur due to short path sections while CDC is active (G39 active) and function "Cam wrapping" (G17Q/G18Q/G19Q) or "Mapping" (G217/G218/G219) is active, then position jumps on the axes are possible.

Backward movement on the path (e.g. with ncNCPR_POS, ncSTART) can lead to an axis position jump when a block with CAM wrapping (G17Q/G18Q/G19Q) is achieved.

Position jumps if operation mode cam wrapping (G17Q/G18Q/G19Q) or mapping (G217/G218/G219) is deactivated

A position jump occurs if the operation mode cam wrapping (G17Q/G18Q/G19Q) or mapping (G217/G218/G219) is activated and deactivated immediately after the first move block.

Wrong target position for positioning on the CNC path (ncNCPR_POS, ncSTART)

In special situations ARNC0 is not handling the acceleration/jolt profile computed by OptMot correctly, which leads to a not smooth dynamic profile.
OptMot - Not smooth jolt profile in combination with specific limit configurations

Specific limit configurations for OptMot result in not smooth acceleration profiles.

ID#400172832 : solved problem, solved since V2.51.6
Error 7743: Position out of 'In-Position-Tolerance' at START/RESTART of CNC move" occurs sporadically at CNC program start

ID#475420 : solved problem, solved since V2.51.6
Status for RESTART-INFO
If RESTART-INFO run is aborted by error "7159: Restart aborted at end of NC program" the restart.info.status.active flag isn't reseted.

ID#400179219 : solved problem, solved since V2.51.6
G39: Contour violation caused by the combination of great tool radius and short blocks
If after a block sequence with allowed contour violation (G39) a CDC deactivation is made, a large contour violation may occur. The algorithm for minimizing an allowed contour violation (G39) has been improved.

NC Software - ARNC0 V2.51.5

ID#400178212 : solved problem, solved since V2.51.5
Deadlock when running a CNC program
A deadlock can occur when a CNC program with synchronous M functions, G170 or G172 is executed. The blocking CNC program can be aborted by the user.

NC Software - ARNC0 V2.51.4

ID#467485 : solved problem, solved since V2.51.4
Waiting time too short for exact stop
For ACP10 axes in a CNC channel (CNC_enabled="1"), the waiting time until the next command is too short for an exact hold (e.g. G60). Correct behavior: As soon as the path section end has passed the path jolt filter (t_jolt) and all ARNC0 axis jolt filters (t_axfilter), the system waits the length of the settling time (t_in_pos) and total delay time (t_total) of the axes. With the new system variable $ACC_HOLD_ADD_T_WAIT, an additional waiting time can be programmed for an exact stop.

ID#462560 : solved problem, solved since V2.51.4
Positions in extended CNC monitor (ARNC0NCMON_EXT_typ) are not displayed with full accuracy

ID#462045 : solved problem, solved since V2.51.4
Systemvariable "$MAPPING_MODE" causes a syntax error
Selection of operating mode for cam wrapping (G17Q/G18Q/G19Q) or mapping (G217/G218/G219) by the system variable "$MAPPING_MODE" or "MAPPING_MODE". So far "$MAPPING_MODE" causes a syntax error.

ID#457850 : new function since V2.51.4
DNC interface: Data type declaration for DNC buffer header (ARNC0DNCHEADER_typ) in the ARNC0 library

ID#457845 : new function since V2.51.4
Calculation of feed forward torque with increased accuracy

ID#400140820 : solved problem, solved since V2.51.4
Error 14198: "Error by acp10 mode switch" if a Quickstop is triggered
For axes on a POWERLink interface on the ARNC0 error 14198: "Error by acp10 mode switch" may occur if a Quickstop was triggered.

NC Software - ARNC0 V2.51.3

ID#455120 : solved problem, solved since V2.51.3
Deadlock by error 40112: "Timeout at processing an ARNC0 command"
Error 40112 may lead to a deadlock. Error 40112 may occur when a ncaction () is called, and this is not allowed in the current state. For example, Error 7146 occurs when ncGLOBAL, ncINIT is called during an active CNC program, and this leads to the subsequent error 40112. This bug is also in ARNC0 V3.10.1 and higher fixed.

ID#455105 : solved problem, solved since V2.51.3
Error when a new program is started immediately after the end of a CNC program or a global init of the CNC system is performed
At the end of the program, the variable move.status.cnt_nc_prog is set to zero although in the ARNC0 not all operations are finished. If e.g. a new program will be started immediately, so this will be aborted. This bug is also in ARNC0 V3.10.1 and higher fixed.
**NC Software - ARNC0 V2.51.2**

ID#44650 : solved problem, solved since V2.51.2

Page fault in IEC/ST function EXEC

The new extension for IEC/ST function EXEC (introduced in V2.51.1) could cause page fault if destination location contained a movement.

---

ID#400160665 : solved problem, solved since V2.51.2

OptMot - first part of bezier spline (G126) skipped

---

**NC Software - ARNC0 V2.51.1**

ID#446150 : new function since V2.51.1

IEC/ST function EXEC extended

New optional arguments (block, line) added to EXEC command (to start from specified block or line number).

---

ID#444895 : solved problem, solved since V2.51.1

Overloaded G-Functions and M-Functions called by alias did not work properly

When calling overloaded G-Function or M-Function using its alias, wrong template attributes could be used (e.g. old evaluation phase).

---

ID#444410 : new function since V2.51.1

Cubic Bezier splines - a description of Bezier points was changed

Bezier points BP_1 and BP_2 are valid for one spline (G806/G807). If they are programmed relatively, then it refers to the current position. If they are not programmed, or if they are programmed only for some axes, then missing coordinates are used from the current position.

---

ID#400161058 : solved problem, solved since V2.51.1

Incorrect position of the tangential axis in circular blocks with an arc length greater than a full circle

In circular blocks with an angle greater than 360° for the path distance of the tangential axis only the modulo value (max. 360°) is used.

---

ID#345222 : solved problem, solved since V2.51.1

Unexpected full circle in circular blocks

A full circle is traveled if a circular block with arc length zero results from cutter diameter compensation, TrigHelp (G183) etc.

Note:
It is not possible to program a circular block with arc length zero in the CNC program.

---

ID#224907 : solved problem, solved since V2.51.1

Initialization of CNC block monitor at program start

CNC block monitor was first time initialized with execution of the first path command.

---

**NC Software - ARNC0 V2.51.0**

ID#442395 : new function since V2.51.0

D_FULL and frame axes additive elements

D_FULL and frame axes additive elements (d_full_add_el[], frame_add_el[]) were introduced. They can be used analogously to existing tcp_add_el[].

---

ID#442390 : solved problem, solved since V2.51.0

Mechanical coupling

If mechanical coupling is defined on joint axes, then workspace monitoring considered wrong position of robot. Joint axes positions themselves were wrong as well on some non-movement NC blocks (e.g. dwell time at the beginning of NC program).

---

ID#441605 : new function since V2.51.0

arn0add_typ file not necessary anymore

Declarations in arncoadd_typ file are moved to arnc0man_typ (arn0man.h) file.

---

ID#400158663 : solved problem, solved since V2.51.0

Path-synchronous function blocks didn’t work properly when negative override

Path-synchronous function blocks were not executed when backwards movement on path. The behavior was not properly, also page fault could occur.
ID#439395: new function since V2.51.0
System variables $AA_IW and $AA_IM will be updated first time some CNC cycles after program start

ID#439265: new function since V2.51.0
Global variables can be used also inside of IEC-ST functions when using prefix ::

ID#439015: solved problem, solved since V2.51.0
Restarting from wrong byte-offset or line
Information about byte offset and line number for restart could be wrongly interpreted when coordinate system was moved/rotated (e.g. G92) inside of a NC program.

ID#438020: solved problem, solved since V2.51.0
G172 may cause a deadlock if G25 (tangential transition circle) or G26 (chamfer) is active

ID#437515: Information valid since V2.51.0
G26 (chamfer) only between linear blocks allowed
G26 (chamfer) is not allowed in block transitions with circular blocks. In this case no transition element is created and warning 10141 arises.

ID#400186881: solved problem, solved since V2.51.0
OptMot - synchronization command (like G170, G172 ...) could be accidentally exposed

ID#400152323: solved problem, solved since V2.51.0
Additional status variables in cnc_obj.restart.status show when valid restart data are available or has been saved

ID#400150026: new function since V2.51.0
OptMot - position jump during tracking section in special combination with frame-operations (G92, G192,..)
It could happen that in tracking sections due to a special combination of frame operations (G92, G192,..) and path synchronization commands (e.g. G172,..) a position jump occurs.

ID#400150026: new function since V2.51.0
Declaration of FUB instances using XML
FUB instances can be declared also using the XML configuration files (see the example in gmcipfub.cnc).

ID#400151155: solved problem, solved since V2.51.0
More than two non-synchronous M functions immediately after one another
If the same non-synchronous M function is set more than two times consecutively, the NC program will be aborted with error 7127.
The program is not aborted when between the blocks with non-synchronous M functions other blocks which are processed by the path generator (e.g. moves, dwell time) are programmed.

ID#400151137: new function since V2.51.0
Tool Data Number with Index=0
With the tool data number D00 a selected tool data record is deselected. If a tool data record with index 0 in the table exists, it is ignored when activating and the warning 10133: "Tool with index=0 in tool data table defined" arises.

ID#421920: solved problem, solved since V2.51.0
Discontinuities in path speed profile at block transition when OptMot is active.

ID#400135535: new function since V2.51.0
Number of records for expanded zero point offset (G159) increased to 500
With G159 zero point records can now be used with the index 0 up to 499.
Warning 10134 arises if a zero point offset table is loaded where for G53 (index 0) an offset unequal to zero is defined.

ID#400128666: solved problem, solved since V2.51.0
ncational(ncPOSMON, ncSWITCH_ON) affects the system variable $AA_IW

ID#400123917: new function since V2.51.0
All system variables are both in the default language (B&R G-code) and in the alternative language 1 available

ID#400111983: solved problem, solved since V2.51.0
G25 (tangential transition circles): Movement normal to active mainplane is not correct
Error 7236: "Contour segment skipped, contour speed too high" splitted into error 7236 (forward direction) and error 7251 (backward direction).

It could happen that due to adverse limit configuration a stop point was generated, which was not the desired position of the program, and the computation ended because of this stop point.

G25 / G26 in consecutive programmed blocks
G25 (tangential transition circles) and G26 (chamfer) can be programmed in consecutive blocks.

Status "active" and status "complete" is not handled in correct way for some data module operations in ARNC0.

Zero point table: ncZERO_TAB, ncSAVE / ncSET
Tool table: ncTOOL_TAB, ncSAVE / ncSET
R-Parameter table: ncR_PARTAB, ncSAVE / ncSET
M-Function group table: ncM_GRP_TAB, ncSAVE / ncSET

Technology Guarding of CNC channels with robots
TG licenses 3D base and 3D standard are identified correctly for CNC channels with configured full transformations (TRF_LIB 2.12.0 or newer required).

MoveC command was extended with the optional parameter 'Angle'. If the parameter is programmed the end point and circle point are used for a determination of the circle (plane, orientation) and the new end point is computed out of the programmed rotation angle. If angle is greater than 360 degrees more than one full rotation is performed on the circle.

Joint additive elements joint_add_el[] were introduced. They can be used analogously to existing tcp_add_el[].

Cubic Bezier splines
Cubic Bezier spline interpolation type (G807, G806) was introduced for OptMot. Bezier points B1 and B2 can be defined via BP_1 and BP_2. Feedrate is considered (G807), or neglected (G806).

Deadlock if backward move on the path and there are short blocks at begin of the CNC program

External tool
External tool was introduced (Trf2 version V2.12.0 or higher needed). Its components (dx, dy, dz, phi, theta, psi) can be set from an NC program via EXTERNAL_D_FULL (similar syntax as D_FULL). The tool can be activated/deactivated using EXTERNAL_D_FULL_ON, EXTERNAL_D_FULL_OFF. For more detailed information about the external tool functionality and mechanical systems which support it, see TRF_LIB documentation.

Wrong content of $AX_T_JOLT[<axisIdx>]
Switch off RTCP (G196) without switching on (G195)

Using only switching off the RTCP (G196) could cause service mode when transformation variable does not exist or full transformation is active.

ID#320972 : solved problem, solved since V2.50.0

Count of PVMacro indexes wasn't checked properly

NC Software - ARNC0 V2.49.3

ID#431935 : new function since V2.49.3

G25 (tangential transition arcs) also for circular blocks available

ID#431915 : solved problem, solved since V2.49.3

Wrong axes are used for moves in the active main plane

It can happen that wrong axes are used for moves in the active main plane if there are only two ncCNC axes defined and G18 or G19 is active.

ID#431520 : solved problem, solved since V2.49.3

Wrong position when restarting with byte offset or block number

If the restart point was defined as byte offset or block number, wrong position could be set in case of restarting CNC program which contained subprograms.

NC Software - ARNC0 V2.49.2

ID#400152655 : solved problem, solved since V2.49.2

Parameter axes.axes[].type=ncNOFEED may not be taken in consideration in circular blocks

ID#428765 : new function since V2.49.2

The calculation of the program run time in the plot buffer and CNC simulation mode has been improved.

ID#428310 : solved problem, solved since V2.49.2

Estimated runtime in plot buffer is not correct

The error appears mainly in CNC programs with many tangential block transitions without reduction of path velocity.

ID#428190 : solved problem, solved since V2.49.2

Error 15314 when unloading a NC program

The status cnt ncprog was cleared too early while the CNC system was still busy. It could caused that next NC action was refused with an error (e.g. 15314: "Invalid operation while program is running" on NC program unload).

ID#425295 : new function since V2.49.2

Starting ncFILE_XL programs with line-offset

NC programs can be started in mode ncFILE_XL with a line-offset using <file name>::<line number>. In comparison to ncFILE start mode the parsing of previous lines is skipped. Starting from subprogram is not supported.

NC Software - ARNC0 V2.49.1

ID#427980 : solved problem, solved since V2.49.1

OptMot - removed changes from V2.49.0

Changes from V2.49.0 removed because of unexpected behavior: OptMot - too low limits in the axes init files (and ACP10 Fub) could limit the movement

NC Software - ARNC0 V2.49.0

ID#427275 : solved problem, solved since V2.49.0

Error 15308 after CDC activation

Combination of CDC activation, zero path-distance block and path-synchronous assignment caused error 15308: "Motion data invalid (old)".

ID#427245 : solved problem, solved since V2.49.0

Possibly not updated parameters of built-in robot

If built-in robot from Trf2 is used, then D_FULL, frame axes, forced mode, maximal change and coupling might not work after ncGLOBAL, ncINIT. A situation can change after next ncGLOBAL, ncINIT, and this problem was in ARNC0 since version 2.47.2.

ID#426665 : solved problem, solved since V2.49.0

Programmed circular block will be skipped if TrigHelp is active
Move block may be skipped if TrigHelp is active and a circular block is programmed immediately after a coordinate system transformation, a tool data number (Dxx) or G16.

ID#425840 : solved problem, solved since V2.49.0
If G93 is active, error 8102 may arise in move blocks with very short move distance

ID#425700 : new function since V2.49.0
G25/G26 transition elements are skipped if tool radius too large
If G25 or G26 is active and the CDC causes contour violations (inside corner and tool radius too large), G25/G26 will be disabled for these block transitions (warning 10410 arises).

ID#424595 : new function since V2.49.0
Type auto-discovery feature available also in IEC/ST programs
New parser option AllowVarTypeAutoDiscovery for activating auto-discovery of unknown types (available in IEC/ST for variables stored at PLC).

ID#423480 : solved problem, solved since V2.49.0
OptMot - too low limits in the axes init files (and ACP10 Fub) could limit the movement

ID#400150361 : solved problem, solved since V2.49.0
Error when workspace monitoring active for mechanical system without TRF_get_joints_pos()
Active workspace monitoring together with Trf2 mechanical system with unavailable TRF_get_joints_pos() resulted into an error. TCP position is checked instead the error now (like if Trf1 was used).

ID#422890 : new function since V2.49.0
GmcIpUserConfig extended with user configuration file for IEC/ST language

ID#421945 : solved problem, solved since V2.49.0
OptMot - velocity jumps at the beginning and at the end of a NC block due to numerical problems

ID#421660 : solved problem, solved since V2.49.0
Some instructions were not supported at lines with single line IF statement

ID#421270 : new function since V2.49.0
Extension of CNC monitor
Due to the extended monitor structure ARNC0NCMON_EXT_typ the axis positions and the path distance are available as LREAL.

ID#420545 : solved problem, solved since V2.49.0
OptMot - ncHALT caused jumps in velocity profile
Calling ncHALT caused jumps in the velocity profile.

ID#420345 : new function since V2.49.0
Additive element for cyclic axis set positions
For each axis, which is used in a CNC channel, a ParID can be defined, the content of this ParId is added on the drive to the cyclically set position of ARNC0. The ParID is connected to the ARNC0-internal used MC_BR_MoveCyclicPositionExt FUB (input AdvancedParameters.AdditiveParID).

ID#400149385 : solved problem, solved since V2.49.0
Non-standard order of axes in CNC init parameter module and using of PLCopen axes can cause wrong consideration of axes parameters
Using the order of axes in CNC init parameter module distinct from ncCNC/ncPOLAR, ncLINEAR/ncROTARY, ncTANGENT together with PLCopen axes may cause wrong consideration of axes parameters (e.g. SW limits, speed limits, acceleration limits, unitfactor, etc.).

ID#416585 : new function since V2.49.0
A new path single step mode ncLINENUMBER
A halt is set after each line including lines with IP synchronous execution.
"move.single_s.mode = ncLINENUMBER"

NC Software - ARNC0 V2.48.2
ID#424445 : new function since V2.48.2
Blocks with move distance zero allowed if TrigHelp (G183) or BackLine (G181) is active.

ID#423100 : solved problem, solved since V2.48.2
A position jump may occur when immediately after a G65 the main plane is changed or a coordinate transformation is programmed.

ID#421215 : solved problem, solved since V2.48.2
CDC - detection inside outside corners
During calculation of the centre point from the programmed radius and from the end point of circular blocks a numerical inaccuracy may occur. In block transitions with 180°, it can happen that the CDC depending on the numerical inaccuracy once an outside corner and the other time an inside corner detects (e.g. TrigHelp on/off).

ID#420845 : new function since V2.48.2
G172/G170 after CDC lead out and multiple blocks with no move in the main plane programmed
If a G172 or G170 is programmed immediately after G40, following on multiple move blocks without a move in the main plane are allowed.

NC Software - ARNC0 V2.48.1
ID#419910 : solved problem, solved since V2.48.1
G193 + accurate hold: feed rate to low
If linear feed mode (G193) is active, the programmed feed rate is not reached in move blocks with an accuracy hold at begin and at the end of the block.

ID#419885 : solved problem, solved since V2.48.1
G102 or non-movement in wrapping/mapping section
G102 or non-movement could cause unexpected movements in the beginning of the wrapping/mapping section if MAPPING_MODE = 1 was used.

ID#419435 : solved problem, solved since V2.48.1
Deadlock when illegal combination with G65
Illegal combination of G65 and G172 inside of one template could cause deadlock of CNC program (instead of an error).

ID#419315 : solved problem, solved since V2.48.1
Error 15308: "Motion data invalid (old)" could occur when Trig help activated

ID#419230 : new function since V2.48.1
With the system variable $LINEAR_FEED_MODE the behavior of G193 (linear feed mode) can be influenced

ID#418435 : new function since V2.48.1
New syntax extension for IEC-ST
New syntax extension for IEC-ST to make access to members of structures at PLC. Each member of a structure can be declared alone by it's own alias.

ID#400145402 : solved problem, solved since V2.48.1
Reloading of CNC program could cause page fault when global variable declared inside of the program with an alias

ID#400139987 : solved problem, solved since V2.48.1
Mapping (G217/G218/G219) - wrong default value of multiplication factor used

ID#400116303 : solved problem, solved since V2.48.1
CAM wrapping (G17/G18/G19 Q) - SW limits of source axis were checked, even if the axis does not move

NC Software - ARNC0 V2.48.0
ID#415215 : solved problem, solved since V2.48.0
Trig Help - Undefined positions at circular-circular block transitions
At tangential circular-circular block transitions and Trig Help (G183) is used, undefined positions (NAN) can occur.

ID#414895 : new function since V2.48.0
New functionality: G138 duplicated programmed points
A duplicate programmed point before and after CDC activation/deactivation forces a G137 like behavior. This behavior is only available if G138 is active and must be enabled with the system variable $CDC_DUPLICATE_POS_PROG.

ID#414310 : solved problem, solved since V2.48.0
Error 1251 to 1259 (Error at writing prior command)
Sporadically, the error 1251 to 1259 (Error at writing prior command) may occur.
ID#413310 : new function since V2.48.0
Mechanical systems with numerical transformations (with Trf2) are supported.

ID#413215 : new function since V2.48.0
G-code can be overloaded with user-defined NC program.
The standard behavior of specified G-Code can be temporarily overloaded (user defined NC program will be called instead of the original G-Code).

ID#412570 : new function since V2.48.0
SG4 target system: Data object type "ncDATOBJ_FILE" for download/upload of data blocks and ACOPOS parameter tables
For the following NC actions it is now possible with "datobj_type=ncDATOBJ_FILE", to use files as data object (so far, only BR modules were possible):
  - ncDATBLOCK, ncDOWNLOAD
  - ncDATBLOCK, ncUPLOAD
  - ncACP_PAR+ncDATBLOCK, ncDOWNLOAD
  - ncACP_PAR+ncDATBLOCK, ncUPLOAD
For this, the substructure "datablock.parameter" in the NC structure of the NC objects with type "ncAXIS" and "ncV_AXIS" was extended by the following components:
datobj_type: Type of data object ("ncDATOBJ_BRMOD" or "ncDATOBJ_FILE")
file_device: Name of the file device (only for type "ncDATOBJ_FILE")

ID#412085 : new function since V2.48.0
CDC deactivation with duplicated programmed points in combination with G170 or G172
If CDC deactivation with duplicated programmed points ($CDC_DUPLICATE_POS_PROG) is used, G170 or G172 may be programmed immediately after G40.

NC Software - ARNC0 V2.47.4
ID#412090 : solved problem, solved since V2.47.4
Loading files using #pragma LOAD could cause usage of invalidated data
Runtime unloading of files caused by #pragma LOAD is now disabled. Loading error aborts this action.

ID#412080 : solved problem, solved since V2.47.4
The cumulative program runtime shown in the plot buffer may be wrong (plot buffer data format 0x02).

ID#412020 : new function since V2.47.4
OnProgEnd - new predefined AIL subprogram
OnProgEnd AIL subprogram (if defined in AILHeader section) is called automatically at the end of NC program.

ID#411935 : solved problem, solved since V2.47.4
OptMot - call of ncHALT in combination with single step mode
When calling ncHALT after continuing from a finished block, the halt was performed in one sample instead of following a stopping ramp that respects the limits.

ID#411670 : solved problem, solved since V2.47.4
Indirect subprogram calling from FILE_XL can cause a deadlock
Combination of main program started in ncFILE_XL mode and global subprograms called indirectly ("CALL" statement or "fcall" AIL instruction) can cause a deadlock.

ID#411610 : new function since V2.47.4
Alternative computation method for axis jolt filter
New G-Codes for switching axis jolt filter computation method have been defined:
G250 <axis name> - standard computation method
G251 <axis name> - accurate computation method
The original (standard) implementation of jolt filter may accumulate numerical errors in rare situations (big filtered values, long-term run). The new method is more CPU demanding.

NC Software - ARNC0 V2.47.3
ID#409955 : new function since V2.47.3
New AIL built-in functions
NC Software - ARNC0 V2.47.2

ID#408300 : solved problem, solved since V2.47.2
Plot buffer, runtime for move blocks (data format 0x02) is not precise

ID#407775 : new function since V2.47.2
"SKIP Function" (Block Skip) accepted also at lines with GOTO
Skip function is generally disabled (ignored) at control block lines (FOR, WHILE, multi-line IF, etc.). GOTO is an exception of this rule.

ID#400143934 : new function since V2.47.2
New variable scope ip_global_at_plc

ID#407565 : new function since V2.47.2
Renaming in ARNC0WSCTRL_typ (arn0add.typ)
Items mlink_index and mlink2_index were renamed to link_index and link2_index, respectively.

ID#407265 : solved problem, solved since V2.47.2
Latch position (G200/G201) has been delayed by "axis_obj->controller.position.t_total" for PLCopen axis.

ID#406670 : solved problem, solved since V2.47.2
OptMot - Consideration of the jolt filter for feedforward
For the computation of the feedforward torques the axes jolt filter (t_axfilter) from the cnc-object were not considered.

ID#406650 : solved problem, solved since V2.47.2
User robots in ARNC0 with TRF_LIB V2.ab.x
User robots are supported since TRF_LIB V2.08.1

ID#40091952 : solved problem, solved since V2.47.2
D_FULL data in CNC monitor
Actual D_FULL is path synchronously published in CNC monitor, array d_full[]. The order of items is dx, dy, dz, phi, theta, psi.

NC Software - ARNC0 V2.47.1

ID#405960 : Information valid since V2.47.1
Included drive operating systems
Library Asp10man with version 2.47.0 to 2.47.9 required.
The drive operating system with the same version as Asp10man is used.

ID#405880 : new function since V2.47.1
Automatic program number
Generation of unique automatic program number can be activated in configuration files.

ID# 400143847 : solved problem, solved since V2.47.1
To use ARNC0 together with ARsim no licensing (Technology Guarding) is required.

ID#405670 : solved problem, solved since V2.47.1
G395 - G399 supported for all robots.
TCP frame and path axes reorientation work for all robots if TRF_LIB V1.16.0 or higher is used.

ID#405210 : new function since V2.47.1
New G code for automatically inserting a chamfer between two linear blocks.

ID#405045 : solved problem, solved since V2.47.1
Trig Help and T- or S-Param can cause error 7128
Writing to Technology parameter (S, T-function) when Trig Help is active (G183) can cause Error 7128: "T-Function is not yet transferred to PLC at third call".

ID#404380 : new function since V2.47.1
Automatic synchronization for "SKIP Function" (Block Skip)
User can define AIL subroutine which takes action before the line marked with block skip specifier "/". The subroutine has to be named "OnBlockSkip" and its calling is inserted into automatically created nc block which precedes lines with the block skip specifier. The OnBlockSkip subroutine is called always regardless settings of block skip operation mode. The AIL subroutine has to be created in <AILHeader> section in language configuration file.

ID#403765 : new function since V2.47.1
G170/G172 immediately after CDC deactivation
In CDC activation/deactivation mode G137 and G138, the interpreter synchronization commands G170 and G172 can be programmed immediately after the CDC deactivation.

ID#403265 : solved problem, solved since V2.47.1
Short block_buffer can cause error 15308: "Motion data invalid (old)"
Error 15308 can occur if a size of buffer for backtracking movement is configured shorter than size of look-ahead buffer.

ID# 400141156, 400141813 : new function since V2.47.1
Latch position (functions G200/G201) hasn't been accurate for PLCopen axis

ID# 400139600, 400142417 : solved problem, solved since V2.47.1
Trigger functions (G200 / G201) didn't work for PLCopen axis
The trigger didn't work if different PLCopen axis has been set as trigger source in "cnc_object.axis.axis[i].trg_source.nc_object_plcopen".

NC Software - ARNC0 V2.47.0
ID#403045 : Information valid since V2.47.0
Included drive operating systems
Library Acp10man with version 2.47.0 to 2.47.9 required.
The drive operating system with the same version as Acp10man is used.

ID#402665 : new function since V2.47.0
Plot Buffer: Expansion of output, new data format 0x02 (with runtime and block length)

ID#402320 : solved problem, solved since V2.47.0
Updating of ARNC0 Monitor structure could get frozen when the File Monitor was active

NC Software - ARNC0 V2.46.0
ID#400350 : solved problem, solved since V2.46.0
OptMot - Implementation of rounding whole lines with G126
The functionality of G126 in combination with OptMot was extended. Now it is possible to round the whole first line or the whole second line using the G - code syntax, as it is used with the common ARNC0. To activate this new functionality it is necessary to set the parameter G126_round_whole_line unequal 0.

ID#400120 : solved problem, solved since V2.46.0
If interpreter single step was turned on, the "decoder.status" hasn't been updated during either RESTART INFO or BLOCKSEARCH.

ID#399935 : Information valid since V2.46.0
Included drive operating systems
Library Acp10man with version 2.46.0 to 2.46.9 required.
The drive operating system with the same version as Acp10man is used.

ID#399695 : solved problem, solved since V2.46.0
Program runtime when RESTART a CNC program
The program runtime in the CNC monitor is counted from program start (previously from the restart point). For program sections, which are simulated (program start to the restart point) the runtime is approximately calculated.

ID#399345 : new function since V2.46.0
G138, CDC deactivation immediately after the activation allowed

ID#398565 : solved problem, solved since V2.46.0
Single step mode, status not correct
In single block mode (ncSTANDARD + ncSBL_CTRL, ncBLOCKNUMBER + ncSBL_CTRL), the status is not displayed correctly when switched off.
ID#397355 : solved problem, solved since V2.46.0
Wrong position when restarting with byte offset
If the restart point was defined as byte offset, wrong position was set in case of restarting CNC program of Linux format (LF as line breaks).

ID#400139160 : solved problem, solved since V2.46.0
Array of Function Blocks was not supported in CNC program

NC Software - ARNC0 V1.46.0

ID#516960 : new function since V1.46.0
Exclusive workspace monitoring
Up to 100 exclusive blocks can be defined. They are shared by all existing CNC channels. No more than one CNC channel can be in any of those blocks at any time (ncHALT, ncSTOP and ncCONTINUE are called automatically if more channels want to enter the same exclusive block).

ID#396170 : new function since V1.46.0
Combination G65 (programmed endpoint shift) and synchronization commands (e.g. G172) allowed
Between a block with G65 (programmed zero offset) and the next movement block synchronization commands (G170, G71, G172 and G173) are allowed.

ID#396055 : new function since V1.46.0
Cyclic workspace monitoring
WS monitoring can be called every sample (WS_CTRL_ON CYCLIC_ON). It includes main workspace control, detection of collisions with protected blocks and protected half-spaces, and self-collision detection.

ID#395995 : solved problem, solved since V1.46.0
Path speed to low in short rapid move blocks
In mode blocktransition = ncAUTO it is possible that short blocks with rapid interpolation are traveled with very low speed.

ID#395650 : solved problem, solved since V1.46.0
Syntax error when using specific preprocessor macros

ID#395440 : solved problem, solved since V1.46.0
Path-synch assignment of system variable $P_EP an $P_EM
Assignment of system variable $P_EP or $P_EM to a path_synch variable caused syntax error (e.g.: MF1 = $P_EP[X]).

ID#395400 : solved problem, solved since V1.46.0
ARNC0 gets stuck after ncation(ncMOVE,ncHALT)
ARNC0 gets stuck when ncation(ncMOVE,ncHALT) is called right after G172 or G60.

ID#394810 : new function since V1.46.0
Addition collision information in workspace monitoring
Data type ARNC0WSCTRL_typ from arnc0add_typ can be used to analyze error from workspace monitoring (address published within the error)

ID#394660 : solved problem, solved since V1.46.0
CycleBurstBG had no effect since ARNC0 V1.44.0

ID#394420 : solved problem, solved since V1.46.0
Use of robot (with Trf2) without dynamic model on optimized section caused "wrong dynamic model" error.

ID#393020 : new function since V1.46.0
Exclusive workspace monitoring
Up to 100 exclusive blocks can be defined. They are shared by all existing CNC channels. No more than one CNC channel can be in any of those blocks at any time (ncHALT, ncSTOP and ncCONTINUE are called automatically if more channels want to enter the same exclusive block).

ID#392175 : new function since V1.46.0
Individual scaling for cartesian axes.
Cartesian axes can be scaled by individual factors independently. The function is only available for linear blocks and coordinate system shifts, but not for circular blocks.
Included drive operating systems

Library Acp10man with version 2.46.0 to 2.46.9 required.

The drive operating system with the same version as Acp10man is used.

IEC-ST syntax: path-synchronous blocks (DO_PATH_SYNCH) accept expressions with structures.

Extension of Plot Buffer

For the representation of the data in the plot buffer, a new data format is available. In addition, the user can write data from the CNC program in the plot buffer.

NC Software - ARNC0 V1.45.3

Included drive operating systems

Library Acp10man with version 2.45.0 to 2.45.9 required.

The drive operating system with the same version as Acp10man is used.

Detailed analysis of a failing TRF_LIB transformation function call from ARNC0 can be now done via a data type ARNC0TRFICODEDATA_typ published in arnc0add.typ.

Monitor of NC program showed only the first 12 characters of subprogram name

Deadlock at backward move on the path

When travelling backward on the path (e.g. negative override) a deadlock could occur at very short block and subsequent stop. The block length must be between 1e-6 and 3e-6.

OptMot - New G-Codes G804/G805 - transition curve

A transition curve movement is programmed using the endpoint and an additional parameter R. The command G804 limits the trajectory independent from the feedrate (rapid movement - considering only the axes limits) in contrary to G805. These functionalities try to create a path with a tangential transition respect to the Cartesian coordinates X, Y and Z to the previous and the next movement without any modification of them. With the additional parameter R a virtual cylinder with hemispheres on the front and back side is generated to limit the movement in space of the transition curve. The command G804/G805 is only available with the usage of OptMot. The transition curve can be programmed with one of the following G-Codes:

- G804 X<x> Y<y> Z<z> R=<r>
- G805 X<x> Y<y> Z<z> R=<r>

For further details see OptMot - documentation "OptMot - Jerk Limited Trajectory Planner”.

TCP additive elements of path axes were not applied correctly if the CNC program got to standstill.

Using of small non-integer factor for modulo function caused a page fault.

Relationship between absolute programmed positions (G90) and target rotary axis position can be influenced by MAPPING_MODE (initial offsets of source and destination axes can be omitted now).

Additional information for command errors is wrong or not complete.
The additional text for errors, which are caused by commands not allowed in current state is not correct or complete, now the command code and a reference to the Automation Studio help is issued. In the Automation Studio help a table with all relevant command codes can be found.

ID#390990 : Information valid since V1.45.2
Included drive operating systems
Library Acp10man with version 2.45.0 to 2.45.9 required.
The drive operating system with the same version as Acp10man is used.

ID#390950 : solved problem, solved since V1.45.2
CNC simulation mode can't be switched off
CNC simulation mode can't be switched off if the previous program was aborted and CNC simulation mode was active.
CNC simulation disable is accepted after at least one CNC program execution ends normally.

ID#389820 : solved problem, solved since V1.45.2
OptMot - CNC monitor data not available
With the usage of OptMot the CNC monitor data pos_ncprog, line_ncprog and v_ncprog are not available.

NC Software - ARNC0 V1.45.1

ID#389110 : solved problem, solved since V1.45.1
OptMot - extension of monitor data
The array OptMotConfig.monitor.v_path[i] provides the velocity profile of the configured path definitions.

ID#386985 : solved problem, solved since V1.45.1
NC program lines containing inline IF statement were triplicated in block monitor
NC program lines containing single line IF statement were triplicated in block monitor and three steps were needed in decoder single step.

ID#400135573 : solved problem, solved since V1.45.1
NC program failed to start with offset in mode ncFILE_XL
NC program start in mode ncFILE_XL could fail with error 15554 when the start position type was set to ncFILEOFFSET + ncSKIP_PARSE and the start position was set to a commented line.

ID#385380 : Information valid since V1.45.1
Included drive operating systems
Library Acp10man with version 2.45.0 to 2.45.9 required.
The drive operating system with the same version as Acp10man is used.

ID#385350 : solved problem, solved since V1.45.1
At initialization of CNC limits (ncLIMITS, ncINIT), the value of "limit.filter_err_trans" was not transferred

ID#384780 : solved problem, solved since V1.45.1
Using of G04 with SingleStep could cause deadlock of NC-program

ID#384665 : new function since V1.45.1
G65 (programmed target position shift) without movement in the active plane
In the first movement block after G65 (Programmed target position shift), it is no longer necessary that the active plane motion is programmed.

ID#400125611 : new function since V1.45.1
Extension of restart buffer for trigger functions G200 and G201
The extension allows to define the maximum number of trigger functions stored in the restart buffer using parameter "restart.parameter.trigger_buffer".

NC Software - ARNC0 V1.45.0

ID#391240 : solved problem, solved since V1.45.0
Additional logger entry for error during installation of cyclic tasks
If the installation of cyclic tasks failed, so far only the following errors were written into the logger:
10560 NC manager error (see "ASCII data"), ASCII data: <TaskName>: Installation
10530 NC Manager: NC SW INIT aborted, ASCII data: ACP10MAN
This problem can usually be corrected by increasing the number of cyclic objects in the AR configuration. For this reason, in this situation the following error is additionally written into the logger before the errors described above:
10560 NC manager error (see "ASCII data"), ASCII data: cypCreateDynamicProg failed (not enough cyclic objects configured?)

ID#384275 : solved problem, solved since V1.45.0
Labels started with L character wasn't supported in B&R G-code

ID#383775 : solved problem, solved since V1.45.0
System variables was reset after NC program started not as expected
Some of system variables (e.g. $LIM_AX_ACC) were reseted to their default values after NC program has started. This may caused overwriting of newly programmed values.

ID#383540 : new function since V1.45.0
Interpreter monitor structure extended
Interpreter monitor was extended with elements cur_prog_name and cur_prog_file_name for interpreter-synchronous monitoring of active CNC programs.

ID#383110 : new function since V1.45.0
Floating-point modulo operator (remainder of division)
Instruction set of the GMC interpreter was extended with floating point operator (token TKG_ARITH_BINARY_FLOATING_MODULO). The floating-point modulo operator is disabled per default, to activate it the appropriate language configuration file (gmciplbr, gmciplsm,...) has to be modified/extended with the new token definition.

ID#382290 : solved problem, solved since V1.45.0
NC program lines get lost when using inline IF statement
When single-line IF statement was programmed at the line which number is divisible by ReadLineBurstMax (default 100) the other lines behind this statement were skipped.

ID#382035 : Information valid since V1.45.0
Included drive operating systems
Library Acp10man with version 2.45.0 to 2.45.9 required.
The drive operating system with the same version as Acp10man is used.

ID#382025 : solved problem, solved since V1.45.0
G65, coordinate system shift added two times
G65 (target position shift), depending on the programming mode (relative - absolute) it could happen that a coordinate shift (e.g. zero point shift with G54) was taken into account twice.

NC Software - ARNC0 V1.44.0

ID#380742 : solved problem, solved since V1.44.0
Path speed at block transition may be too low
By an error in the function for calculating the allowed speed at block transition, it could happen that the speed has reduced too much at block transition.

ID#379410 : new function since V1.44.0
Release of jerk limited trajectory planner called ‘OptMot’
A jerk limited trajectory planner called 'OptMot' is now available.

ID#378657 : Information valid since V1.44.0
Included drive operating systems
Library Acp10man with version 2.44.0 to 2.44.9 required.
The drive operating system with the same version as Acp10man is used.

ID#378000 : solved problem, solved since V1.44.0
Error 15650 occurred, when there was used indirect call many times one after another
When using indirect program call (in Alternate language 1) many times one after another, there could error 15650: "Internal Error - AIL evaluation stack underflow" occur.

ID#377412 : new function since V1.44.0
Context switch for interpreter background task optimized

ID#376115 : solved problem, solved since V1.44.0
Communication with ARNC0 could freeze when calling subprogram using name already used for some folder

The communication with target could freeze, when calling a subprogram using name of any folder. It could happen also when calling a subprogram using directory specification and the folder separator is not defined for actual language.

ID#375877 : new function since V1.44.0
New functionality G183/G184 ("Trig Help")

"Trig Help" allows programming a series of blocks in a simple way. The end point of a block is programmed approximately; the exact intersection point is determined by the system.

NC Software - ARNC0 V1.43.3

ID#374527 : new function since V1.43.3
G65 (programmed target position shift)

In blocks with G65, the prefix IC or AC is allowed for programmed coordinates.

ID#373982 : Information valid since V1.43.3
Included drive operating systems

Library Asp10man with version 2.43.0 to 2.43.9 required.

The drive operating system with the same version as Asp10man is used.

ID#373830 : new function since V1.43.3
Automatic activation of ncFILE_XL+ncBLOCKMONITOR

NC program start mode ncFILE_XL+ncBLOCKMONITOR can be activated automatically depending on size of program file or data object. The limit size can be set in XML configuration file gmcipsys in section <LOAD> by option ForceFileXLfromSize. The size is specified in bytes (eg. ForceFileXLfromSize="2097152").

ID#373787 : solved problem, solved since V1.43.3

G126: Transitions between very short blocks will not be rounded

G126 rounding will be skipped for blocks whose execution time is less than two CNC cycles, this block transitions are treated tangential transitions. In this case, the warning is issued 8274. Due to an error all the short path sections were not detected.

ID#400123939 : solved problem, solved since V1.43.3
Feedback calculation on very short path sections

Under the following conditions, numeric effects can cause discontinuity in the speed profile of auxiliary axes (e.g. ncLINEAR, ncROTARY, ncTANGENT or ncNO_FEED): decoder.parameter.v_path_mode = ncCNC

Very short distance of ncCNC axes (1e-4 CNC units or less)
Auxiliary axes programmed and their distance much longer than the distance of the ncCNC axes (a factor of 1000 or more)

"decoder.parameter.v_path_mode = ncCNC + ncLINEAR" is used for path sections on which the distance of the ncCNC axes is shorter than 1e-4 CNC units and auxiliary axes are programmed. This adjusts the feed that is being used so that the path speed still equals the programmed feed. The CNC Monitor and cyclic trace will show the correct path section (geometric sum of all axes ) for these path sections instead of the programmed path section.

NC Software - ARNC0 V1.43.2

ID#373335 : new function since V1.43.2
Colliding area detection

If extended workspace monitoring is used, then the first colliding area (block or half-space) can be identified via its index.

ID#373295 : Information valid since V1.43.2
Included drive operating systems

Library Asp10man with version 2.43.0 to 2.43.9 required.

The drive operating system with the same version as Asp10man is used.

NC Software - ARNC0 V1.43.1

ID#372472 : Information valid since V1.43.1
Included drive operating systems
Library Acp10man with version 2.43.0 to 2.43.9 required.
The drive operating system with the same version as Acp10man is used.

ID#371752 : new function since V1.43.1
Programmed target position shift for linear blocks (G65): programming with absolute coordinates

ID#371490 : new function since V1.43.1
Optional THEN for IF statements
Optional THEN was supported only for inline IF-statements.

ID#371305 : new function since V1.43.1
Block number support in IEC-ST
Modal settings of block number can be set by following syntax:
#pragma N<number>

ID#371302 : solved problem, solved since V1.43.1
Option IgnoreIfNotExist="Yes" caused a page fault
Option IgnoreIfNotExist="Yes" for the node <CNCAXIS> in Interpreter configuration file caused a page fault since ARNC0 V1.40.0.

ID#371092 : solved problem, solved since V1.43.1
Same behavior if G93 is programmed without the F-word or G94 without F-word
If G93 is programmed without a F word, the last programmed G93 feed rate value will be used and for G94 without F word the last programmed G94 feedrate.
So far, for G93 the feed rate of the last block (G93 or G94) was used, for G94 however the last programmed G94 feed rate has been used.

ID#370732 : solved problem, solved since V1.43.1
Incorrect feed rate if G93 is used in combination with G70/G71

ID#370092 : solved problem, solved since V1.43.1
Path speed at block transition to low
Caused by a restrictive dynamic calculation, it could happen that the path speed at block transition was reduced or even stopped, although the limits of axes acceleration were not reached.

ID#369365 : solved problem, solved since V1.43.1
Path-synch assignment after target position without interpolation type causes an page fault when programmed on the same line

ID#369355 : solved problem, solved since V1.43.1
Syntax error in XML configuration file could cause page fault

ID#368562 : solved problem, solved since V1.43.1
Synchronous action could cause unexpected behavior of NC program when RESTART is switched on
If RESTART is switched on:
1) the first cyclic execution of path synchronous configured synchronous action was omitted
2) interpreter synchronously configured synchronous action could cause unexpected behavior of ARNC0

ID#367152 : solved problem, solved since V1.43.1
First line of subprogram was commented out in motion packet log

ID#362890 : new function since V1.43.1
Limitation of contour violation at block transition caused by axis jolt filter
The allowed contour violation can be programmed with the parameters limit.filter.err_trans and in the CNC program with the system variable $FILTER_ERR_TRANS.

NC Software - ARNC0 V1.43.0

ID#368902 : Information valid since V1.43.0
Included drive operating systems
Library Acp10man with version 2.43.0 to 2.43.9 required.
The drive operating system with the same version as Acp10man is used.

ID#400098000 : solved problem, solved since V1.43.0
Deadlock when switching on the controller at the same time when an ACOPOS error occurs

An error on the ACOPOS (e.g. jitter at the Quick-Stop input) immediately after controller on can cause a deadlock. No further commands will be accepted (Error 14155: "internal command buffer full"). The error occurs only with ARNC0 axes.

ID#367835 : solved problem, solved since V1.43.0
Error 15798: "All jobs active in block_buffer" occurred when program started with synchronous actions

ID#367505 : solved problem, solved since V1.43.0
Page fault at ARNC0 startup (only in V1.37.0 - V1.42.x)
This problem occurs for example, if there is a NC INIT parameter object entered in the NC mapping table, which is not on the target.

ID#365562 : new function since V1.43.0
Display of Warnings (CNC system) can be configured individually

ID#400112670 : solved problem, solved since V1.43.0
Deadlock at short blocks
When traveling with high path speed, it may occur a deadlock at very short blocks with subsequent halt (accurate stop, synchronous M functions, etc.).

ID#364745 : new function since V1.43.0
Syntax for global subprogram call was extended to support full path name
Full path name (including file device) can be used in name of called subprogram. In this case the name must be preceded with file prefix (^, #).

ID#364740 : new function since V1.43.0
Relative path specification for global subprogram defined
The global subprograms can be called using relative path specification. The prefix \ for B&R language or ./ for Alternate language will be substituted by full path of the caller (program).

ID#364195 : solved problem, solved since V1.43.0
Start of NC program with specified subprogram call sequence fails with error 15805 if program begins with label.

ID#363305 : new function since V1.43.0
Setup for induction and synchronous motors: New parameter
The below listed new parameter is offered in following substructures:
setup.motor_induction.parameter
setup.motor_synchron.parameter

\t_tripping_therm: Tripping time at thermal overload [sec]
If "parameter.optional.phase_cross_sect = 0" is specified, then the line cross section of the motor is calculated as follows:
MOTOR_WIND_CROSS_SECT = MOTOR_CURR_RATED * sqrt(t_tripping_therm/360.6)
If for "parameter.optional.phase_cross_sect" a value other than zero is specified, then the line cross section of the motor is initialized with this value:
MOTOR_WIND_CROSS_SECT = phase_cross_sect

ID#360995 : new function since V1.43.0
Extension of working area in workspace monitoring
Working area defined as a cylinder (WS_MAIN_DEF with CYLINDER_IN_Z) has new possibility. If CYLINDER2_IN_Z is used instead of CYLINDER_IN_Z, then a cylinder equipped with lower hemisphere is considered.

ID#400123055 : solved problem, solved since V1.43.0
Error 15650 when calling a subprogram together with other NC command
"Internal Error - All evaluation stack underflow" occurred when an instruction for subprogram call was not programed alone.

ID#357685 : new function since V1.43.0
Searching for global subprogram in local path of caller program
List of searched paths in XML configuration can be extended with <MAINSEARCHPATH Name="@local" /> and <INCLUDEPATH Name="@local" /> settings to enable searching for global subprogram in local path of caller program.

ID#346825 : solved problem, solved since V1.43.0
ST-instruction EXEC not properly passed the arguments when an NC program was reloaded.

ID#346765 : solved problem, solved since V1.43.0
Page fault when starting reloaded IEC programs with user-defined global types
ID#400112488 : solved problem, solved since V1.43.0
G70/G71 for axes of ncTANGENT type
G70/G71 (unit conversion metric/imperial): Axes of ncTANGENT type will also be converted.

NC Software - ARNC0 V1.42.1
ID#363512 : Information valid since V1.42.1
Included drive operating systems
Library Asp10man with version 2.42.0 to 2.42.9 required.
The drive operating system with the same version as Acp10man is used.

ID#363390 : solved problem, solved since V1.42.1
Page fault if number of elements in block buffer for back tracing CNC-path configured low
Page fault occurred if number of elements in block buffer was configured lesser than default and first command in NC program was non-movement path command (M-function, path-synchronous assignment, ...).

ID#361972 : solved problem, solved since V1.42.1
Page fault if G222 is active
A page fault may occur, if G222 is active and the desired remaining distance is not sufficient (warning 7177 arises).

ID#400123054 : solved problem, solved since V1.42.1
Limited range of automatic tangential axis offset
Range of automatic tangential axis offset was only 0 - 360.

ID#400060837 : solved problem, solved since V1.42.1
Page fault on ARNC0 axes
In very rare cases, a page fault may occur when an axis movement is aborted or the controller is switched on. This affects only ARNC0 axes.

ID#357750 : new function since V1.42.1
Inline IF statement support
Syntax of IF statement was extended, inline IF statement is newly supported.
This extension is valid for G-Code and Alternative language.
Syntax:
$IF <condition> [$THEN] [$IF <condition>... ] <commands>
Example:
$IF R1==2 $THEN G0 X10
is equivalent to:
$IF R1==2 (If the value from R1 is equal to 2,)
G01 X10 (the X axis is moved to position 10)
$ENDIF

ID#355955 : solved problem, solved since V1.42.1
Declaration of dynamic PVs
1. Error has been fixed in declaration of dynamic PVs in IEC-ST program.
2. Declaration of dynamic PVs is supported in Interpreter configuration file "gmcipvar" by specifying Ref="Yes"
Syntax:
<PV Alias="alias_name" Type="type_name" Ref="Yes">task_name:pv_name</PV>

NC Software - ARNC0 V1.42.0
ID#358000 : new function since V1.42.0
Extension of workspace monitoring
Main working area can be defined as a cylinder inscribed into the cuboid (base of the cylinder in xy rectangle of the cuboid, centralized in it, and with maximal possible radius and height).
Links of zero length are not considered in self-collision detection. Two links are controlled for a collision only if there is at least one nonzero link between them, independently from their defined diameters (see ARM_DIAMETER, G320.4).
Wire-frame model of the robot can consist of more 3D points now (24 instead of 10 points). Therefore, e.g. wire-frame model of tripod (type B, TRF_DATA13_typ) from transformation library V>=1.23.0 can be used with ARNC0 V>=1.42.0.
Optional stop mode (stop at M01) can be switched on or off even if a CNC program is active.

Skip function operating mode can be switched on or off during active CNC program.

Included drive operating systems
Library Acp10man with version 2.42.0 to 2.42.9 required.
The drive operating system with the same version as Acp10man is used.

ST Motion Extension - wrong scope for default values and temporary variables
Default values and temporary variables was shared among CNC channels.

Use of only fractional part of number for feedrate without space
It was not possible to use a literal of fraction type after the F literal (for feedrate) without a space. (eg: F.1)

G102 with target point in joint space
If full transformation is active in CNC channel, then target point of generally oriented circle (G102) can be defined in joint space.

New option for parser
A new parser option 'AllowConditionsWithAssignEq' was added to allow or deny the use of "assign to" inside a condition with meaning of "equal to". Default: denied.

SET/GET methods for controlling CNC parameter access
It is possible to define get/set methods for controlling CNC parameter access.

New system variable $SIM_S_NCPROG
The system variable $SIM_S_NCPROG provide an access to current path position during run in simulation mode (ncSIMULATION, ncSWITCH_ON).

G211 (blended move mode): Path speed too high
Entrance speed at first block after G211 activation or after a stand still may be too high.

Blended move mode (G211) is not taken into consideration for run time calculation in CNC simulation mode.

Limited range of program number
Range of program number (nr_ncprog) was only 2^31-1 instead of 2^32-1.

G181/G182: Back Line
The "back-line" function allows an easy way to program linear-linear or circular-linear block combinations. The endpoint is programmed and the intersection point will be automatically calculated.

ARNC0 error text: Number of additional error text for all errors limited to maximum 3
To start a CNC program in IEC-ST syntax the program name must end with '_iec' suffix.

**NC Software - ARNC0 V1.41.2**

ID#356337 : solved problem, solved since V1.41.2

Error 1114 and 1120 may occur sporadically.

ID#356247 : Information valid since V1.41.2

Included drive operating systems

Library Acp10man with version 2.41.0 to 2.41.9 required.

The drive operating system with the same version as Acp10man is used.

ID#356237 : solved problem, solved since V1.41.2

Page fault in CNC simulation mode (only V1.41.0 and V1.41.1)

Page fault at program start may occur if CNC simulation mode is active.

**NC Software - ARNC0 V1.41.1**

ID#356157 : Information valid since V1.41.1

Included drive operating systems

Library Acp10man with version 2.41.0 to 2.41.9 required.

The drive operating system with the same version as Acp10man is used.

ID#400120757 : solved problem, solved since V1.41.1

"NC SW INIT aborted" by ARNCOMAN because of "NC object already in NC mapping table" (only in V1.37.0 - V1.41.0)

Under the following conditions it could happen, that the startup of ARNCOMAN was falsely aborted with the logger errors "NC SW INIT aborted" and "NC object already in NC mapping table":
- Two NC objects were defined with the same name length
- These NC objects have the same NC object type
- These NC objects have the same channel number
- These NC objects belong to an NC module with the same node number
- These NC objects belong to the same network interface type (e.g. ncPOWERLINK_IF, ncVIRTUAL_IF)
- These NC objects do not belong to the same network interface

With these logger errors the startup should only then be aborted, if the NC objects belong to the same network interface, and the following parameters are also equal:
- NC object type
- Channel number
- Node number

**NC Software - ARNC0 V1.41.0**

ID#349957 : Information valid since V1.41.0

Included drive operating systems

Library Acp10man with version 2.41.0 to 2.41.9 required.

The drive operating system with the same version as Acp10man is used.

ID#349627 : solved problem, solved since V1.41.0

Error 7742: "ACP10_MC FB error or aborted" splitted

If an error message contains more than three additional information, the determination the error text in the application task provides the message "buffer too small for all text" instead of the error text output.

For this reason, error 7742 with four additional information is splitted:
- Error 7742: "ACP10_MC FB Error" if a FUB error arises
- Error 7746: "ACP10_MC FB aborted by an other command" if a FUB execution will be aborted by a command

ID#349490 : new function since V1.41.0

ST Motion Extensions - TRF_LIB V2.00.2 and higher supported

ID#400117553 : solved problem, solved since V1.41.0

Negative path velocity if short path sections are skipped over.

If path sections are skipped over when traversing a path section transition at high velocity, numeric effects can result in a negative path velocity.

The path velocity is normally limited in the dynamic calculator to the point where at least one scan is included. However, there are cases in which movement is still faster (e.g. v_jump>0) and short path sections may be skipped over.

ID#347635 : new function since V1.41.0
Workspace monitoring without self-collision detection

Workspace monitoring can be activated without self-collision detection (modifier SELF_COLLISION_OFF of WS_CTRL_ON).

**ID#347237 : new function since V1.41.0**

G65: CDC activation and deactivation with auxiliary line and circle

G65 provides an easy CDC activation and deactivation on inner corners.

**ID#346725 : new function since V1.41.0**

Suppress path single step halt on current NC block

New command NOHALT was introduced to suppress path single step for current NC block.

Syntax:

```
[NOHALT] [N<number>] [<NC Commands>]
```

Description:

- NOHALT suppresses halting on the NC block where was programmed
- NOHALT takes effect only in ncSTANDARD + ncSBL_CONTROL and ncBLOCKNUMBER + ncSBL_CONTROL modes
- NOHALT command is valid only on the same NC block where was programmed

**ID#346240 : new function since V1.41.0**

G102 with rotation angle

G102 can be programmed with rotation angle of the circle. If the angle is defined, the programmed end point and the circle point are used for a determination of the circle (its plane and orientation) and the new end point is then computed out of the programmed rotation angle. If angle is greater than 360 degrees, more than one full rotation is done.

**ID#345990 : solved problem, solved since V1.41.0**

Syntax error in macro expansion

If a macro expression was placed behind a G-Code on the same line (in the same NC block), a syntax error occurred.

**ID#400115203 : new function since V1.41.0**

New system variable $V_PATH_MODE

The system variable $V_PATH_MODE allows to program in the CNC program independently for each block the mode for calculation of the path feed (ncCNC or ncCNC+ncLINEAR).

**ID#340820 : new function since V1.41.0**

Programmed target position shift for linear blocks (G65)

The command G65 allows to shift the target position of linear blocks in the CNC program.

**ID#400114300 : solved problem, solved since V1.41.0**

ARNC0: 1115: "Error at writing prior command"

When writing prior commands (i.e. commands which are also allowed during active CNC program, such as aborting a CNC program), it may happen that error 1115: "Error at writing prior command" occurs. Additionally error 1120 "Error with command distribution" may occur.

**NC Software - ARNC0 V1.40.1**

**ID#345520 : Information valid since V1.40.1**

Included drive operating systems

Library Acp10man with version 2.40.0 to 2.40.9 required.

The drive operating system with the same version as Acp10man is used.

**ID#344230 : new function since V1.40.1**

ST Motion Extensions - position latch supported

MoveLSearch command latches positions after trigger event during linear movement.

**ID#343952 : solved problem, solved since V1.40.1**

Not smooth speed profile in G126 spline

If there is a stop directly after G126 spline the speed can unexpectedly decrease. This can happen when programming G126 0 together with a movement and the whole movement is consumed with that spline. Decreasing of a speed is eliminated if EXTENDED_ROUNDING is used before an activation of G126.

**ID#342035 : solved problem, solved since V1.40.1**

Multiplied coupling coefficients
If a coupling on axes is defined (TRF_LIB 2.xx), then unexpected offsets could occur on axes within a start of NC program.

**NC Software - ARNC0 V1.40.0**

**ID#340790 : Information valid since V1.40.0**

Included drive operating systems

Library Acp10man with version 2.40.0 to 2.40.9 required.

The drive operating system with the same version as Acp10man is used.

**ID#338625 : solved problem, solved since V1.40.0**

Conversion functions MXFromFrame and MXToFrame cleared other fields of MXPointType than only expected ‘Pos’ and ‘Orient’.

**ID#336965 : solved problem, solved since V1.40.0**

Out-of-Memory when calling user-defined G-function with many arguments

Loading NC program containing call of user-defined G-function with many arguments (problem was occurring from 16 up) caused following two errors:

1. 15245: Out of Memory
2. 15649: Internal Error - AIL Generator line opcode buffer full

**ID#336845 : new function since V1.40.0**

ST Motion Extensions - Strong type check for motion command arguments

Data type of motion command argument is now checked when NC program is being loaded, not when NC program is running.

**ID#336515 : solved problem, solved since V1.40.0**

Unstable behaviour of NC program related to end of file

Program run was unstable (occasional pagefault or $GOTO was misbehaving) if the program's source code was ended with an empty line just after every 100th line.

**ID#335820 : new function since V1.40.0**

ST Motion Extensions - Motion packet log supported

**ID#335715 : new function since V1.40.0**

ST Motion Extensions - Interface change for MXJointToPointEx and MXPointToJointEx

Data type of conversion function arguments for reference values were changed.

**ID#400106846 : solved problem, solved since V1.40.0**

Path speed to high on circular blocks

Due to an error in the dynamics calculation, it could happen that circle blocks were traveled too fast. At block transitions, the path speed was reduced to the correct value.

**ID#333745 : new function since V1.40.0**

New mode to allow/suppress path single step mode in CNC program

New mode for path single step mode: ncSTANDARD+ncSBL_CONTROL and ncBLOCKNUMBER+ncSBL_CONTROL.

The commands SBLOF_PATH(G720)/SBLON_PATH(G721) allow to define sections in the CNC program where path single step mode is suppressed.

**ID#333707 : new function since V1.40.0**

Setup for induction and synchronous motors: New optional parameters

The below listed new parameters are offered in following sub structures:

setup.motor_induction.parameter.optional
setup.motor_synchron.parameter.optional

**phase:** Motor phase (1,2,3)

**invcl_a1:** Inverter characteristic: Gain factor

**invcl_a2:** Inverter characteristic: Exponent [1/A]

**ID#333702 : new function since V1.40.0**

Setup for controller: New parameter "kv_percent"

The Autotuning function first determines the basic values for the kv parameters of speed and position controller. The percentage defined by "kv_percent" (50..150%) is then multiplied to these values in order to calculate the final controller parameters.

"kv_percent=0" corresponds to 100%.
A value less than 100% increases the robustness of the controller with regard to parameter variations on the machine.

Both ACP10 axes and ARNC0 axes can be used as external axis object for axis limit switch (G107) in the CNC program. If ACP10 axes are used, the parameter "limits.paramater.sw_end_enable" is not available.

ACP10 axes can be used for G107

The drive operating system with the same version as Acp10man is used.

Spindle slope compensation ncCYCLIC caused cyclic time violation

Switching the controller off can cause cyclic time violation while spindle slope compensation ncCYCLIC is on.

Non-movements in the middle of G126 spline

If G170 is programmed between two movements inside a range of active G126 area, and if there are also non-movements programmed between those two movements, then those non-movements would be executed in the middle of the rounding spline by default. If G170 is equipped with a modifier FUNC_BEFORE_ROUNDING (non-modal modifier), then those non-movements are executed before the spline.

There is an unexpected movement if axis mapping (G217/G218/G219) is used more than once.

DPR-Trace: At the command "S5 Initialize axis settings (1)" random values are displayed for rot_period and rot_offset

At CDC activation with G137 in areas with short blocks, it is possible that a contour violation will not be detected. In this case with active G38, the CNC program will not be aborted, with active G39 an unexpected long movement will be travelled.

A meaning of orientation in path axes for user mechanical systems
Fourth, fifth and sixth path axes of the user mechanical system can be combined to form an orientation of the robot. Then G-codes dealing with orientation (G818, G819, G395-G399, etc.) behave the same as it was standard 6ax robot. To enable that, the mechanical description (TRF_DATA00_typ:::mech_ place holder) must start with two special UDINT values. The first one is the accessing code 81881906 (which evokes that G818 and G819 behave like for 6ax robot). The second UDINT in TRF_DATA00_typ:::mech_ is the angles' type which is used to compose 4th, 5th and 6th path axes into the rotation matrix (or vice versa, to decompose the matrix into proper Cardan/Euler angles).

ID#33477 : solved problem, solved since V1.39.0
CDC: Unexpected movement on inner corners if tool radius will be changed online.
If CDC is active and the tool radius will be changed ($RAD), it is possible that at inner corners a contour violation will not be detected. In this case, with active G38, the CNC program will not be aborted, with active G39 an unexpected long movement will be travelled.

ID#33142 : Information valid since V1.39.0
Included drive operating systems
Library Acp10man with version 2.39.0 to 2.39.9 required.
The drive operating system with the same version as Acp10man is used.

ID#332750 : new function since V1.39.0
Segmentation in path planner for G102
Segmentation in path planner (SEG=<n>) was extended from lines to the circle interpolation in general orientation (G102). If such a circle is programmed with its target position in path axes, then the segmentation considers the real circular path, and not only a connection between its start and end point. Joint coordinates for the target position of the circle are correct then, and e.g. a point to point movement can follow.

ID#332060 : new function since V1.39.0
Start NC program with specified subprogram call sequence
When starting NC program by NC Action ncPROGRAM, ncSTART, the sequence of NC-block/line numbers can be specified in name of main program (move.ncprogram.name). This sequence (up to last element) then specifies position of subprogram which should be entered when NC program is started.
Syntax:
<program_name>[:{:N<number>|:<number>}...]

ID#331837 : solved problem, solved since V1.39.0
Automatic deactivation of jolt filter on drive for CNC movements
An axis jolt filter can be defined on both the ARNC0 and the drive. The dynamic calculation takes in consideration only the axis jolt filter defined in ARNC0. With the element <cnc_obj>.axis.axis[].drive_axfilter the user can choice if the filter on the drive is automatically disabled for CNC movements. By default both jolt filter are active during CNC movements.

ID#331235 : new function since V1.39.0
TRF_LIB 2.xx supported in ARNC0
TRF_LIB versions 2.xx can now be used together with ARNC0. Oldre transformation library (versions 1.xx) can be used without any change as well.

ID#330762 : solved problem, solved since V1.39.0
Block transitions not rounded even though G126 is active
With active G126, it is possible that block transitions will not be rounded. Conditions: “decoder.v_path_mode = ncSTANDARD” and axes of type ncLINEAR are involved in the movement. Whether a transition will be rounded depends on how much the linear axes are involved in the path speed.

ID#330495 : new function since V1.39.0
G153 in Workspace monitoring
G153 (Position coordinates in the machine coordinate system) can be used on NC blocks with definitions of protected areas (PROTBLOCK_DEF and PROTHSPACE_DEF) as well as on NC blocks with a definition of workspace boundaries (WS_MAIN_DEF). G153 is not propagated to the next NC blocks.

ID#330080 : solved problem, solved since V1.39.0
Path synchronous jobs in simulation mode
Simulation run of programs containing path-synchronous operations: Jobs (path-synchronous variable assignment, function calls) can be terminated with error 15798: “Insufficient Path Synch jobs configured - all jobs active in block_buffer “.

ID#400075752 : solved problem, solved since V1.39.0
NC progam start mode ncFILE_XL+ncBLOCKMONITOR not working in simulation mode
Run of NC program started in ncFILE_XL+ncBLOCKMONITOR mode terminated with error 15245: “Out of Memory”
NC Software - ARNC0 V1.38.1

ID#333007 : solved problem, solved since V1.38.1
NC manager error (40xxx) during PLC startup could cause a page fault (only in V1.37.0 - V1.38.0)

ID#332937 : solved problem, solved since V1.38.1
Deadlock after aborting a CNC program
A deadlock can occur if a CNC program will be aborted immediately after start and the system load on target is very high.

ID#332752 : Information valid since V1.38.1
Included drive operating systems
Library Acp10man with version 2.38.0 to 2.38.9 required.
The drive operating system with the same version as Acp10man is used.

ID#400103374 : solved problem, solved since V1.38.1
Sporadic page fault if controller will be switched on
If the controller will be switched on after a movement abort (e.g.; basic movement, CNC program), in very rare cases a page fault can occur.

ID#400078325 : new function since V1.38.1
ARNC0 axes: An additive element for cam coupling will be immediately taken in consideration.
ARNC0 axes: With ncaction(ncCAM_PROF, ncSET) the additive element will be taken in consideration immediately. The return value of the NC action is either ncOK or an error number (but never ncACTIVE).

NC Software - ARNC0 V1.38.0

ID#329560 : Information valid since V1.38.0
Included drive operating systems
Library Acp10man with version 2.38.0 to 2.38.9 required.
The drive operating system with the same version as Acp10man is used.

ID#325035 : new function since V1.38.0
Parameter FORCE_FLUSH for NC_BLOCK_SKIP_ON
If full kinematic transformations are switched on, if NC_BLOCK_SKIP_ON is active and equipped with a parameter FORCE_FLUSH, and if a call of the transformation (direct or inverse one) is not successful in path planner, then G171 is generated automatically (and the corresponding movement NC block is skipped as in a case of NC_BLOCK_SKIP_ON without FORCE_FLUSH).

ID#324175 : solved problem, solved since V1.38.0
Interpreter axes system variable not updated
Values of Interpreter axes system variables as $MA_POS_LIMIT_PLUS (pos_sw_end) are not updated if its value is changed (initial value from system start-up remains). The error regards only axes defined in a ACP10 mapping table with CNC_Enabled=“1”.

ID#322580 : new function since V1.38.0
Motion Extensions of IEC-ST Interpreter
New system of motion commands, conversion functions and data structures has been designed for controlling any kind of robots or CNC systems. Motion Extensions are activated by inserting new arnc0mx library to Automation Studio project. The ARNC0 will automatically search for the Motion Extensions and then adopt them to its instruction set. Note: Motion Extensions require presence of TRF_LIB from version V1.16.0 in Automation Studio project and Automation Runtime from version V3.08.

ID#322575 : new function since V1.38.0
IEC-ST supported in the GMC interpreter
The GMC interpreter has been extended to offer possibility to process IEC-ST files. The GMC interpreter supports the syntax used by the Automation Studio IEC-ST compiler, plus extensions related to motion programming (Motion Extensions) and memory organization between CNC channels (e.g. scope definition of variables used in the CNC system).
Note: While IEC-ST is fully supported in ARNC0 from V1.38.0, the IEC-ST Motion Extensions require presence of arnc0mx library in Automation Studio project.

ID#320565 : new function since V1.38.0
Long name for CNC program
The allowed length of the name of CNC programs and CNC ini programs was increased (99 significant characters + string terminator).
G193 - wrong path speed profile at block transition with stand still.

At block transition with not programmed stand still (e.g. reversing of an axis movement) it can happen that a wrong path speed profile (feed rate) will be travelled.

NC action ncPROGRAM, ncLOAD was not properly refused in forbidden states

Wrong determination of restart position after restart

After restarting an NC program from positions where the interpreter execution is stopped (like G170 or synchronous M-function followed by G172) the recording of any value to the restart buffer was turned off even if the restart is switched on. In case of restarting the NC program from a position behind the previous restart point the live instead of recorded values were used for determining the restart position.

G193 (linear feed rate) at block transition with accuracy hold

The feed rate will be continusly ramped down to zero before block transition with accuracy hold (G60, synchronous M-function). In the subsequent block the feed rate will be linearly increased to the programmed value.

NC Software - ARNC0 V1.37.3

Included drive operating systems

Library Acp10man with version 2.37.0 to 2.37.9 required.

The drive operating system with the same version as Acp10man is used.

ARNCO axes: Error 107036 when initializing an ACOPOS device

When using ARNC0 axes on a POWERLINK network, during ACOPOS initialization the actual position is requested for the monitor data with "CYCLIC_MON_PARID=111". If an encoder is not connected (e.g. when using an ACOPOSmulti power supply module), then the ACOPOS module will report error 7036: "Encoder: Interface ID invalid", which in turn results in ARNC0 error 107036.

Remedy:
If no encoder should be used for an axis, it is possible to disable the request of actual position for the monitor data by entering AxisConfig="NoActualPosition" in the "Additional data" column for this axis in the ARNC0 mapping table.

The error occurs only when using ACP10 software V2.37.0 - V2.38.0.

NC Software - ARNC0 V1.37.2

Included drive operating systems

Library Acp10man with version 2.37.0 to 2.37.9 required.

The drive operating system with the same version as Acp10man is used.

Error 7185 at restart or block search if ncROTARY axes are used.

Error 7185: "Position modulo out of 'In-Position-Tolerance' at RESTART of CNC move" may occur, when the restart position is located in the immediate vicinity of an interval boundary. If the error arises depends on the numerical values of the positions and parameters.

NC Software - ARNC0 V1.37.1

Included drive operating systems

Library Acp10man with version 2.37.0 to 2.37.9 required.

The drive operating system with the same version as Acp10man is used.

NC Software - ARNC0 V1.37.0

The maximum number of CNC channels was increased from 16 to 24.

Enable channel-ID wildcard character "*" in file device name
The channel-ID wildcard character '*' can be used in the file device name for the main program files and subprogram files. The wildcard is substituted for the channel identity character 'A' (1st channel), 'B' (2nd channel), etc. The settings is valid for <MAINSEARCHPATH> and <INCLUDEPATH> options in interpreter system configuration files (gmcipsys and gmcipus).

ID#319265 : solved problem, solved since V1.37.0
Wrong restart info in rotated coordinate system
From version 1.29.0, wrong restart info could be shown if the restart point is defined as rcPATHDISTANCE, and if it is inside the second or later movement after change of coordinate system's rotation (Gx92, FRAMEs). Axes' positions were shown in local coordinate system, not in global one.

ID#318595 : solved problem, solved since V1.37.0
NC program can't be reloaded if the init program is specified
Loading or starting of already loaded but modified NC program (reloading) failed with error 15307: "Program failed to load" with additional info "Status 15243". Error occurred only if init program (<cnc_obj>.move.ncprogram.init_prg) was specified.

ID#318365 : solved problem, solved since V1.37.0
Delay before G102 if full transformation is switched on
There might be a delay before G102 (Circle interpolation in general orientation) if full transformation was switched on. Furthermore, during that delay, a nonzero path speed could be shown in CNC monitor.

ID#318330 : new function since V1.37.0
Workspace monitoring available in 2D and 1D
Workspace monitoring and self-collision do not require full 3D Cartesian space anymore (i.e. three ncCNC axes in CNC channel). Namely in case of 2D, xy-plane is considered, and therefore z-coordinates of protected blocks can be omitted. In case of 1D, y- and z-coordinates can be omitted.

ID#317815 : solved problem, solved since V1.37.0
False identifier's ambiguity detected
Name of newly declared item (variable, function, etc.) can be wrongly detected as ambiguous. For example, variable of name 'getpos_status' is stated as ambiguous with already declared function 'getpos()'. Error is presented only in ARNC0 V1.36.1.

ID#315970 : new function since V1.37.0
New interpreter configuration files
Two new interpreter configuration files added:
- gmcipus: user override for (interpreter) system configuration (added to GmcIpConfig package)
- gmcipuc: user override for interpreter instance (channel) configuration (added to GmcIpUserConfig package)
Note:
if any of configurations is not presented on target the warning in AS logger will occur.

ID#315110 : solved problem, solved since V1.37.0
Possible memory violation when configuring max. path jobs
Configuring of the maximum number of path synchronous execution jobs (<MPQUEUE ... MaxPathJobs= "<number>"/> in gmcipsys) to values higher than default (256) may result to memory violation.

ID#314880 : new function since V1.37.0
Overloading of SCF and ICF
Content of SCF (System Configuration File - gmcipsys) and ICF (Interpreter Configuration File - gmcipcfg) can be overridden by later-provided definitions (e.g. by definitions in included XML files).

ID#313837 : Information valid since V1.37.0
ARNC0 manager error will written into the dpr command trace.

ID#313832 : Information valid since V1.37.0
ARNC0 writes axis errors also into the network command trace.

ID#400099982 : solved problem, solved since V1.37.0
Fedrate change if G25 is active
If feed rate will be changed and G25 is active, the new value will not be used in all consecutive movement blocks.

ID#313307 : Information valid since V1.37.0
Included drive operating systems
Library Acp10man with version 2.37.0 to 2.37.9 required.
The drive operating system with the same version as Acp10man is used.

ID#313297 : new function since V1.37.0
New NC actions for transferring any data blocks
A new interface is now available for uploading/downloading all types of data blocks (ParlDs with the type "DATA" or "BRMOD") using the NC actions listed below. The input/output parameters for this interface are contained in the "datblock" substructure of the data structures for real and virtual axes. This makes it also possible to upload/download data blocks in the NC Test.

- ncaction(ax_obj,ncDATBLOCK,ncUPLOAD)
The binary data for the defined data block is read from ACOPOS and saved into the specified data object.

- ncaction(ax_obj,ncDATBLOCK,ncDOWNLOAD)
The binary data for the defined data block is read from the specified data object and transferred to ACOPOS.

Parameters:
- datobj_name: Name of the data object
- datblock_par_id: Parameter ID of the data block
- idx1_par_id: Parameter ID of the data block Index1 (set to 0 if not used)
- idx1: Data block Index1
- idx2_par_id: Parameter ID of the data block Index2 (set to 0 if not used)
- idx2: Data block Index2

ID#313292 : new function since V1.37.0
New NC actions for transferring ACOPOS parameter tables
A new interface is now available for uploading/downloading data blocks. This new interface can also be used to upload/download ACOPOS parameter tables using the NC actions listed below. The input/output parameters for this interface are contained in the "datblock" substructure of the data structures for real and virtual axes. This makes it also possible to upload/download ACOPOS parameter tables in the NC Test.

- ncaction(ax_obj,ncACP_PAR+ncDATBLOCK,ncUPLOAD)
The specified ACOPOS parameter table is processed by the NC manager. ACOPOS reads each parameter in the ACOPOS parameter table individually and writes its value to the ACOPOS parameter table. Once all parameters have been read, the data module for the ACOPOS parameter table is regenerated with the updated parameter values.

- ncaction(ax_obj,ncACP_PAR+ncDATBLOCK,ncDOWNLOAD)
The specified ACOPOS parameter table is processed by the NC manager, and the parameters it contains are transferred to ACOPOS individually.

Parameters:
- datobj_name: Name of the ACOPOS parameter table

ID#310765 : solved problem, solved since V1.37.0
TCP additive elements in standstill
If there are more CNC channels, and if TCP additive elements are used in standstill, then the data might be mixed through channels -- namely if there is no movement in NC program / NC block before the standstill.

ID#309702 : new function since V1.37.0
New mode for interpreter single step and breakpoint halt
With "decoder.s_step.mode = ncSTEP_INTO+ncPATH" or "ncSTEP_OVER+ncPATH", the user can select whether the path follows immediately the interpreter or not.

ID#400095711 : solved problem, solved since V1.37.0
G126: Transitions between very short blocks will not be rounded
Transitions will not be rounded, if the block execution time of the affected blocks is shorter then two CNC cycles.

ID#400093695 : solved problem, known since MC_ARNC0_V1.33.1, solved since V1.37.0
G144 can't be disabled
Continuous alignment of the tangential axis (G144) can't be disabled in the CNC program using G145.

ID#400088453 : solved problem, solved since V1.37.0
Undefined movement according to G217/218/219
If another block than a movement block comes before a G217/218/219 mapping, it is possible that one of the two axes carries out an undefined movement after enabling or disabling the mapping function.

ID#400088110 : solved problem, known since V1.32.0, solved since V1.37.0
Axis from typ ncROTARY can be positioned to the upper bound of the intervall
Example: Interval 360°: Axis can be positioned to 360°, but the allowed range is 0 - 359.99...°.
For this reason, the restart position of the axis can differ one period and the restart will be aborted with an error.
Note: Use the value in the CNC structure, values shown in the CNC test window are rounded.

ID#40081708 : solved problem, solved since V1.37.0
Wrong axis is used in a CNC channel
Wrong axis may be used in a CNC channel if the name of an axis is equal to the first part of the name of an other axis (e.g. "AxX" and "AxX1")

NC Software - ARNC0 V1.36.1

ID#308885 : new function since V1.36.1
Dual-use compliant robots in ARNC0
Dual-use compliant robots from TRF_LIB can be used in ARNC0 even without arnc0ext library. Transformation library version V1.14.0 or higher must be used.

ID#308667 : Information valid since V1.36.1
Included drive operating systems
Library Acp10man with version 2.36.0 to 2.36.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.36.0
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#308600 : new function since V1.36.1
New AIL builtin functions / procedure to access the current block number
SetCurBlockNr() and GetCurBlockNr() allows to overwrite and read respectively the block number of the current block.

ID#307562 : new function since V1.36.1
New system variables to scale programmed values.
New system variables $SCALE_MAIN_PLANE_AXES, $SCALE_NORMAL_MAIN_PLANE_AXIS, $SCALE_LINEAR_AXES, $SCALE_GEOMETRY_PARAMETER to scale programmed coordinates and geometry parameters.

NC Software - ARNC0 V1.36.0

ID#307155 : solved problem, solved since V1.36.0
DISPLOF caused error 15798
DISPLOF (suppress NC block monitor) caused error 15798: "Insufficient Path Synch jobs configured - all jobs active in block_buffer"

ID#305482 : Information valid since V1.36.0
Included drive operating systems
Library Acp10man with version 2.36.0 to 2.36.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.36.0
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#303900 : new function since V1.36.0
New type of start position ncFILEOFFSET+ncSKIP_PARSE
New type of start position introduced for CNC programs (start_pos_type) introduced - ncFILEOFFSET+ncSKIP_PARSE. The ncSKIP_PARSE ensures that a G-code before a specified byte offset (start_pos) is entirely skipped.

ID#302157 : new function since V1.36.0
New G code "G239": Allow contour violation on small blocks if CDC is active.
No program abort if there are contour violations caused by small path sections and a large tool diameter. Contrary to G39 the contour violation will not be minimized and no blocks will be removed.

ID#301602 : solved problem, solved since V1.36.0
G212: No edge or approach process at a full circle
If G212 is active, it can happen that at the begin of a full circle no edge/approach process will be inserted. If G212+G138/G139+CDC activation, it can happen that no edge/approach process will be inserted (dependig on the geometry of the contour).

ID#300152 : new function since V1.36.0
Setup for induction motors: New optional parameter "phase_cross_sect"
setup.motor_induction.parameter.optional.phase_cross_sect: Cross section of a phase

ID#300142 : new function since V1.36.0
Automatic determination of motor parameters for synchronous motors
NC structure component "setup.motor_synchron"
NC actions "ncSETUP+ncMOTOR_SYNCHRON,ncSTART" and "ncSETUP+ncMOTOR_SYNCHRON,ncSAVE"

ID#300117 : new function since V1.36.0
New homing variants: Homing on block, fixed direction
New homing modes:
ncBLOCK_TORQUE: Homing on block with torque limit as condition for "block reached"
ncBLOCK_DS: Homing on block with lag error limit as condition for "block reached"
New homing parameters:
fix_dir: Fixed direction ON/OFF
torque_lim: Torque limit for homing on block
ds_block: Lag error for block detection
ds_stop: Lag error for stopping a movement

ID#400092444 : solved problem, solved since V1.36.0
Program abort (error 8147) if CDC is active
CNC program may be aborted with error 8147: "Radius of Circle equal to 0.0", if the tool radius and the programmed radius are nearly equal.

ID#292675 : new function since V1.36.0
Data type of NC system variables $AA.IM and $AA.IW changed from DINT to LREAL

ID#400087105 : solved problem, solved since V1.36.0
Program with G201 blocks + multiple restart
Program with G201 blocks + multiple restart
Multiple restart of an NC program with more then one G201 block will be aborted with error 7743: "Position out of 'In-Position-Tolerance' at START/RESTART of CNC move".

ID#400085902 : solved problem, known since V1.26.1, solved since V1.36.0
Incompatibility of G102 and kinematic transformations
If a circle is programmed with G102 while kinematic transformations are active, then joint axes could be planned to move to the wrong positions. These wrong positions are used for the axes only if G102 is not followed by another interpolated movement. Therefore, having G102 as the latest instruction in an NC program was producing an incorrect movement, as well as having G102 followed by a PTP movement, or G102 followed by dwell time G4, or G102 in a combination with G60, etc.

NC Software - ARNC0 V1.35.0

ID#299867 : Information valid since V1.35.0
Included drive operating systems
Library Asp10man with version 2.35.0 to 2.35.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.35.0
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#298612 : solved problem, solved since V1.35.0
Program execution stops if CDC is active
If CDC is active, it can happen that the CNC stops in G00 blocks with CDC overshooting is active ($CDC_OVS_DISTANCE > 100%). The NC program can be aborted.

ID#298110 : new function since V1.35.0
Current offsets in ARNC0NCCTR_typ
Current offsets (and not only rotations) are available via ARNC0NCCTR_typ. The offsets originate from G92/191/292 or FRAMEs.
ID#400090906 : new function since V1.35.0
Using of the axis name in NC program for the axis which is not defined in "cnc_obj.axis.ais[i]" structure
The new settings in Interpreter configuration file ensures that no syntax error will be reported if the axis is not defined

ID#400091806 : solved problem, solved since V1.35.0
Error 7745 arises at the start / restart of an CNC program (known since ARNC0 V.33.0)
Caused by a timing problem in the ARNC0, it can happen, that error 7745: "Axis movement already active at START/RESTART of a CNC move" occurs at the start / restart of an CNC program.

ID#296842 : new function since V1.35.0
G212 + CDC: Change in Behaviour
To generate the path speed profile for G212 and CDC is active, the programmed contour and not the equidistant contour will be used. To distinguish inserting an edge process or a fine edge process, the angle in the active main plane will be used.

ID#400090997 : solved problem, solved since V1.35.0
ncVARIABLE, ncGETINFO fails when locking variables from one channel and some variables are already locked on another channel

ID#400088445 : solved problem, solved since V1.35.0
(p)RTCP mechanics from TRF_LIB in ARNC0
If joint axis's properties of any joint axis in (p)RTCP mechanics is set to the non-default value (i.e. to TRF_MATH_NEGATIVE), then this settings is lost if TRF_LIB V>1.07.1 is used together with ARNC0 V<1.35.0.

NC Software - ARNC0 V1.34.1
ID#295515 : Information valid since V1.34.1
Included drive operating systems
Library Acp10man with version 2.34.0 to 2.34.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.34.1
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#294675 : solved problem, solved since V1.34.1
System variable $AX_V_JUMP not updated for G105
System variable $AX_V_JUMP was not updated when G105 (setting the "v_jump" parameter) was programmed. On contrary, the $AX_A_JUMP was updated for G106. The behavior is now consistent.

ID#400089615 : solved problem, solved since V1.34.1
NC actions for variable access: statuses not properly updated
The statuses <cnc_obj>.var_access.status.active and <cnc_obj>.var_access.status.complete was not properly updated for NC actions ncVARIABLE, ncGETINFO and ncVARIABLE, ncRELEASE.

NC Software - ARNC0 V1.34.0
ID#294230 : Information valid since V1.34.0
Included drive operating systems
Library Acp10man with version 2.34.0 to 2.34.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.34.0
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#400082317 : solved problem, solved since V1.34.0
Deadlock for ncFILE_XL programs started with byte offset
If an NC program was started in ncFILE_XL start mode with nonzero byte offset (start_pos_type = ncFILEOFFSET) then a deadlock occured.

ID#400078071 : solved problem, solved since V1.34.0
Kinematic transformations in standstill
Kinematic transformations are active also in situations where no movement is produced but tcp_add_el[] are changed, i.e. in halt, on synchronous M-flags, during dwell times, etc.

**NC Software - ARNC0 V1.33.1**

ID#294405 : Information valid since V1.33.1

Included drive operating systems

Library Acp10man with version 2.33.0 to 2.33.9 required.

CAN:
For ACOPOS 8V1xx.xx-00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xx.xx-00-2: ACP10SYS V2.33.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID# 400089399, 400088645, 400092044 : solved problem, known since 1.32.0, solved since V1.33.1

Missing error text (only in V1.31.1 - V1.33.0)

Error text hasn't been found for some error numbers.

**NC Software - ARNC0 V1.33.0**

ID#288127 : Information valid since V1.33.0

Included drive operating systems

Library Acp10man with version 2.33.0 to 2.33.9 required.

CAN:
For ACOPOS 8V1xx.xx-00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xx.xx-00-2: ACP10SYS V2.33.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#285422 : new function since V1.33.0

Changes in the status message for G212

The content of the system variable $PSM1_PHASE and of the monitor element "cnc_object.monitor.PSM_phase" is extended (see ARNC0 help).

ID#283772 : solved problem, solved since V1.33.0

Start of a CNC program even if an axis is already moving.

A CNC program can be started even if a movement on an axis - used in the CNC system - is already active (only ACP10 and SDC axes).

ID#400082978 : solved problem, solved since V1.33.0

Position jump on axis at program start

If a CNC program will be aborted by an axis error, a position jump on this axis can occur during the next program start. This effects ACP10 and SDC axes.

ID#282442 : solved problem, solved since V1.33.0

Missing set positions in one cycle

In very rare cases it can happen that no set positions are calculated for one cycle.

ID#282127 : solved problem, solved since V1.33.0

G212 (path speed mode 1)

The path speed can be too high on tangential block transitions if G212 (path speed mode 1) is active. This occurs only on "fine edge processes".

ID#282055 : solved problem, solved since V1.33.0

CNC Block Monitor didn't show the NC Block with G102

When the NC block with programmed G102 is executed the element "ncblock" of CNC Block Monitor structure didn't show that NC block.

**NC Software - ARNC0 V1.32.0**

ID#281467 : solved problem, solved since V1.32.0

NC-Actions not accepted if a ParId-Trace is active (only ARNC0 Axes)
Some ncActions (e.g. ncCONTROLLER / ncSWITCH_OFF) are not accepted if a ParId-Trace is active on an ARNC0 axis.

ID#279792 : Information valid since V1.32.0

Included drive operating systems

Library Acp10man with version 2.32.0 to 2.32.9 required.

CAN:
For ACOPOS 8V1xx.xx.xx-01: ACP10SYS V0.55.5
For ACOPOS 8V1xx.xx.xx-02: ACP10SYS V2.32.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#276197 : new function since V1.32.0

G212 (path speed mode #1) new

G212 defines the path speed profile at block transition (dwell time at block transition, reduced speed at begin of the following movement blocks)

ID#270960 : solved problem, solved since V1.32.0

Incorrect target position when activating CDC with G137

If CDC is deactivated and then re-activated one block later, then the target position of the entry block is usually not correct in relation to the active main plane.

The error only occurs when G137 is active.

ID#223882 : solved problem, solved since V1.32.0

Contour violation if s_jump_t = 180° and CDC is active

Contour violation on outer corners with alpha = 180° (inversion of movement) and if s_jump_t is set to 180° and CDC is active.

NC Software - ARNC0 V1.31.1

ID#278170 : Information valid since V1.31.1

Included drive operating systems

Library Acp10man with version 2.31.0 to 2.31.9 required.

CAN:
For ACOPOS 8V1xx.xx.xx-01: ACP10SYS V0.55.5
For ACOPOS 8V1xx.xx.xx-02: ACP10SYS V2.31.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#277232 : solved problem, solved since V1.31.1

Incorrect status cnc_obj->global.init after ncGLOBAL, ncINIT (CNC system)

Status cnc_obj->global.init will not be set to ncTRUE if a NC program was loaded into the memory.

ID#277125 : solved problem, solved since V1.31.1

Occasional error 15256 on loading NC programs

The error 15256: "PLC variable data-length mismatch" occasionally occurred when NC programs containing declarations of function blocks was parallelly loaded on more CNC channels.

ID#277092 : solved problem, solved since V1.31.1

NC program abort (mode ncAXIS) ignored directly after program start

A program abort (ncMOVE, ncE_STOP with parameter ncAXIS) will be ignored, if program abort will be performed directly after program start.

ID#400078230 : solved problem, solved since V1.31.1

Incorrect execution order of M-parameters and synchronous M-function

The order of execution might be incorrect, if a synchronous M-function is programmed after an M-parameter, a path synchronous variable or a path synchronous, non-blocking FUB. The execution order is correct after a movement block or after the program is started.

NC Software - ARNC0 V1.31.0

ID#274257 : Information valid since V1.31.0

Included drive operating systems
Library Acp10man with version 2.30.0 to 2.30.9 required.

CAN:
For ACOPOS BV1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS BV1xxx.00-2: ACP10SYS V2.30.1

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#273690 : new function since V1.31.0
New Restart Mode "ncABORT_LINE"

ID#272810 : new function since V1.31.0
CDC: Overshooting on acute angles ($CDC_OVS_DISTANCE)

NC Software - ARNC0 V1.30.2

ID#272420 : Information valid since V1.30.2
Included drive operating systems
Library Acp10man with version 2.30.0 to 2.30.9 required.

CAN:
For ACOPOS BV1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS BV1xxx.00-2: ACP10SYS V2.30.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#400759801 : solved problem, solved since V1.30.2
ACP10 axes and ARNC0: Fractional part of SW limits (CNC units) will be removed
SW limit values are converted from axis units into CNC units (division by the unit factor) in the ARNC0. The remainder will not be taken in consideration.

NC Software - ARNC0 V1.30.1

ID#271875 : new function since V1.30.1
TRF_direct3() and TRF_inverse3() from TRF_LIB usable in ARNC0
The third kind of direct and inverse transformations provided by TRF_direct3() and TRF_inverse3() functions from TRF_LIB is accessible in ARNC0.

ID#271870 : new function since V1.30.1
Tangential axis inside kinematic transformations
If the tangential axis is configured either as a path or as a joint axis, its position is not overwritten by outputs of the transformations. There can be a need for such a configuration (see e.g. TRF_direct3() and TRF_inverse3() functions for non-perpendicular 5ax mechanics, i.e. TRF_DATA05_typ from TRF_LIB).

ID#271790 : Information valid since V1.30.1
Included drive operating systems
Library Acp10man with version 2.30.0 to 2.30.9 required.

CAN:
For ACOPOS BV1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS BV1xxx.00-2: ACP10SYS V2.30.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#271775 : solved problem, solved since V1.30.1
Fixed memory leak from path-synch assigments (e.g. MF1=1) near end of NC program

ID#271765 : new function since V1.30.1
Introducing ip_global preprocessing macros
Syntax:
define ip_global <name> as <text>
Macro can be removed with undefine statement:
undefine <name>

ID#271755 : new function since V1.30.1
Line continuation and line-break characters
New syntax introduced:
"\"" - Line continuation (continue the current NC block on the next line)
"@@" - Line break (divide the line of G-code to two syntactically separate NC blocks)

ID#271720 : new function since V1.30.1
Reducing memory consumption for very short NC programs
Each NC program is loaded to a separate memory partition for effective memory clean-up during unloading. The minimum size of the memory partition was set to 384KB. This can cause wasting of resources if a large number of short NC programs has to be loaded to memory all at once - e.g. calling of multiple global subprograms.

New attributes for ncMANAGER was introduced that enable user to control size of memory partitions:
ARNCMemPartMin=\"<min_val>\" - Minimum size of Interpreter memory partition [hexadecimal]
ARNCMemPartMax=\"<max_val>\" - Maximum size of Interpreter memory partition [hexadecimal]

ID#400074886 : solved problem, solved since V1.30.1
Errors with additional information in CNC channel after a call of external functions
If an error with additional information was issued in a CNC channel after a call of external functions (e.g. functions from TRF_LIB), then it could lead to a pagefault.

ID#400072686 : new function since V1.30.1
G135 - Intersection path
To keep the contour violations as small as possible, the endpoint of the circular block is approached via the shortest path. In the past, the block was traversed in the same direction of rotation as the programmed block.

ID#263940 : solved problem, solved since V1.30.1
G201 and t_axfilter from "cnc_obj.axis.axis\[i\]" structure
The t_axfilter was set for an axis which is used as a trigger source. If the movement distance of this axis was equal to zero for the programmed NC block with G201 then the trigger wasn’t detected.

NC Software - ARNC0 V1.30.0
ID#268560 : Information valid since V1.30.0
Included drive operating systems
Library Acp10man with version 2.29.0 to 2.29.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.29.1
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#268550 : new function since V1.30.0
The maximum number of CNC channels was increased from 8 to 16

NC Software - ARNC0 V1.29.0
ID#268410 : Information valid since V1.29.0
Included drive operating systems
Library Acp10man with version 2.29.0 to 2.29.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.29.1
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#268390 : Information valid since V1.29.0
CNC plot buffer is extended by programmed feedrate.

ID#268380 : Information valid since V1.29.0
New G-code syntax elements
New G-code syntax elements introduced:
EXTERN - subprogram prototype
VAR - argument passed by reference
MCALL - modal subprogram call

ID#400070918 : solved problem, solved since V1.29.0
Automatic tangential axis (G141): Movement blocks with G00 are ignored.

Movement blocks with G00 are ignored after an automatic tangential axis is reactivated (G141).

ID#263840 : solved problem, solved since V1.29.0
NC action "ncGLOBAL, ncLOAD+ncINIT" fails without showing an error after unsuccessful start of axis compensation

ID#263275 : solved problem, solved since V1.29.0
Deadlock after CDC deactivation
CNC system is blocked if G172 (or STOPRE) is programmed immediately after a block with G40. The program can be aborted by the user.

ID#400061917 : solved problem, solved since V1.29.0
NC program can't be restarted if transformation is used
If the transformation library or G195 (RTCP) is used, the restart will be aborted with error 7743: "Position out of 'In-Position-Tolerance' at START/RESTART of CNC move".

NC Software - ARNC0 V1.28.1
ID#267500 : Information valid since V1.28.1
Included drive operating systems
Library Acp10man with version 2.28.0 to 2.28.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.28.0
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#400072360 : solved problem, solved since V1.28.1
M function programmed before blocks with zero movement distance has not been set if the rounding edges (G126) was active (only in V1.27.0 - V1.28.0).

ID#266790 : solved problem, solved since V1.28.1
Path-synchronous variable has not been assigned before blocks with circular interpolation if the rounding edges (G126) was active (only in V1.27.0 - V1.28.0).

NC Software - ARNC0 V1.28.0
ID#263830 : Information valid since V1.28.0
Included drive operating systems
Library Acp10man with version 2.28.0 to 2.28.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.28.0
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#263495 : solved problem, solved since V1.28.0
Occasional cycle time violation from TC#1
Combination of intensive subprogram calling and programming of multiple path synchronous jobs (e.g. FBs) from an NC program could cause a cycle time violation of TC#1.

ID#263115 : solved problem, solved since V1.28.0
Workspace monitoring - change to modelling based on the robot arm radius
Problem occurred if the protected area lay closer than the robot arm radius.

ID#400051865 : solved problem, solved since V1.28.0
New parameter "ipl_mode"
The new parameter "ipl_mode" in the "cnc_obj.axis.axis[i]" structure - interpolation mode for cyclic position. Parameter is valid only for ACP10 axes. The problem with a position overshoot at the end of an NC program has been solved with this new parameter.

ID# 400051713, 400051247 : solved problem, solved since V1.28.0
Error caused by Axis Factor not equal to 1
Axis Factor not equal 1 can be now set for ACP10 axes used in a CNC channel

Optional brackets for functions with no arguments in the alternative language 1 G-Code
In the alternative language 1 G-Code a function that does not require any arguments can be called without using brackets.

Exception 9101 - address error after <ncMOVE, ncSTOP>
Exception 9101 occurs if <ncMOVE, ncSTOP> NC action is sent before the first <ncPROGRAM, ncLOAD/ncSTART> if any additional LANG_INCLUDE files were loaded during ARNC0 startup.

Program freezes if it runs out of path synchronous commands (only in V1.25.1 - V1.27.2)
Starting with ARNC0 V1.25.1 the interpreter keeps track of all path synchronous jobs (path synchronous variable assignment, function calls) that are in the backtracing buffer (block_buffer).
The interpreter was only keeping track of a fixed amount of path synchronous jobs. The high value for <cnc_obj>.limits.block_buffer together with a high number of path synchronous jobs in an NC program caused the interpreter to run out of available jobs and freeze up.
The number of path synchronous jobs is now configurable in gmcipcfg in <MPQUEUE> section.

Output of blocks with non-synchronous technology functions or path synchronous variables and tool data number in the wrong order.
If a block with a tool data number follows immediately after a block with a non-synchronous technology function (S and T functions) or a calculation with path-synchronous variables, then the block with the tool data number will be output first.
Note:
Both blocks are output in the same ARNC0 cycle, which is why this effect can only be observed in single-step operation in Halt mode after each block (this occurs only in ARNC0 V1.27.0 to V1.27.2).

Restart not possible if restart switched off and on.
If the function for saving restart info is switched off and on (ncRESTART, ncSWITCH_OFF / ncSWITCH_ON) between aborting and restarting an NC program, the restart will be aborted with error 15307: "Program failed to load".
The drive operating system with the same version as Acp10man is used.

ID#260210 : solved problem, solved since V1.27.1
NC block with G170 + non synchronous M function, wrong order of execution (only ARNC0 V1.27.0)
In blocks with G170 + non synchronous M function, the M flag will be set not before G170 will be acknowledged.

ID#232652 : new function since V1.27.1
Program end command (e.g. M30) allowed in controll blocks (e.g. block with $IF)

NC Software - ARNC0 V1.27.0

ID#400065531 : solved problem, solved since V1.27.0
Polar Coordinate Machine: To low path Speed at Circular and Linear Blocks.
Numerical problems in dynamic calculation can cause low path speed.

ID#400064009 : solved problem, solved since V1.27.0
Polar Coordinate Machine: Low Path Speed at tangential Block Transitions.
The axes jolt filter was not be taken in consideraten in the calculation of path speed at tangential block transitions. The calculated value of path speed at block transition was too small. (only for polar coordinate machines)

ID#256647 : Information valid since V1.27.0
Included drive operating systems
Library Acp10man with version 2.27.0 to 2.27.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.27.1
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#400059569 : solved problem, solved since V1.27.0
Deadlock while executing an NC program
A Deadlock in the CNC system can occur, if the following sequence is processed repeatedly with the same NC program:
(1) Start NC program
(2) Stop NC program
The deadlock can only be resolved with a warm restart of the PLC.

ID#253145 : Information valid since V1.27.0
Memory optimization within the interpreter
Both the own memory usage of the interpreter and the memory consumption for loaded programs have been reduced.

ID#400062135 : solved problem, solved since V1.27.0
Non synchronous Technology Functions will set on a wrong path position if G126 is active.
Non synchronous Technology Functions will set on the begin of the bezier spline instead in the centre of the spline if G126 is active.

ID#252125 : new function since V1.27.0
Decomposition of frame to orientation angles and offsets
For determining the angles, depending on the used angle type, the interpreter functions F_TO_ANGLES, F_TO_EULER and F_TO_RPY are provided. For determining the offsets of the frame the function F_TO_TRANS is available.

NC Software - ARNC0 V1.26.1

ID#254322 : Information valid since V1.26.1
Included drive operating systems
Library Acp10man with version 2.26.0 to 2.26.9 required.
CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.26.2
POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#254317 : solved problem, solved since V1.26.1
Startup of ARNC0 axes get stuck in network phase 80 (only in V1.25.1 - V1.26.0)

ARNC0 axes with channel number 2, belonging to a drive connected to the POWERLINK network, get stuck in network phase 80 during startup.

**NC Software - ARNC0 V1.26.0**

ID#252382 : Information valid since V1.26.0
Included drive operating systems, dependency
Library Acp10man with version 2.26.0 to 2.26.9 required.

CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.26.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#251792 : solved problem, solved since V1.26.0
Circular blocks: standstill at block transition.
A standstill can occur at block transition between circular-circular blocks or circular-linear blocks.
The error can occur if the parameter "cnc_obj->limit.blocktransition = ncSTANDARD" or "cnc_obj->limit.blocktransition = ncAUTO" is set.

ID#242667 : new function since V1.26.0
Parameterized subroutine return (RET)
Usually, the end of subroutine returns to the calling program and the lines following the subroutine call will be executed. Parameterized RET allows program resumption at another, user defined position

**NC Software - ARNC0 V1.25.2**

ID#249930 : new function since V1.25.2
CNC programming instructions ADR and SIZEOF
ADR[] - determines the address of a data point
SIZEOF[] - determines the size of a variable in bytes

ID#249740 : new function since V1.25.2
Memory consumption optimization
Size of AIL opcode which was generated when loading NC program has been reduced.

ID#249512 : solved problem, solved since V1.25.2
G211 - Improved speed profile for short NC blocks
G211 (blended move mode) active: The combination of short NC blocks and long programmed acceleration time $TA could lead to dips in path speed profile.

ID#249382 : Information valid since V1.25.2
Included drive operating systems
Library Acp10man with version 2.25.0 to 2.25.9 required.

CAN:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.25.0

POWERLINK:
The drive operating system with the same version as Acp10man is used.

ID#248880 : solved problem, solved since V1.25.2
Reverse movement problem
Pagefault occurred if a path synchronous variable was assigned while moving backwards in an NC block.

ID#246795 : solved problem, solved since V1.25.2
Error in trigger configuration for axes accessed with PLCopen IF
It was not possible to select only the desired trigger events for the axes accessed with PLCopen IF. This can now be done using the configuration parameter "cnc_object.axis.axis[i].trg_source.trg_conf".

ID#234112 : solved problem, solved since V1.25.2
ARNC0 Deadlock if the same axis was connected to a CNC channel more than once

NC Software - ARNC0 V1.25.1

ID#400059370 : solved problem, solved since V1.25.1

Page Fault or Memory Access Violation during backward movement on the path.

During backward movement on the path a page fault or a memory access violation can occur if path synchronous variables or function blocks are used or if subprograms are called.

ID#248300 : Information valid since V1.25.1

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.24.1
For ACOPOSmulti 8BVxxxx.xx-xx: ACP10SYS V2.24.1

ID#400058384 : solved problem, solved since V1.25.1

Startup of POWERLINK axes blocked in phase 80

If POWERLINK axes with node numbers greater or equal 100 are used, the startup of the axes may block in network phase 80.

ID#248045 : solved problem, solved since V1.25.1

Page fault when restarting NC program

A page fault could occur by restarting an NC program with rotary axes (ncROTARY). Problem only in the version V1.25.0.

ID#247817 : new function since V1.25.1

The cyclic ARNC0 task is now installed in the NC Manager task class

The real-time part of the ARNC0 is now installed into the "NC Manager task class" which can be defined via the Aarnocfg.ncc configuration module. Up to now, the "NC Manager task class" was solely used for the communication between ARNC0 and the application tasks, whereas the real-time part of the ARNC0 has always been installed into TC#1 (unless the ForceSIOs parameter was set).

NC Software - ARNC0 V1.25.0

ID#247575 : Information valid since V1.25.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.24.1
For ACOPOSmulti 8BVxxxx.xx-xx: ACP10SYS V2.24.1

ID#400058124 : new function since V1.25.0

New start modes for NC action ncPROGRAM, ncSTART

Two new start modes ncFILE_XL+ncBLOCKMONITOR and ncDNC+ncBLOCKMONITOR have been defined to enable CNC block monitor for large or streamed NC programs.

ID#247420 : new function since V1.25.0

New parameters "rot_period" and "rot_offset"

The new parameters "rot_period" and "rot_offset" in the "cnc_obj.axis.axis[i]" structure can be used to set the period and offset of a rotary axis (ncROTARY+...).

ID#247410 : new function since V1.25.0

A new axis type in ARNC0: ncNOFEED

The single axes can be excluded from the feed rate calculation by adding ncNOFEED to the axis type. This makes it possible to exclude the axes of the types:

ncCNC + ncNOFEED
ncLINEAR + ncNOFEED
ncLINEAR + ncNOSTOP + ncNOFEED
ncROTARY + ncNOFEED
ncROTARY + ncNOSTOP + ncNOFEED
ncROTARY + ncSHORT_PATH + ncNOFEED
ncROTARY + ncNOSTOP + ncSHORT_PATH + ncNOFEED

from the feed rate calculation.

ID# 400056079, 400062509 : solved problem, solved since V1.25.0

Corrupted error text modules
Some of the error texts have not been evaluated correctly after calling of NC action ncMESSAGE, ncTEXT.
NOTE: All error text modules on an automation target must be updated for correct determination of error texts in ARNC0 V1.25.0!

ID#243502 : solved problem, solved since V1.25.0
Error in Restartinfo when read from a data module.
When loading Restartinfo from a data module, the restart may be aborted with error 7150: "NC program length different at 'RESTART'."
Condition: Parameter cnc_obj->restart.parameter.param_buffer = 0

ID#242672 : new function since V1.25.0
New Systemvariables $P_EP and $P_EM

ID#242287 : new function since V1.25.0
G180
(1) Beside the current syntax "G180=000" also "G180" is allowed.
(2) The last circle in a series of joined blocks may be programmed with center point, radius and angle.

ID#400053445 : solved problem, solved since V1.25.0
Incorrect angle of rotation in G102
If the last circular block before G102 is a full circle (360° angle of rotation), then the programmed arc as well as a full circle might be traversed in the block with G102. Whether the error will occur or not depends on the position of the center point.

ID#400052417 : solved problem, solved since V1.25.0
G70/G71 was taken in consideration also for axe of the type ncROTARY.
Inch/mm conversion not allowed for angle units.

NC Software - ARNC0 V1.24.2
ID#247167 : Information valid since V1.24.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.24.1
For ACOPOSmulti 8BVxxxx.xx-: ACP10SYS V2.24.1

ID#246200 : solved problem, solved since V1.24.2
G172 can cause a CNC-system deadlock (only in V1.24.0 - V1.24.1)

NC Software - ARNC0 V1.24.1
ID#245135 : new function since V1.24.1
Check of the orientation axes
When full transformations are switched on, the user is informed via a warning from ARNC0 when the positions of the programmed orientation axes in NC program are not uniquely defined.

ID#245115 : Information valid since V1.24.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.24.0
For ACOPOSmulti 8BVxxxx.xx-: ACP10SYS V2.24.0

NC Software - ARNC0 V1.24.0
ID#244030 : solved problem, solved since V1.24.0
Performance issue fixed: re-loading of modified global subprogram had been very slow.

ID#243945 : Information valid since V1.24.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.24.0
For ACOPOSmulti 8BVxxxx.xx-: ACP10SYS V2.24.0

ID#243765 : new function since V1.24.0
Extended monitor data
NC monitor structure have been extended with the call_level element which displays current call level of subprograms. The hierarchy of subprogram calling can be displayed as NC block numbers in a data buffer provided by an application program. An address of the data buffer has to be specified during ncBLOCKMON, ncSET calling.

ID#243575 : solved problem, solved since V1.24.0
The G201 at NC block with zero movement distance caused a deadlock.

ID#400053501 : solved problem, solved since V1.24.0
Invalid set position values in network interface
In very rare cases, an invalid value (NaN) is entered in the network interface for the decimal part of the set position.

ID#243150 : new function since V1.24.0
Accepting an ncPROGRAM,ncLOAD when an NC program is active
An ncAction ncPROGRAM,ncLOAD should be accepted when an NC program is active. The new functionality should allow user to speed-up loading of global subprograms if they are modified between the NC program start and calling of them.

ID#400030537 : solved problem, solved since V1.24.0
Restart - Deadlock at program abort
ARNC0 may become blocked if the program is aborted while restarting an NC program and ARNC0 is in the state "Waiting for movement to continue". The target must be restarted in order to remove the deadlock.

NC Software - ARNC0 V1.23.2
ID#242465 : solved problem, solved since V1.23.2
Workspace monitoring - self collision reported too early
Self collision error has been reported too early.

ID#242370 : Information valid since V1.23.2
Included drive operating systems
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.23.2

ID#242320 : solved problem, solved since V1.23.2
Unit factor not considered by G200/G201
The CNC unit factor was not taken in consideration by the latch position calculation (functions G200/G201).

NC Software - ARNC0 V1.23.1
ID#400053739 : solved problem, solved since V1.23.1
Workspace monitoring - diameter of the robot arm not considered
Diameter of the robot arm hasn't been considered by crossing a protected area.

ID#240975 : solved problem, solved since V1.23.1
NC monitor status "name_ncprog" not properly updated
The name of current NC program "name_ncprog" has not been properly updated for global subprograms.

ID#240970 : solved problem, solved since V1.23.1
Workspace monitoring - diameter of the robot arm not considered
Diameter of the robot arm hasn't been considered by crossing a protected area.

ID#240955 : new function since V1.23.1
Workspace monitoring - diameters of the robot arms as an array
Diameters of the robot arms can be defined as an array of values - separately for each arm.

ID#400052416 : solved problem, solved since V1.23.1
Contour violation when wrapping is active
A contour violation occurred when wrapping or mapping was active and CDC with arc transitions (G133) was chosen.

ID#240557 : solved problem, solved since V1.23.1
Error determining the RESTART-INFO in single-step operation
If the RESTART-INFO is determined with the parameter "restart.info.parameter.restart_type = ncBLOCKNUMBER") while single-step operation is active (path generator), then cryptic characters might be output for blocks:
- with a shift of the coordinate system (e.g. G92, G54)
- with synchronous or non-synchronous M-functions
- with real-time parameters (M-parameters)
- with dwell time (G04)
Whether the error will occur or not depends on the internal timing of the ARNC0.

ID#239420 : solved problem, solved since V1.23.1
G201 and NC program restart
NC program restart and restart info didn’t run correctly if G201 had been used.

ID#400043500 : solved problem, known since V1.03.8, solved since V1.23.1
Deadlock with negative override
ARNC0 was blocked if the following sequence was executed:
- start NC block or NC program
- set OVR to negative value
- wait until startposition of NC program is reached
- set OVR to zero
- set OVR to a negative value again.
Restart of target was needed to resolve the deadlock.

NC Software - ARNC0 V1.23.0

ID#240450 : new function since V1.23.0
Motion packet log
If enabled in Arnc0cfg, the motion packet log continuously records the contents of each motion packet packet into .mpl files. This functionality is switched on by default.
The log files are written into mplog* file devices which must be created by a user, one device per CNC channel (by default mplogA for 1st CNC channel, mplogB for 2nd CNC channel, ...).
New NC actions have been defined:
"ncMP_LOG, ncSWITCH_OFF" - switches the motion packet logging off
"ncMP_LOG, ncSWITCH_ON" - switches the motion packet logging on

ID#240445 : Information valid since V1.23.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.23.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.23.0

ID#240285 : solved problem, solved since V1.23.0
The movement didn’t slow down to standstill if G201 was used in a rotated product coordinate system
The G92 or SWFRAME was used to rotate the product coordinate system. If an axis with a programmed zero distance was set as a trigger source then the movement didn’t slowdown to standstill at NC block with G201 when the trigger occurred.

ID#240260 : solved problem, solved since V1.23.0
Latch system variables not set for CNC channel higher than 1

ID#400051683 : solved problem, solved since V1.23.0
Pagefault when calling AIL local function
ARNC0 crashed when analyzing a template function that contained a call to an AIL local function.

ID#239200 : solved problem, solved since V1.23.0
The M0 was skipped at the NC block with move distance equal to 0.0

ID#239075 : solved problem, solved since V1.23.0
NC program file not closed if syntax error
NC program file was locked if a global subprogram loaded from the main program contained a syntax error.

Logical operators

Following logical operators have been added to the G-code syntax:
- && - logical binary AND
- || - logical binary OR
- XOR - logical binary XOR
- ! - logical unary NOT

NC Software - ARNC0 V1.22.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.22.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.22.0

SG4 target system with AR A3.08 or higher: Error 9650 when using ARNC0 before V1.22.0

After optimizations some system functions are no longer contained in AR versions A3.08 or higher, which are needed by ARNC0 versions before V1.22.0. If an ARNC0 version before V1.22.0 is used with AR versions A3.08 or higher, then the following error is indicated during the project transfer or registered in the Logger during the PLC startup:
- 9650: Library function not available (System GOT)

For AR versions A3.08 or higher only the ARNC0 versions V1.22.0 or higher can be used.

New system variables have been added:

- $CO_ORDS_MODE represents modal coordinate definition
- $CENTER_MODE represents modal circle center point definition
- $MOVE_CMD_MODE represents modal movement command

New functions for programming of zero point offset and rotation of coordinate system have been added.

The coordinate programming mode (relative or absolute) can be specified non-modally with the AC and IC statements.

A jump to NC block with defined block number can be created with the GOTO statement.

Axis error during emergency stop: Movement state will not be actualized.

If an axis error (e.g. lag error) occurs while an emergency stop is active; it can happen that the movement state of the axis and the CNC-system will not be set correct after stand still. Possibly a restart of the large is necessary.

The G201 didn't work properly if as a trigger source was used an axis on SDC IF
The movement at NC block with G201 didn't slow down to standstill if the trigger was forced inside the trigger path section.

ID#236635 : new function since V1.22.0
Setup ISQ-Ripple for automatically determining the ripple parameters
New NC structure component "setup.isq_ripple".
New NC actions "ncSETUP+ncISQ_RIPPLE,ncSTART" und "ncSETUP+ncISQ_RIPPLE,ncSAVE".

ID#234757 : solved problem, solved since V1.22.0
Exceedance of programmed feed rate at block transition
Although G111 is programmed, the programmed feed rate of the consecutive block can be exceeded at block transition.

ID#234540 : solved problem, solved since V1.22.0
Deadlock issued by G201
NC program got stuck if there was the G201 function in the last NC block.

ID#233727 : new function since V1.22.0
G211 (Blended Move Mode)
Programmable, linear feed rate profile on block transitions.

NC Software - ARNC0 V1.21.1
ID#236465 : solved problem, solved since V1.21.1
Using G172 when the CDC is active will cause an NC program standstill
ID#236285 : Information valid since V1.21.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-1: ACP10SYS V2.21.1
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.21.1

ID#236102 : solved problem, solved since V1.21.1
Interpreter does not find task-local PVs
On rare occasions the Interpreter was not able to find task-local PVs that were declared in the Configuration-Module "gmcipvar" by using the syntax "TaskName:PVName".

ID#236045 : solved problem, solved since V1.21.1
Pagefault G200/G201
Pagefault occurred when an axis in standstill has been used as trigger source for G200/G201 functions and less than 15 axes have been configured in the CNC channel. This problem occurred only in version V1.21.0.

ID#400048448 : solved problem, solved since V1.21.1
Error when calling an NC subprogram
A syntax error occurred when calling a global subprogram with a name that starts with a number.

ID#234760 : solved problem, solved since V1.21.1
Robot orientation axes moved after a change was made to the product coordinate system

NC Software - ARNC0 V1.20.1
ID#235227 : Information valid since V1.21.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.21.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.21.0

NC Software - ARNC0 V1.20.1
ID#234345 : solved problem, solved since V1.20.1
G200 and G201 - axis in standstill as trigger source
Wrong positions of CNC axes have been latched, if an axis in standstill had been used as a trigger source for the function G200 or G201.
Workspace monitoring: Extension for Frames

Workspace could previously only be defined in the global coordinate system, now can it be defined in any coordinate system. The current coordinate system is remembered together with each protected area.

Functions G70 and G71 are ignored for the rotary axes

The units switch (G70 and G71) is from now ignored for the rotary axes (ncROTARY). The system units are always used for the rotary axes.

NC Software - ARNC0 V1.20.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.20.1
For ACOPOSmulti 8BVxxxx.xx-xx: ACP10SYS V2.20.1

The movement state of an axis was not correct after movement abort

After abort of a movement (axis movement or CNC program) the axis state was set to move.mode = ncOFF, before the axis was in standstill.

Exceedance of axis acceleration on tangential corners due to t_axfilter

The axis jolt filter can cause a violation of the allowed axis acceleration limits (acceleration on the path was not taken in consideration).

NC Software - ARNC0 V1.10.2

Memory footprint grew by 4 bytes per timestamp check

The timestamp is checked for each NC program start and each global subprogram call. Each timestamp check consumed 4 bytes of free memory.

The G201 deactivated programmed rotation of the coordinate system

Status of NC action remained "ncACTIVE"

Status of the NC action remained ncACTIVE after global init performed for a CNC object with more than 4 axes in a basis version of ARNC0.

NC program got stuck on short path elements

The NC Program got stuck on very short path elements if the path element had to be skipped because of high path speed (warning 7236: "Contour segment skipped, contour speed too high", and if the element was followed by a non-tangential transition.

NC Software - ARNC0 V1.10.1

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.20.1
For ACOPOSmulti 8BVxxxx.xx-xx: ACP10SYS V2.20.1

The parameter global.init not set
The parameter global.init of an axis object hasn't been set in some cases, if the axis object had been represented by a global PV.

**NC Software - ARNC0 V1.10.0**

ID#232417 : Information valid since V1.10.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.20.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.20.0

ID# 400046336, 400046111 : solved problem, known since 1.09.7, solved since V1.10.0

Page Fault during Boot Phase with Win7/ARsim (AR000)

When working with ARsim (AR000) on Windows 7, the ARNC0 caused a page fault during the system boot phase.

ID#231897 : solved problem, solved since V1.10.0

Member "class" of structure ARNC0MSREC_typ renamed to "errorclass"

In order to use ARNC0 in C++ programs the structure member "class" of ARNC0MSREC_typ has to be renamed to "errorclass". "class" is recognized as keyword for the C++ compiler and leads to an error.

ID#400045497 : solved problem, known since 1.09.6, solved since V1.10.0

Path speed is reduced to zero on short NC blocks.

Sometimes the movement on the path stops at short NC blocks, if mode "cnc_obj->limit.blocktransition = ncAUTO" is set.

**NC Software - ARNC0 V1.09.3**

ID#231787 : Information valid since V1.09.3

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.20.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.20.0

**NC Software - ARNC0 V1.09.2**

ID#231560 : Information valid since V1.09.2

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.19.1
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.19.1

ID#231530 : solved problem, solved since V1.09.2

Homing procedure aborted by error

The homing procedure on an ARNC0 axis was aborted with the error 5112: "Search Home procedure cancelled by Event ". The error started to occur in ARNC0 version 1.05.2 in combination with Automation Runtime B3.01.

ID#231315 : solved problem, solved since V1.09.2

Page Fault during ARsim (AR000) Boot Phase

When working on ARsim (AR000), a rather high number of axes could have caused an ARNC0 page fault during the system boot phase.

ID#226497 : solved problem, solved since V1.09.2

Trajectory speed jump because of different path acceleration at consecutive blocks

If in several blocks a lower path acceleration is programmed (e.g. G110), the path speed will not ramp to zero at a stand still but the speed will be forced to zero by the ARNC0. Furthermore the allowed acceleration on the axes can be violated.

**NC Software - ARNC0 V1.09.1**

ID#230720 : solved problem, solved since V1.09.1

Pagefault trying to configure more than 2 CNC channels

Pagefault occurred if user configured more than 2 CNC channels in the NC deployment table. This error was present from version 1.02.0.

ID#230705 : solved problem, solved since V1.09.1

Error stopping NC block or NC program
The following errors occurred after an NC block or NC program was stopped during the loading phase: 10106: "Event not allowed in current state", 15743: "Errors detected in block text" and 15319; "Operation aborted by user". The error 10106 was additionally followed by a deadlock.

ID#230155 : solved problem, solved since V1.09.1

Error 1114 with NC-Action ncGLOBAL/ncINIT

When there is a high CPU load on the runtime target, the NC action ncGLOBAL/ncINIT could fail with error 1114.

ID#230100 : Information valid since V1.09.1

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.19.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.19.0

ID#230095 : solved problem, solved since V1.09.1

Deadlock if NC-program was aborted when calling global subprogram

A deadlock would occur if an NC-program, which had called a global NC subprogram, was aborted by an error.

ID#230085 : solved problem, solved since V1.09.1

Pagefault when calling NC-subprogram with parameters

A deadlock would occur if an NC-program, which had called a global NC subprogram, was aborted by an error.

ID#400043708 : solved problem, solved since V1.09.1

The synchronized M Function is ignored if a path-synchronous variable is used in the same NC block.

NC Software - ARNC0 V1.09.0

ID#229340 : new function since V1.09.0

PLCopen IF in ARNC0

ACP10 axes kann be accessed with PLCopen IF from ARNC0

New NC structure component "nc_object.axis.axis[i].nc_object_plcopen"

ID#228755 : solved problem, solved since V1.09.0

Interpreter-Error was displayed in wrong CNC-Channel

An Interpreter-Error (e.g. Interpreter detects a syntax error during NC-Action ncPROGRAM/ncLOAD) in a CNC-Channel with Index other than 0 might have affected the CNC-Channel with Index 0.

ID#228750 : solved problem, solved since V1.09.0

ARNCO trace failed with the error 2104: "Invalid NC object for trace test data"

The trace test data were set properly but NC object "ncMODULE" issued the error 2104.

ID#228642 : solved problem, solved since V1.09.0

AILHeader section is limited to 4096 bytes in language configuration file (LCF).

ID#228595 : solved problem, solved since V1.09.0

Path speed drops to zero when assigning path synchronous variable

When path synchronous variable e.g. M-parameter was assigned in NC-program the path speed always dropped to zero. This happened even when the path synchronous variable was assigned between two tangentially connected path elements.

ID#228590 : new function since V1.09.0

Version control on config files according to ARNC0 version number

The standard configuration (AS package "GmclpConfig") files must have a version number that matches the first three digits of the ARNC0 version number (Vx.yy.z)

ID#228575 : new function since V1.09.0

Setup phasing for automatically determining the commutation offset

New NC structure component "setup.motor_phasing".

New NC actions "ncSETUP+ncMOTOR_PHASING,ncSTART" und "ncSETUP+ncMOTOR_PHASING,ncSAVE".

ID#228560 : Information valid since V1.09.0

Included drive operating systems
ID#228555 : new function since V1.09.0
Trigger Configuration for axes accessed with PLCopen IF
New NC structure component "cnc_object.axis.axis[i].trg_source.nc_object_plcopen".
New NC structure component "cnc_object.axis.axis[i].trg_source.trg_conf".

ID#227795 : new function since V1.09.0
Compensation of axis errors for axes accessed with PLCopen IF
New NC structure component "cnc_object.axis.axis[i].ax_compensation".

ID#227440 : solved problem, solved since V1.09.0
ncAUTOSAVE on ARwin (AR010) - Problem regarding access time
Trace with ncAUTOSAVE setting (saves trace data in text file automatically) took a long time when running on ARwin (AR010). The access time has now been optimized.

ID#227400 : solved problem, solved since V1.09.0
Position jump when using CAM wrapping
A position jump could occur on the rotary axis when the CAM wrapping hasn't been switched off in the previous NC program.

ID#227310 : solved problem, solved since V1.09.0
Internal errors after calling of external function from NC-program
Error sequence 10638, 10640, 10636 (internal errors) occurred as a response to a ncPROGRAM/ncSTART NC-Action if a previous program run had been aborted by a runtime error caused by an external function call (in case of a path-synchronous function with return type STATUS that returned a value between 1 and 65533).

ID#226952 : solved problem, solved since V1.09.0
Negative values for G108/G109/G110 ACC=<value> were not rejected
An error is is reported if the path acceleration exceeds the valid range.

ID#400041582 : solved problem, known since ARNC0 V 1.05.6, solved since V1.09.0
NC action ncAXES, ncINIT + cycle time violation
NC action ncAXES,ncINIT was adapted so that it no longer causes a cycle time violation when CPU load is too high.

ID#226460 : solved problem, solved since V1.09.0
G193, G194 not working since ARNC0 V1.00.0
The functions G193/194 (enable/disable "Linear feed characteristic" mode) have not been working since version V1.00.0. The functions G193, G194 are now supported again.

ID#226015 : solved problem, solved since V1.09.0
POWERLINK drive startup
If a drive that had not finished starting up was switched to ncCNCSYS simulation mode, the remaining drives didn’t finish starting up.

ID#225882 : Information valid since V1.09.0
Extension of Interpreter Single Step Functionality
- Step into subprograms or step over
- Definition of a number of blocks to halt after

ID#225877 : new function since V1.09.0
New Debugfunctionality for CNC Program Execution
- Breakpoints
- Variable watch and force function
- Expression execution
- Direct access to interpreter variables from PLC application tasks
- CNC blockmonitor display either path synchronous or interpreter synchronous

ID#224940 : solved problem, solved since V1.09.0
Page fault when initializing the axis settings

A page fault occurs while initializing the axis settings if the axis object "cnc_object.axis.axis[i].nc_object" has not been assigned to a CNC object.

ID#400038567 : solved problem, solved since V1.09.0
Rotation angle H falsely interpreted as variable name
Syntax error occurred when using G02 argument H (rotation angle) after use of variable starting with H.

ID#400034909 : solved problem, solved since V1.09.0
Axis Movement State at NC Program Emergency Stop
At NC program emergency stop (ncMOVE, ncE_STOP) with the parameter cnc_obj->move.e_stop.path = ncAXIS, the axis movement state will be set to axis_obj->move.mode = ncOFF before the axis stands still.

NC Software - ARNC0 V1.07.0
ID#226305 : solved problem, solved since V1.07.0
FRAME independent from TRF_LIB version
Reorientation of axes is now independent from the TRF_LIB version if FRAME is used.

ID#226300 : solved problem, solved since V1.07.0
Usage of local_frame without frame axes
local_frame is used also if frame axes are not defined. Values from the transformation variable are then considered.

ID#226295 : Information valid since V1.07.0
Included drive operating systems
For ACOPOS 8V1xxx.xx-00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.xx-00-2: ACP10SYS V2.18.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.18.0

NC Software - ARNC0 V1.06.3
ID#225365 : Information valid since V1.06.3
Included drive operating systems
For ACOPOS 8V1xxx.xx-00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.xx-00-2: ACP10SYS V2.18.0
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.18.0

ID#224855 : solved problem, solved since V1.06.3
Poor performance accessing files during startup
Reading the XML configuration files from the CF took a long time during ARNC0 startup. The access time has now been made 5 times faster.

ID#224815 : solved problem, solved since V1.06.3
Cutter diameter compensation: $RAD_IDX error
The change to an external parameter (EXF), which was used for indirectly accessing the tool radius, was detected too late. As a result, the tool radius value was updated too late. This error was present from version V1.00.0.

ID#224795 : Information valid since V1.06.3
Default error level for function blocks called from an NC-program changed from error level 4 (warning) to error level 3 (program halt)

ID#224415 : solved problem, solved since V1.06.3
Error 40137: "Internal Error - Invalid Job ID in ARNC0 response" on the axis object
The error 40137: "Internal Error - Invalid Job ID in ARNC0 response" was issued on the axis object if the NC program was aborted due to an interpreter error.

ID# 400036515, 400037166 : solved problem, solved since V1.06.3
Backwards Compatibility: Local Sub-Program and Main-Program may have the same Program-Number
Starting with ARNC0 V1.00.0 using the same program-number for a local sub-program and the main-program resulted in error 15196. For the sake of backwards compatibility, this is now allowed again.
NC Software - ARNC0 V1.06.1

ID#223355 : Information valid since V1.06.1

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.17.1
For ACOPOSmulti 8BVxxxx.xx-x: ACP10SYS V2.17.1

ID#223350 : solved problem, solved since V1.06.1

NC program stops after approx. 250 subprogram calls

ID#223345 : solved problem, solved since V1.06.1

Error 10442 occurs upon return from an NC subprogram

The error 10442: "A line containing an expression must not contain anything else" occurs after returning from an NC subprogram. The frequency of the error depends on the length configured for the MP queue.

NC Software - ARNC0 V1.06.0

ID#223167 : solved problem, solved since V1.06.0

CNC path movement stops after return from global subprogram.

ID#223047 : Information valid since V1.06.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.17.1

ID#400038189 : new function since V1.06.0

Allowed Contour Violation caused by the Axis Jolt Filter can be programmed in the NC Program.

The axis jolt filter causes contour deviation at circular blocks. The allowed contour deviation can be programmed in the NC program with system variable $filter_err_cir.

NC Software - ARNC0 V1.05.2

ID#222985 : Information valid since V1.05.2

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.17.1

ID#222760 : solved problem, solved since V1.05.2

Workspace monitoring: If the function TRF_get_joints_pos is not available

If the function TRF_get_joints_pos from TRF_LIB is not available for certain mechanical constructions, the NC program is no longer interrupted. Instead, only a warning is produced.

ID#222435 : solved problem, solved since V1.05.2

Error 40112 when loading data from INIT parameter module with subsequent global initialization

Loading data from INIT parameter module with subsequent global initialization (NC action ncGLOBAL, ncLOAD+ncINIT) was aborted with the error 40112: "Timeout at processing an ARNC0 command" if the same NC action had previously been aborted with the error 40245: "Error loading init parameter module". Error occurred only on the axis object.

ID#222225 : solved problem, solved since V1.05.2

Deadlock with runtime error

The deadlock occurred when the NC-program was aborted by runtime error (e.g. division by zero).

ID#222215 : new function since V1.05.2

Additive Frame

The user-defined frame is added to the currently used workpiece frame.
Syntax: $WFRAME_ADD = <frame_name>

NC Software - ARNC0 V1.05.1

ID#221365 : Information valid since V1.05.1

Included drive operating systems
Speed limit of tangential axis exceeded
The speed limit of the tangential axis was exceeded when circular interpolation was used.

Workspace monitoring
Workspace monitoring has been implemented. The working area can be defined and monitored.

In_pos_tolerance for full transformations
If a movement is started with full transformations switched on, then path and joint axes have to be in the corresponding positions. These positions are checked via direct transformation, and there is an allowed tolerance for path axes ("axis.transformation.in_pos.tolerance").

Frame axes
In addition to current joint and path axes, frame axes were introduced to the full transformations as well. These new axes are used if the whole robot (i.e. its basement) is moving.

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.17.1

FRAME variable describes new workpiece coordinate system. FRAME variable can be modified by set of operations as translation or rotation.

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.17.1

Error 10726 when calling ncaction() for NC objects, for which ncaccess() or ncalloc() have not been called
It is possible to define a global PV with type "ncAXIS" or "ncCNCSYS" for an NC object. In this case the address of this PV can be used as NC object for ncaction(), i.e. the NC object does not have to be determined with ncaccess() or ncalloc(). However so far this did not function, because with call of ncaction() the error 10726 occurred for all NC objects, for which ncaccess() or ncalloc() have not been called.

Error 40112 for POWERLINK axis in simulation mode "ncCNCSYS"
If the NC action "ncGLOBAL,ncINIT" was called for an POWERLINK axis in simulation mode "ncCNCSYS", when the correspondig ACOPOS was not available on the network, then the following error occurred:
- 40112: "Timeout at processing an ARNC0 command"

ARNC0 Axis: In ARNC0NCMON_typ the structure element "s_ncrecord" was used instead of "s_nccblock"

Error 10704 by ncalloc() and ncaccess() in TC#5 .. TC#8
The error 10704 occurred until now when calling ncalloc() or ncaccess() in TC#5 .. TC#8.
### Included drive operating systems

- For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
- For ACOPOS 8V1xxx.00-2: ACP10SYS V2.16.0

---

**ID#219477**: solved problem, solved since V1.03.4

Axis acceleration Limits exceeded at transient block transitions

On transient block transitions, the axis acceleration limits can be exceeded for one CNC cycle up to twice-time of the limit value.

---

**NC Software - ARNC0 V1.03.3**

**ID#218695**: solved problem, solved since V1.03.3

Cycle time violation of ARNC0 tasks

The cycle time violation occurred when NC-program ran on system with the high CPU load (>= 95%).

---

**ID#218670**: Information valid since V1.03.3

Included drive operating systems

- For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
- For ACOPOS 8V1xxx.00-2: ACP10SYS V2.16.0

---

**ID#217900**: solved problem, solved since V1.03.3

Page fault by a circular interpolation

A page fault occurred if a circular interpolation has been decoded and the first axis of a CNC object (cnc_obj.axis.axis[0]) hasn't been a Cartesian one.

---

**ID#217567**: new function since V1.03.3

Skip Function

There are up to 10 independent skip level available.

---

**NC Software - ARNC0 V1.03.2**

**ID#217175**: solved problem, solved since V1.03.2

Deadlock issued by NC-program loader error

If there is a calling of non-existing global subprogram at NC-program very begin, the program is blocked.

---

**ID#217165**: solved problem, solved since V1.03.2

Deadlock with NC-program started in ncFILE_XL mode

If an NC-program that has been started in ncFILE_XL mode was stopped, it couldn't be started again unless it was explicitly removed from memory.

---

**ID#217150**: Information valid since V1.03.2

Included drive operating systems

- For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
- For ACOPOS 8V1xxx.00-2: ACP10SYS V2.16.0

---

**ID#217120**: new function since V1.03.2

Error message 5152: 'In-Position-Tolerance' at START/RESTART of CNC move " has been replaced

The error message 5152: "Position out of 'In-Position-Tolerance' at START/RESTART of CNC move " has been replaced by the error message 7743 with the same meaning. The error 7743 is now issued by a CNC-object.

---

**ID#215385**: solved problem, solved since V1.03.2

Error 15138: "Bad arguments to exec built-in function " on ARsim (AR000)

Starting of an NC-program can cause the error 15138: "Bad arguments to exec built-in function "

---

**NC Software - ARNC0 V1.03.1**

**ID#215925**: Information valid since V1.03.1

Included drive operating systems

- For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
- For ACOPOS 8V1xxx.00-2: ACP10SYS V2.16.0
ID#215915 : solved problem, solved since V1.03.1
  Speed jump on ncLINEAR+ncNOSTOP axis
  s_jump_t is correctly interpreted for ncLINEAR+ncNOSTOP axes.

ID#215910 : solved problem, solved since V1.03.1
  G92 and full transformations
  G92 is usable for path axes without any limitation.

NC Software - ARNC0 V1.03.0

ID#215692 : Information valid since V1.03.0
  Included drive operating systems
  For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
  For ACOPOS 8V1xxx.00-2: ACP10SYS V2.16.0

ID#215407 : solved problem, solved since V1.03.0
  With active axis jolt filter sometimes an axis movement was not completed
  Sometimes with very high position values, the target position was reached but the positioning was not completed. This error occurred only with active axis jolt filter. Also the abort command (NC action "ncMOVE, ncSTOP") could be affected by this problem.

ID#214372 : new function since V1.03.0
  New NC structure component "nc_obj_inf.hardware" with information to identify the connected ACOPOS hardware module.

ID#213295 : new function since V1.03.0
  New NC action "ncSTOP,ncINIT" for POWERLINK and SDC axes for initialization of the stop configuration.

ID#210922 : new function since V1.03.0
  New NC structure components "limit.parameter.dv_stop" and "dv_stop_mode" to configure speed error monitoring.

ID#210917 : new function since V1.03.0
  New NC structure component "move.stop.drive_error" to define the deceleration ramp after occurrence of a drive error.

NC Software - ARNC0 V1.02.2

ID#215340 : Information valid since V1.02.2
  Included drive operating systems
  For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
  For ACOPOS 8V1xxx.00-2: ACP10SYS V2.15.3

ID#215305 : solved problem, solved since V1.02.2
  The same node numbers on the axes on the different interfaces
  Some problems like "Timeout at processing an ARNC0 command" could have occurred if the axes with the same node- and channel-numbers have been used on different interfaces. This problem occurred starting in version V0.24.0.

ID#215255 : solved problem, solved since V1.02.2
  NC-program restart with rotary axis
  Second program restart resulted to deadlock when rotary axis was out of 'In-Position-Tolerance'

ID#214415 : solved problem, solved since V1.02.2
  Trace not saved when more network Interfaces used
  The recorded Trace data wasn't been automatically saved to file when more than one network Interface had been configured.

ID#214130 : solved problem, known since V1.02.1, solved since V1.02.2
  Pagefault while NC-program running on ARsim (AR000)
  An NC-program running on ARsim (AR000) causes an exception followed by the service mode.

NC Software - ARNC0 V1.02.1

ID#214140 : Information valid since V1.02.1
  Included drive operating systems
ID#214135 : solved problem, solved since V1.02.1
Error 14126: "No cyclic positions from drive" on ARsim (AR000)
Error 14126: "No cyclic positions from drive" (only on ARsim (AR000)) is caused by usage of the SDC axes.

ID#213660 : new function since V1.02.1
Parameter "line_ncprog" in nc-monitor structure
A new parameter "line_ncprog" (line number in CNC program) in ARNC0NCMON_typ data type.

ID#213550 : solved problem, solved since V1.02.1
NC action ncLIMITS, ncINIT aborted with an error (only in V1.02.0)
NC action ncLIMITS, ncINIT may be aborted with an error 14198: "Error by acp10 mode switch".

ID#213447 : solved problem, known since V1.02.0, solved since V1.02.1
Timestamp check fails for subprograms
The time stamp was checked only for main program. The subprograms weren't reloaded when changed.

NC Software - ARNC0 V1.02.0
ID#213320 : solved problem, solved since V1.02.0
Bigger lookahead causes deadlock
If the size of the lookahead buffer was significantly increased, the deadlock could occur with several combination of ncaction called in short time interval.

ID#213310 : Information valid since V1.02.0
Binary AIL-based Interpreter
The G-code Interpreter has been modified to work on binary AIL. The modification reduced memory consumption and enhanced performance of the AIL interpretation. The syntax of the Interpreter configuration files was changed. The G-code which is defined in the language configuration files in text-based AIL is translated to binary AIL during ARNC0 startup. The <AILHeader> section is only translated to binary AIL and not executed while NC-program starts.

ID#213270 : solved problem, solved since V1.02.0
ParID Trace only for one ParID possible
Only the first configured ParID can be recorded with the ParID Trace, although more ParIDs has been configured.

ID#212522 : solved problem, solved since V1.02.0
Watch Dog Error
If the function ncaction() was not called in that task class, which was configured as "Task class for NC Manager task", then in very rare cases a watch dog error (logger error 9206) occurred.

ID#212080 : solved problem, solved since V1.02.0
Pagefault if a global PV is used as an NC object
A pagefault occurs when a global PV is used as an NC object.

ID#211150 : new function since V1.02.0
Error Levels of Function Block
Error level of function block determines what should happen when the FB returned an error. There are four error levels according severity of the error to define the reaction (1 - Emergency Stop, 2 - Program Stop, 3 - Program Halt, 4 - Warning).

ID#210827 : Information valid since V1.02.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.15.2

NC Software - ARNC0 V1.01.1
ID#210755 : Information valid since V1.01.1
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V2.15.1

**NC Software - ARNC0 V1.0.0**

**ID#209635 : Information valid since V1.01.0**

Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V2.15.0

**ID#209625 : solved problem, solved since V1.01.0**

G100 and G101

Point to point (PTP) movements are integrated into ARNC0. Transformation functions from TRF_LIB are called. Path axes and joint axes are still hold in the corresponding positions.

**ID#400023876 : solved problem, known since V0.84.3, solved since V1.01.0**

**Page Fault during Init-Phase in Service Mode**

On rare occasions a page fault occured in service mode during the initialisation phase of ARNC0. This page fault resulted in a cyclic (i.e. infinite) rebooting of the runtime target, which consequently inhibited online connections to the target.

**NC Software - ARNC0 V1.0.0**

**ID#216600 : Information valid since V1.00.0**

Position latch (G200/G201) syntax changed

Latched axes positions are copied to dedicated latch system variables. The automatic decoder and path movement synchronisation performed when EV-flag had been accessed was removed.

**ID#216585 : Information valid since V1.00.0**

C2 spline boundary conditions syntax changed

G801/G802: spline boundary conditions consist of axis name and boundary condition value.

Example:

G801 CE=0.1 BC1 X2.3 C0 A5.5 Z1.2 B3.8

**ID#206605 : Information valid since V1.00.0**

Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V2.14.0

**ID#205065 : solved problem, solved since V1.00.0**

ACP10MAN as axis driver for Powerlink and SDC axes

ACP10MAN is used as axis driver for Powerlink and SDC axes. Library acp10man has to be installed and is added automatically with the library arnc0man.

**ID#400023139 : solved problem, solved since V1.00.0**

Pagefault and CAN DDC error after ARNC0 startup

A pagefault occurred when no configured ARNC0 CAN-axis has been connected during ARNC0 startup.

**ID#201445 : solved problem, solved since V1.00.0**

New NC actions for PLK and SDC axes

New NC actions for PLK and SDC axes have been implemented:

*ncDAT_MOD,ncSERVICE,ncSAVE*
*ncPAR_SEQI,ncSERVICE,ncDOWNLOAD*
*ncPAR_SEQI,ncSERVICE,ncDOWNLOAD+ncINIT*
*ncPAR_SEQI,ncSERVICE,ncINIT*
*ncPAR_LIST,ncSERVICE,ncINIT*
*ncPAR_LIST,ncSERVICE,ncREAD*
*ncSETUP,ncCONTROLLER,ncSTART*
*ncSETUP,ncSAVE*
*ncSTOP*

**ID#400017838 : solved problem, solved since V1.00.0**

Delay of a set position on an axis on a virtual interface in mode 3
Usage of an ACOPOS or an ACOPOSmulti on an virtual interface in mode 3 caused a delay by copying fromACP10PAR_CYC_MASTER_SET_POS to ACP10PAR_SGEN_S_SET.

ID#184365 : solved problem, known since V0.67.0, solved since V1.00.0
Now block number allowed in the line with the programmnumber (before ARNC0 V1.00.0)

ID#176370 : solved problem, solved since V1.00.0
Reading the EPROM serial number
The reading of an EPROM serial number did not run correctly on the powerlink axes.

ID#162920 : solved problem, solved since V1.00.0
Error if last line was a control block
If the last row of a text file is any control block (e.g. $ENDIF) without <end-line> character, then the program was aborted with "Error 10259 (unknown statement)"

ID#161067 : solved problem, solved since V1.00.0
Error if NC program was started with noFILEOFFSET
If a program was started in a block with a tool place number or a tool data number close before a subroutine call, the programm could be aborted by the error 10212 (Multiple use or illegal combination of NC block type)

ID#146537 : solved problem, solved since V1.00.0
Everything programmed together with G17 or G18 or G19 was ignored.

ID#137557 : solved problem, solved since V1.00.0
No Program Abort at syntax error
NC program was not aborted at error 10276 "Unknown Term (syntax error)"

NC Software - ARNC0 V0.85.3
ID#210497 : solved problem, solved since V0.85.3
CNC Monitor was not updated immediately after Program Start
It could take some cycles after program start until the CNC monitor was updated first time (depending on system configuration and CPU load).

ID#210427 : Information valid since V0.85.3
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.15.1

NC Software - ARNC0 V0.85.2
ID#206607 : Information valid since V0.85.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.15.0

ID#400022995 : solved problem, solved since V0.85.2
Pagefault by a circular interpolation
A page fault occurred if a circular interpolation has been decoded and the first axis of a CNC object (cnc_obj.axis.axis[0]) hasn't been a Cartesian one.

NC Software - ARNC0 V0.85.1
ID#206477 : Information valid since V0.85.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.14.0

ID#205612 : solved problem, solved since V0.85.1
Position error on the automatic tangential axis at very short path sections when CDC is active.
When CDC is active, position errors can occur on an automatic tangential axis at very short path sections with allowed contour violations.
Contour Violation - Tool Radius to large (error 9247)

At a contour violation in consequence of a large tool radius the program will be aborted or only a warning will be displayed. In both cases the error number 9247 was used. Now for warning 9247 and for error 9274 is used.

G99 - Position error at very short path sections if coordinate system is rotated

When CDC is active and the coordinate system is rotated (G92, G192, G292), position jumps can occur if contour violations are allowed.

Page Fault at the end of a movement block

A page fault can arise if there is a gap between entries in the CNC axis configuration. The page fault occurs at the end of a movement block.

Strange Characters in NC block monitor

If the decoder has reached the end of NC program and the path is still running, strange characters can be displayed in NC block monitor.

System Crash if NC Program is active

Caused by a memory limitation in the ARNC0, the system can crash under certain conditions (overflow of an internal stack).

Table data (tool data, R parameter or zero point data) can be set also if a NC program is active. Saving the data is only allowed if no NC program is active.

A page fault occurs if a cycle time less than 800µs is set.

ARwin (AR010) + ARNC0 - Cycle time violation or page fault if cycle time is set to 400µs

The CNC Plot Buffer outputs one more circle if a full circle is defined in a CNC program.

Full circle detection in plot buffer

The CNC Plot Buffer outputs one more circle if a full circle is defined in a CNC program.

G144/G145: Continuous Alignment of Automatic Tangential Axis

The functions G144 and G145 are used to modify behavior of the automatic tangential axis on linear interpolation section (G01)
G144: Continuous alignment of automatic tangential axis

The tangential axis is continuously positioned during current NC block to be aligned with the start tangent of the next path section. This mode is analogous to the behavior of automatic tangential axis on the rapid interpolation section (G00).

G145: Standard automatic tangential axis behavior

The tangential axis keeps the position tangential to current path section. In tangential block transition (if maximum speed jump is exceeded) or in non-tangential block transition a stop is made and the tangential axis aligned.
When the CDC is active and the tool radius equals the radius of the programmed circle, then the ARNC0 can block.

New mode for calculation of acceleration on the path

With the parameter "cnc_obj.decoder.parameter.v_path_mode" can be selected, if the acceleration on the path is always limited to the value in "cnc_obj.limit.a_pos" or "cnc_obj.limit.a_neg" or only if cartesian axes are programmed in the NC block.

New restart option ncNO_CHECK. New NC action "ncPROGRAM, ncBLOCKSEARCH"

A restart of a modified NC-program is now possible with option noNO_CHECK.

The NC action "ncPROGRAM, ncBLOCKSEARCH" determines the starting positions of all axes when an NC program is started with a simulation run to a specified starting point.
ACOPOS-Multi doesn’t run correct in ARNC0.

ACOPOS-Multi couldn’t be used as Acp10 axis by ARNC0.

**NC Software - ARNC0 V0.62.1**

ID#167080 : Information valid since V0.62.1

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V2.02.0

ID#167025 : new function since V0.62.1

Usage of Acp10 axes in ARNC0

Acp10 axes can be used by ARNC0. It is possible to use e.g. ACOPOS-Multi for ARNC0 axes and the complete range of Acp10 manager functions (e.g. auto-tuning).

**NC Software - ARNC0 V0.62.0**

ID#167017 : Information valid since V0.62.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#167012 : solved problem, solved since V0.62.0

Restart for blocks without traverse path and without block number

The restart point is defined by the byte offset of the NC block (please note changes in data structure).

ID#166485 : solved problem, solved since V0.62.0

Halt in an NC block containing a synchronized M-Function

The NC program can not be continued when a halt is performed in an NC block containing a synchronized M-Function.

ID#162135 : solved problem, solved since V0.62.0

Restart in NC block with G201 generates error 7169

This problem occurs only when the NC program was aborted due to an axis error and the trigger input had not been activated.

**NC Software - ARNC0 V0.61.1**

ID#165912 : Information valid since V0.61.1

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#165907 : new function since V0.61.1

Length of ARNC0DBLST_typ now matches the length of the ACP10 data type

**NC Software - ARNC0 V0.61.0**

ID#165647 : Information valid since V0.61.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#165577 : solved problem, solved since V0.61.0

Positioning with target position 0 before first movement block

The first statement in an NC-program is M0 or M1. When a positioning with target position zero is startet, then the NC-program will be executed until the beginning of the first movement block.

ID#165572 : solved problem, solved since V0.61.0

Continue movement with negative override before the first movement block.

The first statement in an NC-program is M0 or M1. When the movement is continued with negative override, then the error 5107 "Event not allowed in current state" occurs on all CNC axes and the NC-program is aborted.
Position jump for G92 + G126
If, after a block with G92 (rotation of the coordinate system), a linear block is immediately followed by a block with G126, a position jump occurs (ARNC0 V0.55.0 and higher).

Position jump for G92 + Subprogram Call
If, after a block with G92, subprogram is called immediately, a position jump occurs.

Restart-info contains wrong axes positions
When the restart-info is determined with the block number of the first movement block, then the axes positions of the restart point are incorrect.

Filter time for contour jolt filter is not considered when running in simulation mode
The filter time for the contour jolt filter is considered only after an NC-program is run with the simulation mode switched off.

NC Software - ARNC0 V0.60.5
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

Polar coordinate motor + G114: Path speed too slow at circle path sections.
Even though G114 has no effect on the polar coordinate motor, the path speed was reduced to a very low value when programming with G114.
Note:
In calculation of path dynamics, a reduction of the path speed, which may be necessary, is carried out for the polar coordinate motor so that the relationship between the path speed and the path acceleration after the reduction is the same as the relationship between the limit values for the path speed and the path acceleration.

CNC Restart with sync. M-Functions generates error 7134
The error occurs only when M-function groups are defined such that a synchronized M-function is set at the restart point.

NC Software - ARNC0 V0.60.4
Polar coordinate motor: Axis limits exceeded
The axis limits for the rotating axis could be exceeded in a coordinate system that is rotated or shifted with G92/G192.

Polar coordinate motor: Path speed too slow at circle path sections.
Programmed speed sometimes not reached at circle path sections. Depending on the position and dimensions of the circle path section, parts of the circle which were not traversed were also included when calculating the limit values.

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4
NC action "ncGLOBAL,ncSAVE": The hardware assignment was not applied if an existing module was newly created

If an existing INIT parameter module is newly created with "ncGLOBAL,ncSAVE" and this module was assigned to an NC object via hardware configuration, then this hardware assignment should also be applied to the newly created module.

In previous versions this hardware assignment was not applied to the newly created module.

Page fault in ARNC0 after ncaction(ncBLOCK, ncSTART)

At first an NC program that does not exist is started. Then after ncaction(ncBLOCK, ncSTART) a page fault occurs in the ARNC0.

Parameter "sl_chain" has no effect

If the master axis is an CNC axis, the set positions of all coupled axes are not delayed according to "sl_chain". This produces a position difference between the master axis and the slaves axes.

Page fault, if a global NC subprogram is started with (ncBLOCK, ncSTART)

The page fault occurs upon return from the global NC subprogram.

The automatic switch off the compensation of the mechanical impreciseness of an axis.

The compensation of the mechanical impreciseness of an axis is switched off automatically if the controller is switched off. The compensation switch on is possible only if controller is on.

Rebuilding: Absolute positioning of the axes ncROTARY in a CNC program.

The behaviour has been so changed that the start position of the axis ncROTARY is not taken into consideration.

G171 after non-synchronized M Function

When a non-synchronized M Function in the DNC-Interface is followed by G171, then the flag for this M Function is not set.

G90 and G91 or G161 and G162 in the same NC block

Absolute and relative coordinates can be used in the same movement block.

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.5
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

G90 and G91 or G161 and G162 in the same NC block

Absolute and relative coordinates can be used in the same movement block.

Error in circle programming with angle specification.

If a block with circular interpolation with angle specification immediately follows a block with a coordinate transformation (e.g. G92 or G192), then the NC program is aborted with the error E154 (Radius difference between beginning and end).

Restart at block number, S, T and M functions are not updated.
When an NC program is restarted at a defined block number, the S, T and M functions are not updated at the restart point.

ID#159302 : new function since V0.60.1
Improved synchronization of CAN communication (ACOPOS - ARNC0)
Starting with AR P2.90, B2.92, the ARNC0 can detect system jitters in Automation Runtime and take them into consideration during synchronization.

NC Software - ARNC0 V0.60.0
ID#158345 : Information valid since V0.60.0
Improved synchronization of CAN communication (ACOPOS - ARNC0)

ID#158330 : solved problem, solved since V0.60.0
Free definition of the active plane using the mapping functions
The functions G217, G218, G219 allow the free definition of active plane using the mapping. These combinations of definition axis-types are possible: ncCNC-ncROTARY, ncCNC-ncLINEAR, ncCNC-ncCNC (standard case).

ID#158325 : solved problem, solved since V0.60.0
Set positions of the axes ncROTARY in the monitor of the CNC object
For the set positions is used the interval (0.0, 360.0) instead of the interval (-360.0, 360.0).

ID#158320 : new function since V0.60.0
Absolute positioning of the axes ncROTARY in a CNC program.
The behaviour has been modified in this way that now it more matches the characteristic of "rotary" axes.

ID#158067 : new function since V0.60.0
Runtime of an NC-program in simulation mode
The runtime of an NC-program can be determined by running an NC-program in simulation mode.

NC Software - ARNC0 V0.59.2
ID#157670 : Information valid since V0.59.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#157665 : solved problem, solved since V0.59.2
Problem with CAM coupling
If a step change in desired position comes for a slave axis the peek in acceleration in oposite direction occures at the 3rd sample after the step change.

NC Software - ARNC0 V0.59.1
ID#157257 : solved problem, solved since V0.59.1
Circle calculation error in the CDC
An error in the CDC could cause a radius of zero to be calculated for a circle path section. In such a case, the NC program was stopped (G36 active) or the circle was replaced by a straight line (G37 active).

ID#157252 : solved problem, solved since V0.59.1
Combination of G220 + G170 + G40 produces a deadlock of the ARNC0.
The ARNC0 stops if blocks with G220 (signal before path section end over multiple path sections), G40 and G170 follow one another consecutively, without having a traverse path programmed in between.

ID#157075 : Information valid since V0.59.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4
Position not equal zero by Wrapping on/off allowed.

It is no more necessary that the positions of "source" and "destination" axes are by Wrapping on/off equal zero.

NC Software - ARNC0 V0.59.0

ID#156877 : Information valid since V0.59.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#155532 : solved problem, solved since V0.59.0
Wrong path speed in blocks with G126
In CNC systems without a tangential axis the path speed may be wrong in blocks with G126.

NC Software - ARNC0 V0.58.4

ID#156505 : new function since V0.58.4
New G-codes G114, G115
With G114, bigger part of acceleration limits is allowed to be consumed by centripetal acceleration on arcs. G115 means the original behaviour, and it is default. G114 is valid from its first occurrence till G115 or till the end of CNC program.

NC Software - ARNC0 V0.58.3

ID#155947 : Information valid since V0.58.3
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

NC Software - ARNC0 V0.58.2

ID#155652 : Information valid since V0.58.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

NC Software - ARNC0 V0.58.1

ID#154897 : solved problem, solved since V0.58.1
Restart at block number Restart point not found.
Processing a NC program with very short path sections or using a block buffer with few elements (limit.block_buffer), it could happen that the restart point will not be found. There is no error if the restart point is defined by path distance.

ID#154865 : Information valid since V0.58.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4
ID#154860 : solved problem, solved since V0.58.1
CNC Plot Buffer: - The decoding-end is not reported in Plot Buffer Header in simulation mode.
In simulation mode remains the old value of the parameter "plot_header.status" (ARNC0PLOTHEADER_typ), although decoding has been finished.

NC Software - ARNC0 V0.58.0
ID#154645 : solved problem, solved since V0.58.0
CNC Plot Buffer doesn't take the functions G92 and G126 into consideration.
The functions G92 (Programmed Zero Point Offset and Rotation of the Coordinate System) and G126 (Rounding Edges) had no influence on the CNC Plot Buffer.

ID#154392 : solved problem, solved since V0.58.0
Synchronizataion ACOPOS via CAN
The ARNC0 synchronization mechanism for CAN network calculates the start for an ideal time frame according to the mean jitter of the system. The start is set in a way that the mean jitter (for 96 synchronization periods) tends to zero. If the mean jitter is above or below, the start is shifted by 1µs per period in order to achieve zero jitter.

ID#154387 : Information valid since V0.58.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#154382 : new function since V0.58.0
Single step mode - Stop also at blocks with no movement

NC Software - ARNC0 V0.57.1
ID#154470 : solved problem, solved since V0.57.1
CNC-Plot-Buffer didn't function in simulation mode.
The writing to the CNC-Plot-Buffer didn't function, if the CNC object was switched to the simulation mode.

NC Software - ARNC0 V0.57.0
ID#154310 : Information valid since V0.57.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#154305 : new function since V0.57.0
CNC-Plot-Buffer.
A memory-area allocated in the application can be used as a CNC-Plot-Buffer.
New / changed elements in the data structure:
German:
"cnc_object.grenzwert.plot.access_adr"
English:
"cnc_object.limits.plot.access_adr"

NC Software - ARNC0 V0.56.0
ID#153742 : Information valid since V0.56.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

ID#153737 : new function since V0.56.0
Restart for blocks without traverse path
NC programs restart when using blocks. The reset point is defined by the block number (please note changes in data structure).

NC Software - ARNC0 V0.55.2
ID#153657 : Information valid since V0.55.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.4

**NC Software - ARNC0 V0.55.1**

ID#153452 : Information valid since V0.55.1
- Included drive operating systems
  - For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
  - For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.2

ID#153185 : solved problem, solved since V0.55.1
- Pagefault in CNC channels with ncROTARY Axes (only ARNC0 V0.55.0).
  - Pagefault was caused by using a CNC channel with ncROTARY axes, if fewer than 9 axes were defined.

ID#153107 : solved problem, solved since V0.55.1
- Positioning in the current NC Program fails to reach target position 0
  - When a positioning with target position 0 is started while s_ncprog in the CNC-monitor is negative, then this positioning fails.

ID#153027 : solved problem, solved since V0.55.1
- Set position jump due to a G92 sequence (only ARNC0 V0.55.0).
  - A position jump occurs on the axes if two blocks with G92 follow each other with only one block programmed between them having a traverse distance s=0.

**NC Software - ARNC0 V0.55.0**

ID#152690 : Information valid since V0.55.0
- Included drive operating systems
  - For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
  - For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.2

ID#152680 : new function since V0.55.0
- ncROTARY - the new axis type
  - The new axis type has been implemented to allow the Flat Cam Programming.

ID#152347 : solved problem, solved since V0.55.0
- Wrong speed at the path section transition.
  - In a rotated coordinate system (G92, G192), the speed at the path section transition is sometimes calculated incorrectly.

ID#152342 : new function since V0.55.0
- No stop after blocks with G92
  - Until now, a stop was always implemented at transition blocks after blocks with G92/G192, a tool data number or an absolute zero point offset (G54 etc.). The permissible speed at the path section transition is now calculated according to the axis and path limits.

**NC Software - ARNC0 V0.54.1**

ID#152097 : Information valid since V0.54.1
- Included drive operating systems
  - For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
  - For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.2

ID#152077 : solved problem, solved since V0.54.1
- Page fault in global NC subprogram
  - When a global NC subprogram is terminated by %progno instead of M2, M29 or M30, then a page fault occurs.

**NC Software - ARNC0 V0.54.0**

ID#151282 : new function since V0.54.0
- New trace data points in the CNC block monitor
  - These variables in the CNC block monitor can now be traced: path length until end of current NC-block, path length of current NC-block, path length of next NC-block in the direction of motion.
New path length variables in CNC block monitor

The CNC block monitor additionally displays the path length of the current NC-block and the path length of the next NC-block in the direction of motion.

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.2

Restart - Current axis positions in the DPR trace

The current axis positions are now recorded in the DPR trace when restarting an NC program. In previous versions, the positions were stored when starting the programming.

Path distance in CNC Monitor after CNC Init

The ncaction(..., ncLIMITS, ncINIT) resets all variables in the CNC monitor except the set positions. It also resets the path length variables in the CNC block monitor.

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.24.2

External encoder - Homing error

The mode without reference pulse is always used no matter which homing mode has been defined (with or without pulse). An error message is not given.

Save data to an INIT-Parameter-Module. Target memory User ROM.

The INIT-Parameters contained in the NC-object user data are saved to the given INIT-Parameter-Module in User ROM using the NC-action "ncGLOBAL,ncSAVE".

Overlapping signal function in a path section with zero length

When an overlapping signal function is programmed in a path section with zero length, the error 8155 “Distance is equal to 0.0, signal will be ignored” results, although there is enough distance for the signal ahead of this path section.

Save data to an INIT-Parameter-Module. Target memory User RAM.

The INIT-Parameters contained in the NC-object user data are saved to the given INIT-Parameter-Module using the NC-action "ncGLOBAL,ncSAVE".
NC Software - ARNC0 V0.51.2
ID#149452 : Information valid since V0.51.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#146767 : solved problem, solved since V0.51.2
Non synchronized M-functions between dwell times
If non synchronized M-functions are programmed between two dwell times, then the M-functions are output after the second dwell time has passed.

NC Software - ARNC0 V0.51.1
ID#146905 : solved problem, solved since V0.51.1
Deadlock with function combination G221+G220/G222+G170
The CNC blocks at the G221, when G221 is programmed together with a synchronized M-function and the NC-block containing a G220 or G222 has zero length and is followed by a G170.

ID#146897 : Information valid since V0.51.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#146892 : solved problem, solved since V0.51.1
Standstill when changing the active working plane.
Movement is not stopped when changing the working plane if tool length compensation is not active (tool length = 0) or the programmed plane is already selected.

NC Software - ARNC0 V0.51.0
ID#146585 : solved problem, solved since V0.51.0
Turning of a circle & its dynamics
In case of too low axis acceleration limits (lower than limits of CNC object), the path-speed on the circles (G2, G3) could be reduced more than necessary.

ID#146575 : solved problem, solved since V0.51.0
Problem with Restart of an NC-program
NC-program creation time was not saved correctly in case of "move.ncprogram.start_mode == ncFILE" or "move.ncprogram.start_mode == ncFILE_XL", Error 7151 could occur sometimes.

ID#146037 : solved problem, known since V0.50.0, solved since V0.51.0
Error in german user data structure
The german user data structure for the NC Object ncCNCSYS contained the member monitor.typ_ncblock instead of monitor.typ_ncsatz.

ID#146032 : new function since V0.51.0
Control structure for NC test
The behaviour of NC test functions for the NC objects ncAXIS and ncCNCSYS can be selected by the new component nc_test.
nc_test.Open_UseApplNcObj:
1 = Test function with application object
0 = Test function with standalone test object
nc_test.Close_NoMoveAbort:
1 = No move abort when the test function is closed.
0 = Abort all active movements when the test function is closed.
The global initialization is done with the attribute NoManCtrl of the NC Object ncMANAGER in the NC mappin table.

ID#146027 : Information valid since V0.51.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6
NC Software - ARNC0 V0.50.3
ID#145952 : solved problem, solved since V0.50.3
Incorrect cutter diameter.
The radius from the previous NC program is used when activating CDC if a cutter diameter has not been defined in an NC program (tool data number or $RAD).

ID#145937 : solved problem, solved since V0.50.3
The tool length and tool offset are wrongly accounted for in CNC systems with less than three path axes.
In a CNC System with two cartesian axes, the tool length correction and the tool offset of the non-existent third cartesian axis are calculated in the first linear axis.
In a CNC system with one cartesian axis, the compensations for both of the non-existent cartesian axes are incorporated in the first two linear axes.

ID#145862 : Information valid since V0.50.3
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#145857 : solved problem, solved since V0.50.3
Position error on the tangential axis when activating the cutter diameter.
If cutter diameter compensation is activated with the following conditions, the automatic tangential axis in the path section that immediately follows the activation will be aligned incorrectly. As a result, the tangential axis will be aligned to the end position of the block during movement as in a block with G00.
- Automatic tangential axis is active
- Linear interpolation block (G01)
- Indirect activation with transition block (G137)
- The transition angle of the subsequent path section transition is greater than 180° (outside corner)

ID#144385 : solved problem, solved since V0.50.3
Unjustified error 8134: "Radius difference between start and end".
The error occurs under the following conditions: There are only two cartesian axes and the ZX-plane has been selected.

NC Software - ARNC0 V0.50.1
ID#145102 : Information valid since V0.50.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#145092 : solved problem, solved since V0.50.1
Compensation matrix for cartesian axis is not taken in consideration.
It could happen, that the the compensation matrix ("p_cnc_obj.axis.compensation.matrix_el[]") is not taken in consideration.

ID#144130 : solved problem, known since V0.47.1, solved since V0.50.1
Page fault while calling a local subroutine.
A page fault occurs if in a line with the call of a local subroutine are additional syntax elements.

NC Software - ARNC0 V0.50.0
ID#145082 : Information valid since V0.50.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#144907 : new function since V0.50.0
New display mode and data in the CNC monitor, new system variables for tool data and zero point offset.
New display mode for position in the CNC monitor:
- Machine coordinates
- Consideration of the coordinate system transformation, tool data offset and tool length
- Consideration of the coordinate system transformation
- Consideration of tool data offset and tool length
CNC monitor shows number of the active tool data record and index of absolute zero point offset.
New system variables: Active tool data record, tool location number and index of absolute zero point offset can be read in the NC program.

ID#133367 : new function since V0.50.0

Unification of the trace record number calculation (ACP10 - ARNC0).

NC Software - ARNC0 V0.49.1
ID#144955 : Information valid since V0.49.1

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#144940 : solved problem, solved since V0.49.1

CNC program in form of text file can lie in any directory
Till version V0.49.0 the text files had to lie in directory "C:\CNC_Prg". Now can the directory be chosen using the "CPU-properties-File Devices".

NC Software - ARNC0 V0.49.0
ID#144950 : Information valid since V0.49.0

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#144135 : solved problem, solved since V0.49.0

Start of CNC programs and subprograms from text files
CNC programs can be used in form of text files without the previous conversion. Parameter "move.ncprogram.start_mode" defines the start mode of a CNC program.

NC Software - ARNC0 V0.48.3
ID#144022 : solved problem, solved since V0.48.3

Automatic tangential axis: first and last synch. M-function can not be used as sync1_t or sync2_t
Use of the first or last synch. M-function for sync1_t or sync2_t, generates the error 8244 "G141 M-function sync1_t or sync2_t not in the admissible range".

ID#144017 : solved problem, solved since V0.48.3

Rotation angle of 90 degrees results in a half circle
If a circular arc is programmed via a rotation angle immediately after a circular arc, it can happen that a wrong arc is traversed or the error 8134: "Radius difference between start and end" occurs.

ID#143262 : solved problem, solved since V0.48.3

Reduction in path speed along short NC-blocks
An undesired reduction in path speed along short NC-blocks can occur with the parameter setting "cnc_obj->limit.blocktransition=ncAUTO".

ID#141325 : solved problem, known since V0.45.1, solved since V0.48.3

Unjustified error after ncaction(ncAUTOMAT, ncINIT)
When initializing the Cam Profile Automat with correct data, the error 5157: "Cam Profile Automat INIT - Invalid parameter in the basis state" occurs. This bug exists since version V0.43.0.

ID#138380 : solved problem, known since V0.42.6, solved since V0.48.3

"cnt_ncprog" remains "1" after stopping a movement
If immediately after starting an NC-program a "ncMOVE, ncHALT" followed by a "ncMOVE, ncSTOP" is given, the CNC-System blocks.

NC Software - ARNC0 V0.48.2
ID#142662 : Information valid since V0.48.2

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#142652 : solved problem, solved since V0.48.2
Incorrect display in the CNC monitor.
Incorrect positions could be displayed in the CNC monitor during standstill.

ID#142647: solved problem, solved since V0.48.2
Page Fault at ncaction(nc_object, ncZEROPMON, ncSWITCH_ON/ncSWITCH_ON)
A Page Fault could occur if one of the two actions were called before an NC program was started.

NC Software - ARNC0 V0.48.1
ID#142507: solved problem, solved since V0.48.1
Cutter diameter accounted for at G153.
The cutter diameter was taken into consideration at G153 (position entry in the machine coordinates) if cutter diameter compensation was active. The corresponding point on the equidistant segments was moved to instead of the programmed point.

ID#142297: Information valid since V0.48.1
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.19.6

ID#141601: solved problem, known since V0.46.0, solved since V0.48.1
NC program aborted when selecting an automatic tangential axis.
The NC program was aborted with error 8244 if one of the two parameters "limit.s_sync1_t" or "limit.s_sync2_t" was set to the value ncOFF when selecting the automatic tangential axis (G141). An M-function for lifting the tool is not output if "limit.s_sync1_t" has the value ncOFF. An M-function is also not output for lowering the tool if "limit.s_sync2_t" = ncOFF. In both cases, a warning is generated when activating the tangential axis.

ID#141555: solved problem, solved since V0.48.1
Page Fault when restarting an NC program (since ARNC0 V0.40.0).
A Page Fault could occur when restarting an NC program.

NC Software - ARNC0 V0.48.0
ID#141995: new function since V0.48.0
Saved trace file includes informations about the traced values.
Saved trace file format has been changed from MathCad to MathLab, which includes also the header with the information about the traced data points. Header carries an information if the traced data point is CNC or Axis (including its index)parameter, data point name, trace date and time, X and Y units.

ID#141987: new function since V0.48.0
M-Functions are set after a restart.
It is now possible to define groups of M-Functions. After a restart the last programmed M-Function of every group is set.

ID#141835: solved problem, solved since V0.48.0
Pagefault after Calling ncaction(ncCOMP, ncSTART)
Pagefault was caused by calling ncaction(ncCOMP, ncSTART) with the parameter*move_compensation.parameter.mode = ncOFF*.

ID#141772: Information valid since V0.48.0
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.19.6

ID#141767: solved problem, solved since V0.48.0
Incorrect tangential axis position in the first movement block after multiple rotations of the coordinate system.
The position of the tangential axis in the first movement block could be incorrect if the coordinate system is rotated several times consecutively without moving an axis in the meantime.

ID#141762: new function since V0.48.0
New ARNC0 behavior after a block with a tool data number.
After a block with a tool data number, the tool length and the tool offset are only traversed with absolute programming (G90) and with programmed axes. The compensation movements are not executed with relative programming (G91) or non-programmed axes (see ARNC0
Monitor display: The setting "monitor.status.nullpver = on" causes the position of the tool bit in the programmed coordinate system to be displayed in the CNC monitor. The position of the new tool bit is displayed immediately after a block with a new tool data number. The setting "monitor.status.nullpver = off" causes the position (tool clamping point) in the machine coordinate system to be displayed.

ID#137597 : solved problem, solved since V0.48.0
Position error on the tangential axis after rotating the coordinate system.
The tangential axis is aligned to the wrong angle in the transition block if a subprogram is called before G92 or G192 with a rotation of the coordinate system.

NC Software - ARNC0 V0.47.1
ID#141862 : Information valid since V0.47.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#141230 : solved problem, known since V0.45.1, solved since V0.47.1
Pagefault after Calling a Global Subprogram
Pagefault was caused by calling a global subprogram (presented as a BR-module on PLC) from a main program started from file.

NC Software - ARNC0 V0.47.0
ID#141382 : Information valid since V0.47.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#141375 : solved problem, solved since V0.47.0
Inverse feed rate (G93) by circular interpolation (G2/G3).
Inverse Feed Rate (G93) by circular interpolation (G2/G3) functioned correctly only if the rotation angle was defined.

ID#141372 : solved problem, solved since V0.47.0
NC-program blocks upon CDC activation with G137
Upon CDC activation with G137 in the range of 180 to 360 degrees, the NC-program sometimes blocks.

ID#141367 : solved problem, solved since V0.47.0
CDC: incorrect arc transition after circles programmed with a rotation angle
If the CDC inserts an arc transition after a circle programmed with a rotation angle, then the arc transition has the same rotation angle as the programmed circle. This error exists since version V0.42.8.

ID#141362 : solved problem, solved since V0.47.0
CDC: Full circle instead of arc transition.
If the CDC inserts an arc transition <= 90 degrees after a programmed full circle, then instead of the arc transition a full circle is traversed.

ID#140870 : solved problem, solved since V0.47.0
Full circle is traversed two times
With active CDC it can happen, that a full circle is traversed two times. This error can occur since Version V0.42.8

ID#139937 : solved problem, solved since V0.47.0
Restart: the values of the S and T parameters are incorrect after a restart.

ID#139932 : new function since V0.47.0
Override for G0 (R_override)
For G0 function new "rapid" override "move.R_override" is used instead of the "move.F_override". The allowed value range spans 0 to 10000 (0.00 - 100.00% of the axes limits). If greater value would be inserted by the user, value will be limited to 10000 by ARNC0.

ID#139927 : solved problem, solved since V0.47.0
Feed override was valid also for G0
Parameter "move.F_override" was incorrectly taken into consideration also by G0.

**NC Software - ARNC0 V0.46.1**

ID#140560 : new function since V0.46.1

Inverse Feed Rate, Functions G93, G94.

Function G93 switches the inverse feed rate on. Parameter F defines the inverse of the time (in minutes) needed to complete the NC-block. Function G94 switches the inverse feed rate off.

ID#140377 : solved problem, solved since V0.46.1

Incorrect behavior when disabling CDC with G39

*If a cutter diameter that is too large was detected right before deactivating CDC with active G39, then a point located far from the contour was traversed to while deactivating CDC.*

ID#140187 : solved problem, known since V0.40.0, solved since V0.46.1

Polar coordinate motor not possible.

*In case of polar coordinate motor the decoder initialization have been cancelled with the error 10103 ("PKM - No compensation parameters for Cartesian axis").*

ID#139992 : Information valid since V0.46.1

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#134500 : new function since V0.46.1

New Speed Profile for a Circular Interpolation.

A dynamics on an arc/circle/helix has been changed. A movement with G02/G03 can be faster in general. All axes limits, as well as CNC limits are respected.

ID#139817 : new function since V0.46.0

Maximum number of tool data blocks and tool placement numbers has been raised to 500.

ID#139812 : new function since V0.46.0

G171 - immediate processing of NC blocks.

G171 forces immediate processing of the blocks which previously were buffered.

ID#139737 : solved problem, solved since V0.46.0

Incorrect conversion of ACOPOS parameters from input texts.

In ACOPOS parameter tables and with the "service.data_text" component, values for ACOPOS parameters can be defined in hexadecimal as input text. For ACOPOS parameters of data type "DINT", input texts in the range "0x80000000" to "0xFFFFFFFF" were previously converted to the value "0xFFFFFFFF" by mistake.

ID#139702 : Information valid since V0.46.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

ID#139512 : new function since V0.46.0

G153 - Specifying the positions as absolute coordinates in the machine coordinate system

In rapid feed blocks and interpolation blocks, all coordinate specifications are interpreted as absolute coordinates in the machine coordinate system independent of any transformations or compensation (G54 G59, G92, G119, G159, tool offset and tool length).

**NC Software - ARNC0 V0.45.3**

ID#139602 : new function since V0.45.3

Additional module format for ARNC0 error text modules

Until now, ARNC0 error text modules were managed as standard data modules. Starting now, an additional module format, that enables management of error texts according to version in AutomationStudio, will be supported for error text modules. See also A&P#138900.
Standard priority for background tasks too low

Until now, the standard priority of ARNC0 background tasks was set quite low. This sometimes caused the processing of CNC programs to be interrupted, particularly on target platforms with low processing power or high load. From now on, the standard priority of the background tasks will be increased so that the CNC background tasks have a higher priority than the tasks for online communication and visualization. The previous state (low priority of the background tasks) can be restored by setting the attribute ARNC0SystemConfig="1,10,1" for an NC object with the type ‘ncMANAGER’ in an NC deployment table.

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

NC Software - ARNC0 V0.45.2

Transition angle of very short path section defined as non-tangential.

The transition angle of very short path sections was defined as non-tangential (alpha > s_jump_t) if v_path_mode == 1.

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

NC Software - ARNC0 V0.45.1

In G113, the unit factor k must be absolute in the NC program instead of being specified as a percentage (since ARNC0 V0.44.0).

Tangential transitions on linear axes

New mode of CNC axis (cnc_obj.axis.axis[i].type == 17) has been introduced. If there is a transition between NC-blocks, realized exclusively by axes of such type, then this transition should be considered as tangential.

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.6

NC Software - ARNC0 V0.45.0

In DPR-Trace was the ARNC0 Sampling Time 0 Microseconds

In DPR-Trace was the ARNC0 Sampling Time 0 Microseconds and wrong values by ARNC0MAN-ID and ARNC0SSY-ID.

CDC deactivation with G137 caused erroneous axes movements normal to the main plane

The programmed Z-movement was already done in the transition block instead of in the programmed NC-block after G40. (in ARNC0 V0.44.1 only)

Incorrect values of remaining distance for linear axes.

In CNC system with only linear axes the monitor of remaining distance (monitor.s_ncsatz and monitor.s_ncblock) were not valid.
ID#138552 : Information valid since V0.45.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.4

ID#136072 : solved problem, solved since V0.45.0
Exit movement of CDC deactivation immediately after G40
In combination with the new mode and G137, the deactivation movement is executed immediately after G40.
Activate this behavior by making the following entry in the data structure:
German: "cnc_object.decoder.parameter.wrk.abwahl" = ncAUTO
English: "cnc_object.decoder.parameter.cdc.exit" = ncAUTO
This makes it possible to deactivate CDC immediately before the program is ended.

NC Software - ARNC0 V0.44.1
ID#138072 : solved problem, solved since V0.44.1
With activated CDC the z-axis and the slave axes do not move.
The CDC is active and only the z-axis is programmed in an NC-block. When the z-axis is programmed again in the next NC-block as part of a move in the main plane then it does not move. This error also affects both the linear and tangential slave axes.

ID#136292 : solved problem, solved since V0.44.1
CDC activation with G137 causes erroneous axes movements normal to the main plane
The programmed Z-movement is already done in the transition block instead of in the programmed NC-block. This error affects both the linear and tangential slave axes too.

ID#134987 : Information valid since V0.44.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.4

NC Software - ARNC0 V0.44.0
ID#137502 : Information valid since V0.44.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.4

ID#137497 : new function since V0.44.0
Reduction of path speed at tangential path section transitions depending on the transition angle.
G113 makes it possible to reduce the path speed at tangential path section transitions depending on the transition angle.

ID#137492 : new function since V0.44.0
G108/G109/G110 Set Path Acceleration/Path Deceleration
With the command G108, G109 and G110 the acceleration or deceleration can be programmed as absolute or relative value.

ID#137487 : new function since V0.44.0
G103/G104 (Radius Dependent Feed Adjustment)
Radius dependent feed adjustment can be programmed proportional to the radius or reciprocal proportional to the radius of the path section.

NC Software - ARNC0 V0.43.0
ID#136155 : new function since V0.43.0
Compensation of the mechanical impreciseness of an axis
Compensation of the mechanical impreciseness of the axis (backlash, spindle slope and combination) has been added. Call from an application:
ncaction(nc_obj,ncCOMP,ncSTART), ncaction(nc_obj,ncCOMP,ncSTOP).

ID#135360 : Information valid since V0.43.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.3
Warning 10459 when activating CDC
The warning 10459 is output if the first movement block after CDC activation does not contain a movement in the active plane.

Error deactivating CDC with G137
If the movement block does not contain a path in the active plane immediately before or after CDC deactivation (G40), then an additional movement normal to the active plane is executed in the inserted exit block (G137).

Wrong contour results from G180=0 with active mirroring
Only the rotational direction of the circle is reflected, but not the centerpoint coordinates.

Violation of the Software Ends
Starting with version V0.40.0 the errors 8141 "Position on the Circle > positive SW End" and 8142 "Position on the Circle < negative SW End" could be erroneously reported.

Circle programming with an angle.
The angle of the circle is specified with H. It is possible to program full circles and circles with more than one full revolution.

Only for internal tests

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.3

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.3

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.2
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.2

Access to NC data modules did not function with certain AR versions:
Management for BR modules has been changed in the following AR versions:
- AR for SG4 E2.73 - V2.79
- AR for SG4 starting with F2.85

For this reason, access to the following NC data modules did not function with the AR versions listed above:
- NC Deployment tables
- NC INIT Parameter modules
Position jump during reverse movement of an NC program
During reverse movement of an NC program, a position jump occurred on all axes when entering a path section.

NC Software - ARNC0 V0.42.3

ID#135355: Information valid since V0.42.3
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.0
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.19.0

ID#134882: solved problem, solved since V0.42.3
Standstill after a path section following G60, synchronous M-function or G04
Movement was always stopped at the end of a path section after an accuracy hold G60, after a synchronous M-function or after a dwell time G04 (since ARNC0 V0.42.1).

NC Software - ARNC0 V0.42.2

ID#134647: Information valid since V0.42.2
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.0
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.19.0

ID#134527: new function since V0.42.2
Switching from inches to mm.
G70/G71 can be used to select the unit for the NC program (inch or mm).

ID#134402: solved problem, solved since V0.42.2
Division by zero at G103/G104.
A division by zero is performed if the center point is not completely specified using center point programming for blocks with G103/G104.

NC Software - ARNC0 V0.42.1

ID#134652: Information valid since V0.42.1
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.0
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.19.0

ID#134432: solved problem, known since ARNC0 V0.40.2 bis V0.42.0, solved since V0.42.1
Violation of axis acceleration and axis speed limits at circular blocks
At circular blocks the limits of axis acceleration and axis speed limits can be exceeded. The limits of path speed and path acceleration are taken in consideration in a correct way. (ARNC0 V0.40.2 - V0.42.0)

NC Software - ARNC0 V0.42.0

ID#134657: Information valid since V0.42.0
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.55.0
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.19.0

ID#134262: solved problem, solved since V0.42.0
Contour error with a combination of G40 and G141.
Arcs are replaced by straight sections if G40 and G141 follow each other one right after another.

ID#134257: solved problem, solved since V0.42.0
NC program blocked with a combination of G170 and G141.
In some cases, the NC program might stop being processed if G170 and G141 follow each other one right after another.

ID#134232: solved problem, solved since V0.42.0
NC program is not terminated.
In some cases, the path speed on very short path sections is reduced to the value 0 and the program stops being processed. The NC program remains active, but can be cancelled.

ID#134172 : solved problem, solved since V0.42.0

Limit value violation (axis acceleration) at the path section transition.
The axis acceleration on very short path sections could exceed the defined limit values at the path section transition (path section runtime shorter than the axis jolt filter time).
On very short path sections where the programmed offset is not reached, the maximum path speed is now reduced to the entrance speed in the next path section. This prevents accelerating and braking on the path.
If the path section runtime is shorter than the axis jolt filter time, then multiple path section transitions might be present in the filter, which can cause the axis limit values (acceleration) to be exceeded at the path section transition. The permissible speed jump on the axes is now reduced at these path section transitions.
Activating can be done by setting the variable as it's written below:
German: "cnc_object.grenzwert.satzuebergang" = 1
English: "cnc_object.limit.blocktransition" = 1

ID#120492 : solved problem, solved since V0.42.0

ARNC0 trace buffer size is configurable
The trace size can be changed in the NC configuration "Size of data buffer for cyclic ARNC0 trace" for ARNC0. For the trace data uploading of the different size than the default value of 20kB (0x5000), version 2.5.2.0004 of Automation Studio is required. When the trace data size is changed and Automation Studio is not capable to upload the data, a warning is output to the logbook.

NC Software - ARNC0 V0.41.0

ID#134682 : Information valid since V0.41.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.55.0
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.19.0

ID#133207 : solved problem, solved since V0.41.0
ACOPOS parameter tables, handling the "VersionFrom" attribute
Some ACOPOS parameters can only be used starting with a specific version of the ACOPOS operating system. For such parameters, this minimum version is entered in the module created from an ACOPOS parameter table with the attribute "VersionFrom" when using AS versions V2.5.2.0002 and higher during the Build procedure.
The "VersionFrom" attribute is now evaluated by the ARNC0 software on the PLC when processing ACOPOS parameter tables. The parameter is not transferred to the ACOPOS if the ACOPOS operating system version there is older than the minimum version defined with "VersionFrom".
Transferring such a parameter (e.g. .849 "MOTOR_TAU_THERM") with older versions of the ARNC0 software will cause the response error "1: Invalid parameter ID" and transfer of the ACOPOS parameter table is aborted. This problem can be avoided by disabling this parameter in the ACOPOS parameter table.

ID#126657 : solved problem, solved since V0.41.0
ACOPOS Parameter table: Parameters with more than 6 bytes data are now transferred
Up to now, if Parameters with data length greater than 6 bytes were contained in an ACOPOS Parameter table (e.g. the parameter MOTOR_ORDER_TEXT), then the transfer of this table was aborted with following error:
- 14180:”Error transferring ACOPOS-Parameter “
- Info: "Length of parameter data too large for ACOPOS parameter in XML data"

NC Software - ARNC0 V0.40.3

ID#134677 : Information valid since V0.40.3
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

NC Software - ARNC0 V0.40.2

ID#134672 : Information valid since V0.40.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

ID#133377 : new function since V0.40.2
Automatic tangential axis at straight line path section transition.
With the new constant ncS_SPR_T the behaviour at straight line path section transitions (alpha equal or less than s_sprung_t) can be influenced.
Position jump of tangential axis at G126 + G60
If G60 was used in combination with G126 the tangential axis jumped to a wrong position and back again to the correct one at the path section.

NC Software - ARNC0 V0.40.1
ID#134667 : Information valid since V0.40.1
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

ID#132102 : solved problem, solved since V0.40.1
Tangential axis orients itself on a path section.
If the angle difference at a non-tangential path section transition is less than the parameter s_jump_t and the actual speed jump of the tangential axis is less than the parameter v_jump_t, then the tangential axis now jumps to its new orientation at the start of the path section.

NC Software - ARNC0 V0.40.0
ID#134662 : Information valid since V0.40.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

ID#130892 : new function since V0.40.0
Rotating the coordinate system freely in space.
Programmed zero point offset G92: The coordinate system can be shifted and rotated in space as needed.
Span correction: The coordinate system can be shifted and rotated again in addition to G92.
Correction to the Cartesian coordinate system: A matrix can be defined for correcting any angles that are not exactly 90° in the machine coordinate system.

ID#130867 : solved problem, solved since V0.40.0
Incorrect tangential axis position at G92 transition blocks.
If the coordinate system is rotated and shifted when a tangential axis is switched on and if blocks without a traverse path (e.g.: M-Functions) follow immediately after the shift, then the tangential axis could be set to an incorrect angle in the transition block.

ID#129727 : solved problem, solved since V0.40.0
Position jump at a full circle after a block with G92.
A position jump occurs if a full circle without coordinate specification is programmed after a block with G92.

NC Software - ARNC0 V0.39.3
ID#135547 : Information valid since V0.39.3
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

ID#130317 : solved problem, solved since V0.39.3
Error 9263:"Angle calculation impossible, vector length is 0"
When an NC-program contained very short path sections (e.g. 0.0001 CNC-units), then with activated CDC the error 9263:"Angle calculation impossible, vector length is 0" was reported.

ID#130247 : solved problem, solved since V0.39.3
Path speed is too high for the tangential axis.
At tangential path section transitions the path speed was too high for the tangential axis, if the limit speed of the second path section was higher than the limit speed of the first path section.

ID#130065 : solved problem, solved since V0.39.3
Error when using eight and more ACOPOS on ETHERNET Powerlink
When using eight and more ACOPOS on an ETHERNET Powerlink line, error 14126: "No cyclic positions from drive" was output accidentally.

ID#128132 : solved problem, solved since V0.39.3
Error while determining the RESTART-INFO.

Determining the RESTART-INFO delivers a wrong name of the NC init program (from ARNC0 V0.38.0).

ID#124975 : solved problem, solved since V0.39.3

CNC parameter "halt=nc\_V\_JUMP" does not work properly.
When the CNC parameter "cnc\_obj->limit.halt" was set to "nc\_V\_JUMP", the axes stopped at every non-tangential path section transition.

NC Software - ARNC0 V0.39.2

ID#126357 : Information valid since V0.39.2

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

ID#126137 : solved problem, solved since V0.39.2

ARNC0 V0.39.0 with ACOPOS 8V1xxx.xx-1: Response errors for FFCTRL parameters
With ARNC0 software V0.39.0 after calling the NC actions "ncGLOBAL,nc\_INIT" or "nc\_CONTROLLER,nc\_INIT" all FFCTRL parameters are always transferred to the ACOPOS. For ACOPOS 8V1xxx.xx-1 this causes the following response error for each FFCTRL parameter:
- 1: Invalid parameter ID
Apart from these response errors has this no further effects.

NC Software - ARNC0 V0.39.1

ID#126347 : Information valid since V0.39.1

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

ID#125760 : solved problem, solved since V0.39.1

NC-block-number was not set by SW limit error.
The errors 8137/8138 were set instead of errors 8258/8259.

ID#125570 : solved problem, solved since V0.39.1

CNC program was stopped after G126 with $LIN.
A CNC program was interrupted at the end of a path section after a linearized rounding edges function (G126 + $LIN), depending on the set movement parameters.

NC Software - ARNC0 V0.39.0

ID#124572 : Information valid since V0.39.0

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.18.0

ID#124682 : solved problem, solved since V0.39.0

Full circle together with G180.
If a full circle was programmed together with G180, the full circle was not traversed. The NC program was interrupted and could not be continued at this point.

ID#124635 : solved problem, solved since V0.39.0

Problem during the restart of a NC program
The CNC program cannot be continued at the restart point (since ARNC0 V0.37.0).

ID#124460 : solved problem, solved since V0.39.0

Wrong axis positions in the CNC monitor.
When an NC-program was started after homing, the last CNC positions were displayed for one ARNC0-cycle, instead of the current axes positions.

ID#124377 : new function since V0.39.0

New parameter 'mode' for Cam Profiles
mode == ncSTANDARD: default. The slave axis behaves in the usual manner. i.e. it follows the master axis as closely as the slave’s software end values and acceleration and speed limit values permit it to do.
mode == ncGANTRY: the slave axis follows the master axis without regard to its own software end values and acceleration and speed limit values.

ID#123950 : solved problem, solved since V0.39.0
Position jumps when using G25 or G180 together with G92.
Position jumps occurred on the CNC axes if G25 (tangential arc transitions) or G180 (tangential line-circle programming) was used in a coordinate system rotated with G92.

ID#115792 : new function since V0.39.0
Maximum length for the names of data modules extended to 12 bytes
The maximum length of the data module names is now 12 bytes (was 10).

NC Software - ARNC0 V0.38.0
ID#124372 : Information valid since V0.38.0
Included drive operating systems
For ACOPOS 8V1xxxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxxx.00-2: ACP10SYS V1.18.0

NC Software - ARNC0 V0.37.2
ID#125567 : Information valid since V0.37.2
Included drive operating systems
For ACOPOS 8V1xxxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxxx.00-2: ACP10SYS V1.17.0
ID#124630 : solved problem, solved since V0.37.2
Refresh of monitor for ncAXIS and ncEXTENCOD
The monitor structure nc_obj->monitor of the NC objects ncAXIS and ncEXTENCOD is refreshed in every cycle of the ARNC0 Manager. Until now, the data was refreshed in every second cycle.

NC Software - ARNC0 V0.37.1
ID#123732 : Information valid since V0.37.1
Included drive operating systems
For ACOPOS 8V1xxxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxxx.00-2: ACP10SYS V1.17.0
ID#123727 : new function since V0.37.1
NC object name in NC object information.
The NC Object Name from a deployment table is copied to the NC Object Information structure: nc_obj->nc_obj_inf.nc_obj_name

ID#119675 : solved problem, solved since V0.37.1
Status of ncaccess() wrong
If an NC INIT parameter module was defined in a NC mapping table which was not present on the target, function ncaccess() output ncOK, which was wrong.

NC Software - ARNC0 V0.37.0
ID#122427 : Information valid since V0.37.0
Included drive operating systems
For ACOPOS 8V1xxxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxxx.00-2: ACP10SYS V1.16.4
ID#122257 : solved problem, solved since V0.37.0
Stop after a trigger path section (G201).
No other NC programs can be started if a stop is executed immediately after a trigger path section (G201).
ID#121532 : solved problem, solved since V0.37.0
Position jumps when using multiple interfaces simultaneously in one CNC system.
Position jumps could occur on the Cartesian axes when a stop or accuracy hold is executed if the axes in a CNC system are distributed on multiple interfaces (e.g. Powerlink IF and virtual IF) (since ARNC0 V0.13.0).

NC Software - ARNC0 V0.36.8

ID#121767 : Information valid since V0.36.8

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.16.4

ID#120907 : solved problem, solved since V0.36.8

Error at expanded zero-point offset (G159).
In blocks with G159 (zero-point offset), incorrect values are sometimes used for the zero-point offset due to an error during the index calculation (since ARNC0 V0.26.0).

ID#120720 : solved problem, solved since V0.36.8

NC Program will not be terminated.
In NC programs with very large values for the programmed coordinates, the ends of path sections may sometimes not be detected and therefore the NC program will not be terminated.

ID#120072 : solved problem, solved since V0.36.8

NC program is terminated at the first block with a zero-point offset (G53 - G59, G159).
Occasionally, blocks may no longer be processed after a block with a zero-point offset (G53 - G59, G159). The program is terminated at this block.

ID#119265 : solved problem, solved since V0.36.8

The start of an NC block caused the simulation mode "ncCNCSYS" to be switched off.

NC Software - ARNC0 V0.36.7

ID#121557 : Information valid since V0.36.7

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.16.4

ID#120067 : solved problem, solved since V0.36.7

Axis limit exceeded during circular movement.
The acceleration limits for the circle axes were sometimes exceeded in circular blocks.

ID#119842 : new function since V0.36.7

Compensation movement when changing the tool length and axis offset.
The entire compensation movement is processed at once if multiple NC blocks occur consecutively, causing a change in the tool length or axis offset (G16, G17, G18, G19, G53 to G59, G158 or tool data number). This occurs either in the subsequent movement block (if linear block) or immediately before the next movement block (if circular block).

NC Software - ARNC0 V0.36.6

ID#120917 : Information valid since V0.36.6

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.16.4

ID#119267 : solved problem, solved since V0.36.6

Changing the direction of the tool length compensation while changing the main level.
The direction of the tool length compensation is automatically changed to the new main level when the main level is changed with G17/G18/G19.

Note:
The level switching does not affect the direction of the tool length compensation if G16 was used to define the direction of the tool length compensation in the direction of an axis (e.g. G16 Q+).

NC Software - ARNC0 V0.36.5

ID#118227 : Information valid since V0.36.5
The automatic tangential axis may not reach the target position on the shortest distance.

In NC blocks with G00 it can happen, that the automatic tangential axis may not reach the target position on the shortest distance.

NC Software - ARNC0 V0.36.4

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.16.3

ID#116917 : new function since V0.36.4

Addition to the time signal function
The override to be used for calculating the runtime can be specified using an optional parameter for the G function G221.

ID#116912 : solved problem, solved since V0.36.4

The program is not aborted when a positioning movement is held up at M0/M1.
A positioning movement held up at M0/M1 in the NC program can now be aborted using the NC action "ncMOVE, ncHALT".

ID#116907 : solved problem, solved since V0.36.4

NC action "ncMOVE, ncHALT" not allowed while in the "ncHALT_SST" state.
It is now also possible to apply the NC action "ncMOVE, ncHALT" while in the "ncHALT_SST" state (halt via single step operating mode).

ID#116902 : solved problem, solved since V0.36.4

Positioning not allowed in the NC program while in the "ncHALT_PRG" state.
It is now also possible to start a positioning movement in the NC program while in the "ncHALT_PRG" state (halt via M0/M1).

NC Software - ARNC0 V0.36.3

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.16.3

NC Software - ARNC0 V0.36.2

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.16.2

NC Software - ARNC0 V0.36.1

Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.7

ID#114447 : solved problem, solved since V0.36.1

Error message in response to correct time signal output
When the path section was programmed with the signal function G220 immediately after a G221, the error 7178:“Signal time longer than path section runtime”, occurred even though the time signal was output correctly.

ID#114407 : solved problem, solved since V0.36.1

Error in edge detection for trigger2 by using the force function.
When forcing Trigger2, positive edges were detected as negative edges.

ID#114402 : new function since V0.36.1
New G function: G12
G12 makes it possible to influence the override on the transition arc inserted by the CDC.

ID#114012 : solved problem, solved since V0.36.1
Incorrect shift of coordinate system with G92 for linear axes
The calculation for shifting the coordinate system with G92 was made incorrectly for linear axes (ARNC0 versions 0.345 to 0.360).

NC Software - ARNC0 V0.36.0
ID#114757 : Information valid since V0.36.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.7

ID#113837 : new function since V0.36.0
ARNC0 sampling time not dependent on TC1
Starting with AutomationRuntime V2.80, it is possible to set the ARNC0 sampling time independently from the cycle time of TC 1 when using ETHERNET Powerlink. However, the ARNC0 sampling time must still be a multiple of the ETHERNET Powerlink cycle time and the ACOPOS position controller cycle (400µs).

NC Software - ARNC0 V0.35.0
ID#113832 : Information valid since V0.35.0
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.7

ID#112712 : solved problem, solved since V0.35.0
No reverse movement at G170 or trigger event delay.
If a G170 or a trigger event delay ($IF <axis><edge>.EV) was moved in the reverse direction, the CNC system could come to a halt, depending on the course of the contour. This error was present starting with version V0.32.0.

ID#112707 : solved problem, solved since V0.35.0
Reverse movement not possible on path sections with G201.
The CNC system came to a halt if reverse movement was made on a path section with G201.

ID#112702 : solved problem, solved since V0.35.0
CNC monitor not initialized.
The previous values were still displayed in the CNC monitor after starting an NC program. This caused problems if a program halt already occurred at the beginning of an NC program (e.g. M0/M1, G170 or override at zero). The fields are initialized as follows:
NC block monitor: The s_ncblock field is zero.
CNC monitor: The fields s_ncblock, v_path, pos_ncprog, block_ncprog, t_ncprog, s_ncprog and v_ncprog all contain zero. The fields name_ncprog and nr_ncprog contain the name and program number of the current NC program.

ID#112697 : solved problem, solved since V0.35.0
CAN synchronization PP2xx
On PP2xx, large jitter in the system timing could cause generated set positions to be transferred to the ACOPOS at the wrong time (too early). The error 104007:"Lag error stop limit exceeded" or 105024:"Cyclic set value mode aborted: Set positions missing" could be triggered if large interruptions occur.

ID#110785 : solved problem, solved since V0.35.0
CNC halt after program was aborted.
The CNC system came to a halt if an NC program was aborted in the next PLC cycle after startup. "move.status.cnt_ncprog" remained at 1. The error state could only be exited by resetting the controller.

NC Software - ARNC0 V0.34.5
ID#113827 : Information valid since V0.34.5
Included drive operating systems
ID#110847 : solved problem, solved since V0.34.5

Position error after G92 and active tool compensation (tool offset or tool length compensation)

Starting with ARNC0 0.260: The target position was not calculated correctly in the automatically created transition block after G92 and during active tool offset or tool length compensation.

Up to ARNC0 0.250: The target position was not calculated correctly in the relatively programmed transition blocks after G92 and during active tool offset or tool length compensation.

NC Software - ARNC0 V0.34.4

ID#113822 : Information valid since V0.34.4

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.6

ID#110677 : solved problem, solved since V0.34.4

After some restart errors, cnt_ncprog remains at 1.

During restart, if the NC program does not match the saved restart information, an appropriate error message appeared, but the system was not restarted, and the restart was not aborted.

ID#110672 : solved problem, solved since V0.34.4

Not an error message if multiple signals of the same type are contained in the same path section.

In certain situations, time or path entries require multiple G220 or G222 signals be output in a single path section. However, since only one signal of each type can be output at a time, an error message appears. The previously entered signal is retained.

ID#110667 : solved problem, solved since V0.34.4

G222 signal is ignored during restart.

Reset occurred in a path section with G222, but the G222 signal was not output.

ID#110662 : solved problem, solved since V0.34.4

Incorrect G222 signal output during path positioning.

The G222 signal was only output one time during positioning within the path section. When the path section was entered backwards, the G222 signal was immediately output forwards, and the change in direction occurred between the correct signal position and the end of the path section.

ID#110657 : solved problem, solved since V0.34.4

G222 signal output in the wrong position.

The length of a path section was not a whole number, and the G222 signal was output from the wrong position.

ID#110652 : solved problem, solved since V0.34.4

CNC system deadlock during G220 / G222.

Multiple path sections with G220 or G222 followed directly after one another and no G221 with synchronous M function was found, causing the CNC system to halt.

ID#110647 : solved problem, solved since V0.34.4

CNC system deadlock during G221.

A G221 with synchronous M function was run in an NC program, and the NC program finished before a path section was completed with G220 or G222, causing the CNC system to halt at G221.

ID#110642 : solved problem, solved since V0.34.4

Error message appears without cause for signal functions.

One of the following error messages appeared during the G220 and G222 signal functions, which overlap different path sections, although sufficient runtime and path distance were available: 7177:”Remaining path distance for signal is too long”, 7178:”Signal time longer than path section runtime”, or 8228: ”Path command with undefined length or runtime.”

NC Software - ARNC0 V0.34.3

ID#109112 : Information valid since V0.34.3

Included drive operating systems
ID#106350 : solved problem, solved since V0.34.3
  G126 with G141 together in an NC block
  If G126 and G141 were together used, it could occur that the NC program did not terminate.

**NC Software - ARNC0 V0.34.2**

ID#108477 : solved problem, solved since V0.34.2
  Signal function G220 and path jolt time > 0.
  The signal function G220, which overlaps different path sections, registers error 7163: "Time is longer than section run time" if a path jolt time was set > 0. This error mostly occurred in NC programs with short path sections.

ID#108472 : solved problem, solved since V0.34.2
  Signal function G220 and override.
  The override was not correctly calculated during the signal function G220, which overlaps different path sections.

ID#108467 : new function since V0.34.2
  Dwell time 3 NC cycles too long.
  The delay was 3 NC cycles longer than the programmed dwell time.

ID#108147 : Information valid since V0.34.2
  Included drive operating systems
  For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
  For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.2

ID#107017 : solved problem, solved since V0.34.2
  Invalid Job ID at ARNC0 response
  It was possible that error 40137: "Internal Error - Invalid Job ID in ARNC0 response" occurred after the completion of a CNC program and after the controller of an axis used in the CNC channel was switched off.

**NC Software - ARNC0 V0.34.1**

ID#107492 : Information valid since V0.34.1
  Included drive operating systems
  For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.8
  For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.2

ID#105257 : solved problem, solved since V0.34.1
  Global NC subprograms are not terminated correctly
  A global NC subprogram could not be correctly terminated if it was not completed with M02, M29 or M30 or if more than three axes were used. In this case, the CNC system would freeze.

ID#103052 : new function since V0.34.1
  Init ACOPOS parameter tables
  The transfer of parameters from an Init Acopos parameter table (from NC mapping table) to the drive is now also performed when parameters for this drive are present in the NC configuration.

**NC Software - ARNC0 V0.34.0**

ID#102615 : solved problem, solved since V0.34.0
  CNC program cannot be aborted if the override is <= 0 when the program is started
  If the override was negative or zero at the start of an NC program, then the NC program could no longer be aborted.

ID#106487 : Information valid since V0.34.0
  Included drive operating systems
  For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.7
  For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.1
Signal functions G221 and G222

The signal function G220, together with G221, outputs a signal to the PLC at a specific time before the target is reached. The signal function G222 outputs a signal to the PLC at a specific remaining path distance before the target is reached. Both signal functions overlap different path sections.

"Divide error" after $TIME or G04

If a dwell time ($TIME bzw. G04) was used in a NC programm when the CNC cycle time was < 1 ms, the system error 25300: "Divide error" (division by 0) occurred.

Position error after rotation of coordinate system without shift offset (G92)

If the first block of a NC programm contained G92 with rotation but without offset, an additional offset (depending on previous NC programs), was included to the coordinate system.

Incorrect rotation direction of the automatic tangential axis after G92

After a block with G92, the automatic tangential axis may not be aligned to the new position via the shortest distance.

NC program can get held up if the sign for the movement override is changed

An NC program could get held up if a positive movement override and than a negative movement override was set several times in the NC program.

NC Software - ARNC0 V0.33.0

Corrected drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.7
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.1

Wrong INIT states after switching off the simulation mode "ncCNCSYS"

If one of the following NC actions
  - ncCONTROLLER,ncINIT
  - ncDIG_IN,ncINIT
  - ncENCODER_IF,ncINIT
  - nclLIMITS,ncINIT
  - ncGLOBAL,ncINIT
was called in the simulation mode "ncCNCSYS" or "ncCNCSYS+ncDRIVE", the corresponding INIT states ("controller.init", "dig_in.init", "encoder_if.init", "limit.init" or "global.init") should be set to "ncFALSE" after switching off the simulation mode "ncCNCSYS". Up to now the values of this INIT states could falsely remain on "ncTRUE".

New NC structure component "dig_in.force"

New NC structure component "dig_in.force"

Transfer errors during initialization of digital inputs blocked further commands

If an error occurs when transferring a parameter to the ACOPOS during initializing of digital inputs, error 14170: "Error initialize digital Inputs" was indicated.
Up to now, in this case the execution of nearly all further commands was blocked. This problem could be eliminated only with a restart of the PLC-CPU.

Force function for digital Inputs

Digital inputs can now be set with an application using the NC action "ncDIG_IN,ncFORCE".

NC Software - ARNC0 V0.32.4

Information valid since V0.32.4
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.6
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.0

ID#103692 : solved problem, solved since V0.32.4
Tool radius from the tool table sometimes was not used (only in V0.27.0 - 0.323)
If for a tool in the tool table a length or an offset not equal to zero were defined, then the tool radius defined in the tool table was not used.

NC Software - ARNC0 V0.32.3
ID#103257 : Information valid since V0.32.3
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.6
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.0

ID#102907 : new function since V0.32.3
Inconsistent referenced status
An inconsistency between the user status "move.homing.status.ok" and the corresponding status on the drive could occur after a change in the simulation mode (ncDRIVE) of an axis (ncSIMULATION, ncSWITCH_ON/ncSWITCH_OFF).

NC Software - ARNC0 V0.32.2
ID#87847 : solved problem, solved since V0.32.2
Incorrect target position after backwards positioning before a G170.
An incorrect target position was calculated on the path during backwards positioning if a stop was executed after the last path section before a G170 using a halt command. Backwards positioning first started when the decoder was resumed using the SYNC command. After error correction, the positioning now starts immediately and reaches the correct target.

ID#102552 : solved problem, solved since V0.32.2
Error processing structure blocks such as $IF, etc.
If parts of structure blocks were skipped over (e.g. the section from $ELSE to $ENDIF), $ functions contained in the structure block such as $TIME, $RAD etc. were evaluated instead of being ignored.

ID#102547 : new function since V0.32.2
EnDat parameters transferred to ACOPOS always work
From now on, parameters will be transferred from ARNC0 to the ACOPOS as soon as reading motor parameters from EnDat encoders has been completed.
Until now, a motor parameter could be already transferred the ACOPOS, before reading of the motor parameters of the corresponding EnDat encoder has been completed. In this case the value which was read from the encoder worked and not the value transferred from PLC to the ACOPOS.
From now on, always that value works, which is transferred from PLC to the ACOPOS.

ID#102542 : Information valid since V0.32.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.6
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.15.0

ID#102532 : solved problem, solved since V0.32.2
Cycle time violation TC1 during the initialization of a PV interface
A cycle time violation could occur in TC 1 during a PV interface initialization.

ID#102047 : new function since V0.32.2
Change to the user status, "controller.ready"
The user status, "controller.ready" is always set to ncTRUE for drives in the simulation mode ncCNC_SYS or ncCNC_SYS+ncDRIVE as well as drives on a virtual interface.

ID#101897 : solved problem, solved since V0.32.2
CNC standstill after consecutive NC blocks with M parameters.
Writing more than 10 NC blocks with M parameters immediately following each other causes a CNC standstill. This error also occurred if more than 10 NC blocks immediately followed each other, which just contained non-synchronous M functions, S or T functions.
Command Acceptance (only in Version 0.32.0)

If a drive that has been successfully initialized (“network.init==ncTRUE”), it is possible that service commands (ncSERVICE, ncSET/ncREAD) are not accepted and error 1220 "Command not accepted network not ready" is given. The same could also occur when downloading ACOPOS parameter table (via service interface) (ncACP_PAR+ncSERVICE, ncDOWNLOAD).

NC Software - ARNC0 V0.32.0

ID#100712 : new function since V0.32.0

New NC structure component "network.nc_sys_restart"

ID#100707 : Information valid since V0.32.0

Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.6
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.14.0

ID#100677 : solved problem, solved since V0.32.0

After stopping an NC program, the CNC system sometimes gets stuck.

If an NC program is stopped shortly before a G170, it is possible that the axes stopped but the number of active NC programs gets stuck at 1. Then no other NC programs can be started.

ID#100672 : solved problem, solved since V0.32.0

Path speed too high, if a G170 is moved over in the backwards direction.

If v_jump > 0 was configured and a short path section and a G170 follow after a contour corner, then braking does not occur before this contour corner when moving backwards.

ID#100627 : solved problem, solved since V0.32.0

Error when initializing the external encoder

When initializing an external encoder with the NC action "ncENCODER_IF, ncINIT", the count direction was not transferred to the drive correctly. The count direction of the motor encoder was transferred, which is incorrect.

ID#100557 : solved problem, solved since V0.32.0

Error when transferring parameters for controller initialization

During controller initialization (ncCONTROLLER,ncINIT or ncGLOBAL,ncINIT), the parameters PARID_SCTRL_TI_FIL (#283 - Speed Filter) and PARID_CONTROLLER_MODE (#328 - Controller Mode) were not transferred under the following circumstances:
1. ncSIMULATION,ncON (ncCNCSYS mode)
2. ncGLOBAL,ncINIT => Parameters not transferred to drive (because of simulation mode "ncCNCSYS")
3. ncSIMULATION,ncOFF
4. ncGLOBAL,ncINIT => The two parameters are not transferred to the drive!

ID#100547 : solved problem, solved since V0.32.0

G92 + G170, Decoder stopped a movement block too early.

If a G170 block immediately followed G92 with a transition block without traverse path, then the decoder stopped a movement block too early.
(Starting with ARNC0 0.310, transition blocks without traverse paths are automatically inserted after G92, G54-59 and G159.)

ID#100357 : solved problem, solved since V0.32.0

Position jump when network communication drops out

If communication dropped out during cam profile coupling on a slave axis, then a jump in speed could occur. In this case, the internal deceleration ramp uses the speed resulting from this jump, which is incorrect (the data is no longer transferred to the drive because of the communication failure).

ID#100217 : solved problem, solved since V0.32.0

Position latch function

A position latch function activated with the NC action "ncLATCH1(2), ncSW_ON" was not completely deactivated with the NC action "ncLATCH1(2), ncSW_OFF". Then it was not possible to activate the other respective position latch function.
ACOPOS CAN synchronization

If systematic deviations occurred between the system time and CAN time (different timers), ACOPOS sometimes reported the error 106002: "Sync Controller: Error tolerance of system time difference exceeded". Starting now, only the timer from the corresponding CAN interface is used for CAN synchronization.

Position error at the end of a program without first deactivating an automatic tangential axis:

If an NC program with active automatic tangential axis (G141) is ended without first deactivating the tangential axis (G140) and if the last NC block is a rapid feed block (G00), the tangential axis in this block will travel to the position it had before the activation (G141).

Automatic transfer of motor and encoder parameters during ACOPOS startup:

In the past, encoder parameters for ACOPOS (AC122, AC123) were not automatically transferred to the drive via ETHERNET Powerlink upon drive startup. This problem occurred for CAN starting in version V0.24.0.

No compensation movement at NC blocks with tool data number or tool orientation:

The tool compensation (tool offset or tool length) is not taken into consideration if a block without programmed traverse path (e.g.: only M-functions or only G91) immediately follows an NC block with a tool data number (Dxx) or G16. This error only occurs during relative programming.

(since V0.27.0)

Change of CAN response timeout

During the start up of an ACOPOS with AC140 the error 14112: "Timeout for parameter ID" could be output. The extended CAN response timeout only works in ARNC0 V0.29.2, but not in V0.30.0 - V0.31.1.

Included drive operating systems

For ACOPOS 8V1xxx.xx-01-1: ACP10SYS V0.54.6
For ACOPOS 8V1xxx.xx-02-2: ACP10SYS V1.13.1

ACOPOS Parameter Tables: Disabling parameter records

Parameter records in an ACOPOS parameter table can now be disabled using the XML data "Disabled='TRUE'". In the editor for ACOPOS parameter tables disabling parameters is possible starting with V2.4.0.1106.

NC Deployment Tables: Disabling NC object blocks

NC object blocks in a NC deployment table can now be disabled using the XML data "Disabled='TRUE'". With Automation Studio V2.x this data can only be entered in the editor for NC deployment tables in the "Additional Data" column.

The data structure of the NC configuration for the virtual interface was extended with "pv_name".

Included drive operating systems

For ACOPOS 8V1xxx.xx-00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xxx.xx-00-2: ACP10SYS V1.13.0
ID#94162 : Information valid since V0.29.1
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.13.0

ID#94157 : solved problem, solved since V0.29.1
CNC program aborted at circle-circle transitions if tool radius nearly equal to programmed radius
If the tool radius used was equal to the programmed radius of circle segments, the CNC program could have been aborted at circle-circle transitions with error 9221:"Excentric Circles".

ID#93927 : solved problem, solved since V0.29.1
NC program freezes. 'Illegal float value' is displayed in the CNC monitor as position, remaining distance and speed.
If circle programming in the NC program is inaccurate, NaN or -NaN could occur as result of an internal angle calculation. This would then also cause the target position of the NC block to be NaN or -NaN.

ID#93902 : new function since V0.29.1
G90/G91 with G16, G53 up to G59, G92, G159 and Dxx allowed in an NC block.

ID#93692 : new function since V0.29.0
New NC structure component "limit.parameter.a_stop"

ID#93687 : new function since V0.29.0
New NC structure component "controller.uf" for controller mode "U/f Control"

ID#93682 : Information valid since V0.29.0
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.13.0

ID#78992 : new function since V0.29.0
New NC structure component "controller.mode" for parameter CONTROLLER_MODE
Up to now, the parameter CONTROLLER_MODE could only be transferred to an ACOPOS via service interface. Therefore due to compatibility reasons, after call of NC action "ncCONTROLLER,ncINIT" this parameter is only transferred to the ACOPOS, if the value of "controller.mode" was changed.

ID#92062 : Information valid since V0.28.0
Included drive operating systems
For ACOPOS 8V1xx.xx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xx.xx.00-2: ACP10SYS V1.12.0

ID#91887 : new function since V0.28.0
‘External encoder’ as coupling master:
Starting now, ‘ncEXTENCOD’ NC objects can be used as master for cam profile couplings. The slave is not compensated for a master position jump (e.g. after referencing the external encoder).

ID#91712 : new function since V0.28.0
New NC structure component "controller.speed.t_filter" for parameter SCTRL_TI_FIL
Up to now, the parameter SCTRL_TI_FIL could be transferred to an ACOPOS via service interface only. Therefore due to compatibility reasons, after call of NC action "ncCONTROLLER,ncINIT" this parameter is transferred only, if the value of "controller.speed.t_filter" was changed.

ID#91707 : new function since V0.28.0
New status "ok" for download of ACOPOS Parameter data
If processing of NC action "ncACP_PAR,ncSERVICE+ncDOWNLOAD" has been successfully completed, from now on additionally "status.ok=ncTRUE" is set after successful completion of operation, because ACOPOS Parameter data without any parameter to be transferred do not cause an error and after successful completion of operation "status.daten_len=0" is displayed as well as "status.error=ncFALSE".
ACOPOS startup synchronization between ACP10 and ARNC0

If the same ETHERNET Powerlink interface is used to operate ACOPOS modules simultaneously from the ACP10 software and the ARNC0 software, then within the basis initialization the network initialization (ACOPOS startup) is executed synchronized between the ACP10 software (from V1.12.0 on) and the ARNC0 software. This now makes it possible to alternately connect ACOPOS modules for ACP10 software and ARNC0 software within one Powerlink line. Up to now, using a common Powerlink interface could lead to errors, whereby an unsynchronized ACOPOS reset command triggered by one of the NC software programs interrupted the network communication of the other NC software program.

NC Software - ARNC0 V0.27.3

ID#92057 : Information valid since V0.27.3
- Included drive operating systems
  - For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
  - For ACOPOS 8V1xxx.00-2: ACP10SYS V1.10.6

ID#91992 : solved problem, solved since V0.27.3
- Position of the automatic tangential axis after deactivation (G140)
  - When deactivating the automatic tangential axis (G140), the axis was moved to the position, which it had upon activation (G141). This could cause large traverse paths for 'non periodic' tangential axes.

ID#91987 : solved problem, solved since V0.27.3
- Position error when restarting an NC program with automatic tangential axis.
  - Error 5152: "Position out of 'In-Position-Tolerance' at START/RESTART of CNC move" could occur while restarting an NC program with an automatic tangential axis. Requirements:
    - Activation of the automatic tangential axis before the first movement block in the NC program
    - Another NC program must be executed between the abort and the restart.

NC Software - ARNC0 V0.27.2

ID#92052 : Information valid since V0.27.2
- Included drive operating systems
  - For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
  - For ACOPOS 8V1xxx.00-2: ACP10SYS V1.10.6

ID#91582 : new function since V0.27.2
- Extended possible combination for S- and T-function.
  - S and T functions are allowed in NC blocks with G16, G53-G59, G92, G159 or tool data number.

ID#91562 : solved problem, solved since V0.27.2
- Drive start up after change of simulation mode.
  - If the start up of an ACOPOS was interrupted by switching on the ncCNCSYS simulation mode, the start up could not be completed after switching off the simulation mode.

NC Software - ARNC0 V0.27.1

ID#91662 : Information valid since V0.27.1
- Included drive operating systems
  - For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
  - For ACOPOS 8V1xxx.00-2: ACP10SYS V1.10.6

ID#91552 : solved problem, solved since V0.27.1
- NC operating system download
  - If a drive had no NC operating system, the startup of this drive was aborted and an NC operating system download could not be executed.

ID#91522 : solved problem, solved since V0.27.1
- ACOPOS operating system start abort.
  - In certain ACOPOS firmware versions (Boot loader), the operating system start was aborted with the error 100018: "The Data segment at data block read is not yet the last".
Contour violation at circular-linear transitions when CDC is active.

A contour violation sometimes occurs at circular-linear transitions with 360° or 0° when the CDC is active.

NC Software - ARNC0 V0.27.0

ID#91242 : new function since V0.27.0
Selection of CNC after transition block

It is possible to select the CDC (G41, G42) after a transition block (G16, G53-G59, G92, G159, Dxx).

ID#91012 : new function since V0.27.0
Function expansion of Stop / E-Stop.

Using two new parameters in the user data structure ARNC0CNC_typ cnc_obj.move.stop and cnc_obj.move.e_stop, the behaviour of stopping and emergency stopping of NC programs can be defined.

ID#90852 : new function since V0.27.0
Function expansion of the tool compensation (G16 and G130/G131)

Orientation of the tool length correction can be selected using G16. The tool length correction and the tool offset are taken into consideration in the machine coordinate system (G130) or in the programmed coordinate system (G131).

ID#90412 : Information valid since V0.27.0
Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.10.6

ID#84922 : new function since V0.27.0
NC deployment table for ARNC0

An NC deployment table can be evaluated for ARNC0. Version 2.4.0 with upgrade 1105 or higher is required to support this function.
ATTENTION: If the Upgrade was not installed, the new data structure element cnc_obj.axes.axis[...].nc_object_name must be set to an empty string (**)

NC Software - ARNC0 V0.26.0

ID#89272 : Information valid since V0.26.0
Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.10.5

ID#89117 : new function since V0.26.0
Behavior of non-programmed axes with G92

The behavior of non-programmed axes in the transition block after G92 has been changed. See "ARNC0: CNC Programming Instructions, G-Functions, G92".

NC Software - ARNC0 V0.25.0

ID#87812 : solved problem, solved since V0.25.0
Position jump after G92 during reverse movement

If another block with G92 immediately follows a transition block (G92) during reverse movement, then a position jump occurs.

ID#87677 : Information valid since V0.25.0
Included drive operating systems

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.10.2

ID#87647 : solved problem, solved since V0.25.0
NC program aborted at G92 + G141

If the automatic tangential axis (G141) was used together with the coordinate system shift (G92), the NC program was aborted with error 10368:"G141 Automatic tangential axis cannot be programmed" (starting with V0.24.0).

ID#87642 : new function since V0.25.0
Mirroring allowed in combination with G92
The coordinate system can be shifted using G92 during active mirroring (G21, G22, G23). However, a change in angle is not allowed during active mirroring.

ID#86467 : solved problem, solved since V0.25.0
Wrong transition block at G92 + G141
Wrong position of the automatic tangential axis in the transition block (G92).

ID#86462 : solved problem, solved since V0.25.0
Error changing the parameter "limit.block_buffer".
If the value of the "limit.block_buffer" parameter was changed, then the next following NC program started remained inactive (not started) (starting with V0.20.8).

ID#86457 : solved problem, solved since V0.25.0
Error using $LIN with G126
If G126 with $LIN was activated between two straight lines, $LIN first takes effect starting at the second rounding path section. If G126 with active $LIN was deactivated in a straight line, $LIN is only effective until the second-to-last rounding path section. As a result, both cases resulted in an axis jump and path speed jump which further caused a violation of the axis and path acceleration limits.

ID#82222 : new function since V0.25.0
CNC program restart: Saving of decoder synchronous parameters for restart
Decoder synchronous parameters (R, P, EX parameter and cutter diameter at $RAD_IDX) can be saved during the processing of an NC program.
When a restart is executed, the saved parameters are used up to the restart point. This guarantees that the desired restart point is reached even if the parameters from the application have been changed.

NC Software - ARNC0 V0.24.3
ID#86097 : new function since V0.24.3
CAN drive startup
During drive startup on the CAN bus, a delay for checking the boot status (2 or 3 seconds) was added after Phase 20: "BsLoader start after SW reset" and Phase 50: "ACOPOS operating system start".

ID#86092 : solved problem, solved since V0.24.3
CDC: incorrect 0/360° distinction at circle/circle transitions
At circle-circle transitions with interior contact and opposite direction of rotation, the distinction between the transition angles 0° and 360° was made incorrectly.

ID#86087 : Information valid since V0.24.3
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.9

ID#86082 : solved problem, solved since V0.24.3
Incorrect CDC calculation at circle/circle transitions with transition angle near 180°
At a transition angle equal to 180°, the circle intersections were calculated correctly, but the wrong intersection was selected.

NC Software - ARNC0 V0.24.2
ID#85407 : Information valid since V0.24.2
Included drive operating systems
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.9

ID#84947 : solved problem, solved since V0.24.2
Abort of an CNC programm at the end of a path section not possible.
Abort of an CNC programm at the end of a path section not possible.

NC Software - ARNC0 V0.24.1
ID#84927 : Information valid since V0.24.1
Included drive operating systems
Für ACOPOS BV1xxx.00-1: ACP10SYS V0.54.3
Für ACOPOS BV1xxx.00-2: ACP10SYS V1.06.9

ID#84402 : solved problem, solved since V0.24.1
System blocked at blocks with G126
At blocks with G126, the ARNC0 system can become blocked when there are low values for speed or acceleration.

ID#83882 : solved problem, solved since V0.24.1
Error rotating the coordinate system (G92)
A position jump occurs in the transition block if a block only with commentary follows immediately after G92.

ID#83877 : solved problem, solved since V0.24.1
Error accepting the CDC selection from the Init parameter module
If "CDC selection = ncBLOCK" (G137) is set in the Init parameter module, the decoder initialization is aborted with the error 10105: "Incorrect Parameter ". CDC selection "ncDIRECT" and "ncINDIRECT" both function.

ID#83797 : solved problem, solved since V0.24.1
Incorrect monitor data for the current actual position of an axis
When using an external encoder, sometimes the actual position of an axis was displayed incorrectly in the monitor data. This problem only occurred on axes connected with ETHERNET Powerlink.

ID#83382 : solved problem, solved since V0.24.1
Reverse movement in NC programs with G92
In the first movement block (transition block) after G92, sometimes the axis positions jumped during the reverse movement.

ID#83377 : solved problem, solved since V0.24.1
G92 with rotation of the coordinate system
If all axes used in the system are not programmed after G92 in the first movement block (transition block), sometimes the target position of the transition block is calculated incorrectly.

ID#81692 : solved problem, solved since V0.24.1
Incorrect transfer of ACOPOS parameters
Parameter values, which were only valid in ACP10sys - versions 1.xxx, were not correctly transferred to the ACOPOS with the NC action "ncSERVICE, ncSET+ncDATA_TEXT". Instead of sending an error message, the last value transferred was used for the new parameter. Starting now, when a parameter ID is defined which is not contained in the ACOPOS operating system version being used, error 14187 (CAN) or 15187 (Powerlink) is output: "Invalid parameter ID for the ACOPOS operating system version being used".

ID#78932 : solved problem, solved since V0.24.1
Dip in the path speed when using the G170
If a G170 was used in a CNC program, the path speed was reduced at the corresponding transitions, even though the decoder was synchronized in time. This problem only occurred when the deceleration distance before the G170 was longer than the path section before the G170.

NC Software - ARNC0 V0.24.0
ID#83362 : Information valid since V0.24.0
Included drive operating systems
Für ACOPOS BV1xxx.00-1: ACP10SYS V0.54.3
Für ACOPOS BV1xxx.00-2: ACP10SYS V1.06.8

ID#83357 : solved problem, solved since V0.24.0
Error when restarting an NC program
If the axes were positioned to the restart point using a positioning command, the saved restart information was sometimes overwritten.

ID#82310 : solved problem, solved since V0.24.0
NC program cannot be resumed after G170
If an NC program was interrupted with movement HALT when the axes arrived at a G170, the NC program could no longer be resumed.

ID#82217 : new function since V0.24.0
Restart function update
- Restart of CNC programs started with offset
- Restart point at preset path position or at preset block number
- Restart point can lie anywhere before or after the stop point

ID#82207 : new function since V0.24.0
Starting CNC programs at preset block number "Nxxxx"
The start position for a NC program can be defined in bytes as file offset or as block number.

ID#82202 : new function since V0.24.0
Extended value range for "F_override"
The value range for "F_override" has been extended and is now 0 to 42,949,672.95%. The data type of the corresponding structure components was increased from UINT to UDINT. This makes it possible to implement a 'fast mode' for CNC systems. However, the maximum possible feed rate is limited by the preset axis limit value.

ID#81180 : solved problem, solved since V0.24.0
CNC parameter 'limit.elements'
As of now, the CNC parameter 'limit.elements' is being evaluated again. It was not being evaluated after version V0.13.2.

ID#80107 : new function since V0.24.0
Maximum number of CNC channels is now 8
AutomationStudio 2.4.0 with Upgrade V2.4.0.1103 or higher is required.

ID#78607 : solved problem, solved since V0.24.0
Dip in the path speed when feed override >100%
If a feed override "F_override" >100% was used, the path speed was reduced to the programmed feed at the transition between two NC blocks. The new operating mode G112 makes it possible to prevent the path speed from reducing at block transitions.

NC Software - ARNC0 V0.23.5
ID#81622 : solved problem, solved since V0.23.5
Error when switching on the controller
Sometimes the controller was switched off by the ARNC0 without an error message immediately after being switched on. This was a very rare error.

ID#81452 : Information valid since V0.23.5
Included drive operating systems
Für ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
Für ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.6

NC Software - ARNC0 V0.23.4
ID#81447 : Information valid since V0.23.4
Included drive operating systems
Für ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
Für ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.6

NC Software - ARNC0 V0.23.3
ID#80952 : Information valid since V0.23.3
Included drive operating systems
Für ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
Für ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.6

ID#80922 : solved problem, solved since V0.23.3
Incorrect assignment of error and NC action (command)
A JobID for the NC action "ncCONTROLLER, ncSWITCH_ON" was acknowledged immediately after the command was accepted. If an error then occurred on the drive (e.g. due to activation of the quickstop input), the reported errors were entered with this JobID. Because this JobID was already acknowledged, it was possible that it was already occupied by another command. As a result, the reported errors would be transferred to the wrong command. If positioning was active, the "move.basis.status.in_pos" would not be set to "ncWAHR" after the movement was successfully completed.
NC Software - ARNC0 V0.23.2

ID#80927 : Information valid since V0.23.2

Included drive operating systems

Für ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
Für ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.6

ID#80215 : new function since V0.23.2

Additional information was added to the individual error texts using the block number.

ID#80170 : new function since V0.23.2

Function expansion G126
- $LIN is used so that the rounding radius is traversed at constant path speed.
- The entrance speed to the rounding radius can be increased using $VE <factor>.
This function can cause the axis acceleration limits to be exceeded.

NC Software - ARNC0 V0.23.1

ID#80917 : Information valid since V0.23.1

Included drive operating systems

Für ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.3
Für ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.6

NC Software - ARNC0 V0.23.0

ID#80435 : Information valid since V0.23.0

Included drive operating systems:

Für ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
Für ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.6

ID#80225 : solved problem, solved since V0.23.0

Full circle when radius programming

When a very small arc was programmed using radius programming (dx and dy smaller than 0.001 CNC units), a full circle was traversed instead of the arc.

ID#80067 : solved problem, solved since V0.23.0

Unwarranted message 8154: "NC block with distance equal to 0.0" at short path sections

If several path sections with the length 0.001 CNC units were traversed consecutively, the message 8154: "NC block with distance equal to 0.0" was output.

ID#80007 : new function since V0.23.0

CNC System: Initialization subprogram

Starting now, the name of a CNC program which is called each time the program starts, can be specified in the CNC data structure "cnc_object.bewegung.ncprogram.init_prg" (German) or "cnc_object.move.ncprogram.init_prg" (English). When a CNC program is started using the NC action "nPROGRAM, nSTART" the CNC program specified in the data structure is first executed as global subprogram.

ID#80002 : solved problem, solved since V0.23.0

Movement reversal on the circle when CDC is active

When there was a full circle made up of several circle segments, a straight line was inserted as segment counter to the circular direction.

NC Software - ARNC0 V0.22.8

ID#80430 : Information valid since V0.22.8

Included drive operating systems:

Für ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
Für ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.6

NC Software - ARNC0 V0.22.7

ID#80425 : Information valid since V0.22.7

Included drive operating systems:

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2
NC Software - ARNC0 V0.22.6

ID#80420 : Information valid since V0.22.6

Included drive operating systems:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2

ID#80092 : solved problem, solved since V0.22.6

Unsteadiness in the path speed:
If Cartesian axes were programmed together with linear axes in a block, unsteadiness could occur in the path speed at a path section transition (spikes with the length of a scanning cycle).

ID#80087 : solved problem, solved since V0.22.6

Adjust v_jump, a_jump parameters (G105 / G106):
In certain circumstances, the parameters v_jump and a_jump, programmed with G105/G106, were assigned to the wrong axis.

ID#80082 : solved problem, solved since V0.22.6

Select/deselect CDC with transition block (G137):
If the CDC is selected and deselected in one block, the CNC program is not always completed. The error could also occur when a change is made to the selected side right after being selected.

NC Software - ARNC0 V0.22.5

ID#80415 : Information valid since V0.22.5

Included drive operating systems:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2

NC Software - ARNC0 V0.22.4

ID#80410 : Information valid since V0.22.4

Included drive operating systems:
For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2

ID#80102 : new function since V0.22.4

CNC monitor update:
The name of the active NC program or global NC subprogram is displayed in the CNC monitor.

ID#80097 : new function since V0.22.4

Updates to the cutter diameter compensation
The following new functions have been added to the mode "Activate/Deactivate CDC Indirectly with Entry or Exit Block" (G137):
- Deactivate CDC with exit block
- Activate CDC in the circular block
- Deactivate CDC in the circular block
- Change editing side in the circular block
- Change editing side with transition block

ID#78437 : solved problem, solved since V0.22.4

Dip in the path speed after G170 + unsync. M-function
If a combination of G170 and an unsynchronized M-function was used in a CNC program, undesired path speed reductions could occur at tangential transitions in the subsequent path sections.

ID#76215 : solved problem, solved since V0.22.4

System crash when updating the NC block monitor
If an NC program which calls global NC subprograms was moved backwards with a turned-on NC block monitor, the CNC crashed upon entry into the global NC subprogram (page fault).

**NC Software - ARNC0 V0.22.3**

ID#80437 : Information valid since V0.22.3

Included drive operating systems:

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2

ID#78015 : solved problem, solved since V0.22.3

Path speed reduction after G92+rotation

After a block with G92+rotation and the parameter \( v_{\text{jump}} = 0 \), a stop was always made at tangential path section transitions.

ID#77710 : solved problem, solved since V0.22.3

Motor parameters not transferred for resolver motors to the drive

The motor parameters from resolver motors were no longer transferred automatically from the NC configuration object to the drive (starting with V0.22.0).

ID#77685 : solved problem, solved since V0.22.3

Syntax error at G200 / G201

If G200/G201 <axis><edge> was followed immediately by 'End of Line', error 10200:”Unknown or missing letter in the NC program” was output (starting with V0.20.5).

**NC Software - ARNC0 V0.22.2**

ID#80405 : Information valid since V0.22.2

Included drive operating systems:

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2

ID#77330 : solved problem, solved since V0.22.2

CNC System: Path speed interruption at circle-circle transitions (r=const)

An undesired reduction in path speed can occur at tangential transitions between circle segments with the same radius.

**NC Software - ARNC0 V0.22.1**

ID#80400 : Information valid since V0.22.1

Included drive operating systems:

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2

ID#80210 : solved problem, solved since V0.22.1

Full circle with cutter diameter compensation (CDC)

If a full circle was traversed with active CDC, a position jump could occur on the circle axes at the end of the circle (starting with V0.22.0).

ID#80205 : new function since V0.22.1

CNC System: Software connection of trigger sources

Any axis within the ARNC0 can be defined as trigger source for a CNC axis. To do this, the reference to the axis object with the desired (HW) trigger inputs must be specified in the data structure

- "cnc_object.achse.achse[]\[\].trg_quelle.nc_object" (German)
- "cnc_object.axis.axis[]\[\].trg_source.nc_object" (English).

The defined trigger source is transferred using the NC action "ncAXIS, ncINIT".

**NC Software - ARNC0 V0.22.0**

ID#80337 : Information valid since V0.22.0

Included drive operating systems:

For ACOPOS 8V1xxx.00-1: ACP10SYS V0.54.1
For ACOPOS 8V1xxx.00-2: ACP10SYS V1.06.2

ID#80200 : solved problem, solved since V0.22.0
Error message for G201 together with tangential axis:

If G201 was used together with the tangential axis, error 10106: "Event not allowed at current module state" occurred on an axis.

ID#80195: solved problem, solved since V0.22.0
CNC program blocked with M0/M1:
The processing of a CNC program was blocked if a synchronous M-function and M0 or M1 occurred together in an NC block (starting with V0.20.2).

ID#80190: solved problem, solved since V0.22.0
Position jump during reverse movement of a CNC program
During reverse movement of a CNC program, a position jump could occur on all axes when entering a path section.

ID#80185: solved problem, solved since V0.22.0
CNC System: Reverse movement of CNC program not possible with G92+rotation
CNC programs in which the coordinate system rotates, could not be moved backwards (starting with V0.20.8).

ID#80180: solved problem, solved since V0.22.0
CNC program cannot be resumed after restart.
The CNC program was unable to be resumed following a CNC program restart (starting with V0.20.8).

ID#80165: solved problem, solved since V0.22.0
SW limit monitor:
A rounding error sometimes caused the values for the SW limits to differ from the default values in the axis structures. The values could deviate in the positive direction by a maximum of the unit factor of the corresponding axis ("cnc_object.axis.axis[]..unitfactor").

ID#80160: solved problem, solved since V0.22.0
Straight-circle transitions with cutter diameter compensation
At straight segments which are almost perpendicular, sometimes the intersection between straight segment and circle were not calculated correctly. The CNC program was aborted with Error 8134 or 8136: "Radius difference between start and end ".

ID#80155: solved problem, solved since V0.22.0
Full circle with cutter diameter compensation (CDC)
If the path section transition in a full circle is not exactly tangential, sometimes the full circle is not traversed when CDC is active.

ID#80150: new function since V0.22.0
The dependent libraries "ncda_lib" and "nccnccom" transferred to "ncglobal"
The functions from the libraries "ncda_lib" and "nccnccom" were transferred to "ncglobal" (starting with V0.30.0).
NOTE: These libraries must first be removed from the current project using the Library Manager before updating to ARNC0 0.220.

ID#80145: new function since V0.22.0
Download of ACOPOS parameter data:
New NC actions "ncSERVICE+ncACP_PAR, ncDOWNLOAD" for transferring the parameters contained in the ACOPOS parameter data (XML data with the format used for ACOPOS parameter tables) to the ACOPOS.

ID#80140: new function since V0.22.0
Loading the INIT parameter module:
Update to the data structure in the NC objects ncAXIS and ncCNCSYS (German/English): "global.init_par". The data from the specified INIT parameter module can be loaded to the corresponding user data structure of the NC object using the new NC action "ncGLOBAL, ncLOAD" or "ncGLOBAL, ncLOAD+ncINIT". If the option "+ncINIT" is set, the NC action "ncGLOBAL, ncINIT" is then automatically executed.

ID#80135: new function since V0.22.0

Revision Information ARNC0 118
CNC System: Updated monitor data structure

In the CNC monitor data structure "cnc_object.monitor", the currently processed NC block number (N...)
"satz_ncprog" (German) or "block_ncprog" (English) are displayed.

ID#80130 : new function since V0.22.0

Change of CNC movement parameter

The maximum size of an NC block was increased from 36 to 52:
German: "cnc_object.bewegung.ncsatz[52]"
English: "cnc_object.move.ncblock[52]" 

ID#80125 : new function since V0.22.0

CNC decoder parameter update

Update to CNC data structure with initialization variables for CNC special functions:
German: "cnc_object.decoder.parameter.wrk"
- "kreiersatz" (für G37)
- "konturverletzung" (für G39)
- "schnittpunktsatz" (für G134)

English: "cnc_object.decoder.parameter.cdc"
- "circ_replace" (für G37)
- "contour_violation" (für G39)
- "intersection_path" (für G134)
The element entries are transferred by calling the NC action "ncDECODER, ncINIT".

ID#76802 : solved problem, solved since V0.22.0

CNC System: CNC program is blocked with G201

The processing of a CNC program was blocked using G201 (starting with 0.208).