

SINUMERIK ONE / 828D / MC

Appendix Siemens Cycle Packages for SW 6.25, SW 5.25 und SW 1.25

1.	Reference documents.....	2
1.1.	List of reference documents and editions	2
2.	Compatibility	3
2.1.	Compatibility with cycle packages 840D sl / 828D as of SW 4.5 SP3	3
2.2.	Compatibility with grinding programs with 802D sl SW 1.4	5
2.3.	Adapting machine manufacturer cycles	5
2.3.1.	Change Status of CUST	5
2.3.2.	Special notes regarding the upgrade	6
3.	Änderungen zum vorherigen Stand.....	7
3.1.	Changes from SW 6.25	7
4.	Supplementary conditions.....	8
5.	Notes on the application of this cycle package.....	9
5.1.	Summary notes on the manufacturer cycle CUST_800	9
5.2.	Notes on block search with G code programs with CYCLE800.....	9
5.3.	Notes on cycle processing in a handling channel of the 828D	9
5.3.1.	General information on the function and test scope in step 3	9
5.3.2.	Additional notes on ISO mode and handling channel	9
5.4.	Note on the reverse translation of "old" measurement cycle calls without interface	10
5.5.	Note on technology Turning.....	10
5.6.	Notes for Measuring in the JOG for the Turning Technology	10
5.7.	Note for CYCLE9960.....	11

1. Reference documents

1.1. List of reference documents and editions

- /1/ SINUMERIK 828D / ONE
Measuring Cycles – Programming Manual 01/2025 (V6.25 / V5.25)
- /2/ SINUMERIK 828D SINUMERIK Operate (IM9)
Commissioning Manual 01/2025 (V5.25)
- /3/ SINUMERIK ONE / 828D
Milling – Operating Manual 01/2025 (V6.25 / V5.25)
- /4/ SINUMERIK ONE / 828D
Turning – Operating Manual 01/2025 (V6.25 / V5.25)
- /5/ SINUMERIK ONE / 828D
Programming Manual 01/2025 (V6.25 / V5.25)
- /6/ SINUMERIK ONE / 828D
Grinding – Operating Manual 01/2025 (V6.25 / V5.25)
- /7/ SINUMERIK ONE / 828D
Function Manual Special Functions 01/2025 (V6.25 / V5.25)

2. Compatibility

2.1. Compatibility with cycle packages 840D sl / 828D as of SW 4.5 SP3

In the following points, the behavior of the cycles has changed when compared to software releases as of SW 4.5 SP3:

Technology cycles:

- **As of 4.5 SP6:** Changes in the drilling cycles if no absolute drilling depth is programmed and the incremental drilling depth = 0 or is not programmed either:
Alarm 61019, "Parameter DP, DPR defined incorrectly", is now output if DPR = 0 or is not programmed and DP is not programmed.
Example cases:
 - CYCLE81(7,5,2, ,0) ; Alarm 61019
 - CYCLE81(7,5,2, ,) ; Alarm 61019
 - CYCLE81(7,5,2,0,0) ; absolute DP = 0 applies
 - CYCLE81(7,5,2,0,) ; absolute DP = 0 applies
- **As of SW 4.7:** Changes in the thread cutting (CYCLE99, CYCLE97) and thread chain (CYCLE98) cycles
The behavior for thread synchronization (operation via softkey in JOG) has been aligned to CYCLE 99 in CYCLE97 and CYCLE98. This means thread synchronization is only possible if no transformation, no TCARR and no rotations around X or Y are active.
- **As of SW 4.7 SP1:** Changes to the High Speed Settings cycle (CYCLE832) concerning Top Surface:
CTOL output in Top Surface as contour tolerance
CTOL output in Advanced Surface as axis tolerance (*root(3)) (compatibility)
- **As of SW 4.7 SP2 HF1:** Change in behavior in the swivel cycle (CYCLE800) or directly with the tool holder:
The following machine data must be taken into account:
 - MD 20196 \$MC_TOCARR_ROTAX_MODE Bit 2
 - Bit 2 = 0 ... compatibility, recommended when upgrading software
 - Bit 2 = 1 ... new function, recommended when commissioning for the first time

The Bit 2 = 1 setting means that the control system only calculates a definitive solution when swiveling in the pole position (basic position).
Existing NC programs with swivel function must subsequently be checked as the positioning behavior may have changed!
- **As of SW 4.8 SP1:** Until now, the tool holder position was retained when retracting to the retraction position in Z and Z, XY. This sometimes resulted in the software limit position (\$MA_POS_LIMIT_MINUS) being violated when a swiveled B axis was in the head, for example, when retracting to the retraction position.
When retracting Z and Z, XY to the retraction position, the retraction position entered is now approached in the machine (MCS). This prevents violation of the software limit position.
The new behavior has been implemented in CUST_800; the previous behavior can be restored by adjusting CUST_800.

SINUMERIK ONE / 828D / MC

Appendix Siemens Cycle Packages for SW 6.25, SW 5.25 und SW 1.25

- **As of SW 6.23;** If the CYCLE832 multi-axis program is selected in the High Speed Setting mask and no TRAORI is activated before the cycle is called, the cycle alarm "61147 Transformation not active: TRAORI before CYCLE832" will be displayed from V4.95 SP4/ V6.15 SP4/ V5.23 /V6.23 onwards.
There are the following options for adapting the program:
 1. Activate TRAORI before calling CYCLE832
 2. If no TRAORI is used in the program, then the multi-axis program in the mask can also be deselected.
 3. If the program compatibility is necessary, the alarm can be suppressed with Bit5 in the SD55220 \$SCS_FUNCTION_MASK_MILL_TOL_SET

Measuring cycles:

- **As of 4.7 SP4:** Functional improvement for workpiece measurement with automatic tool offset: The tool offset is precisely oriented to the tool management statuses

"Tool is active" and "Tool has been in use"
These statuses are evaluated even if no tool groups are provided (so-called "replacement tools"). Disabled tools are not offset. In this case, alarm 61404, "Tool offset could not be performed", is output.
- **As of 4.92:** the calibration behavior has changed during simultaneous measurement. If the "simultaneous measurement" function is active, calibrate once with cutting edge D1 and once with cutting edge D2 successively with the corresponding matching data set. The measurement input is determined automatically during the two processes and no longer has to be changed manually.
- **As of 4.95:** If in the BA JOG an edge is aligned over two measuring points ("Align edge", "Measure 2 holes", "Measure 2 pins") with active tool carriers, then the edge can be aligned using CYCLE800 after correction. Up to software 4.94, the machine was always swiveled back to the basic position. Since Software 4.95, the current level is retained. The plane is corrected exclusively by the measured angle values. If no tool carrier is active during the measurement. As before, only the measured rotary axis is positioned at 0 degrees in the active WKS.

ShopTurn cycles:

- **As of 4.5 SP5:** Regarding the note "Starting point or end point of the machining outside the retraction planes":
For programs with contour machining from earlier software releases, it is possible that, during an NC start, one of the alarms 61281, "Starting point of machining outside retraction planes", or 61282, "End point of machining outside retraction planes", is output.
In this case, adapt the retraction planes in the program header.

As of 4.94: Block search with program test has been modified such that the new variant "without approach" is used automatically. The old variant can be offered additionally via the MD 51024 \$MNS_BLOCK_SEARCH_MODE_JS or 51028 \$MNS_BLOCK_SEARCH_MODE by setting bit 6, if required When block search on a ShopTurn cycle, the new variant "without starting" must be used!

51024:\$MNS_BLOCK_SEARCH_MODE_MASK_JS
51028:\$MNS_BLOCK_SEARCH_MODE_MASK :
☐ Bit 0: Block search with calculation but no approach
☐ Bit 1: Block search with calculation with approach
☐ Bit 2: Reserved
☐ Bit 3: Skip EXTCALL programs
☐ Bit 4: Reserved
☒ Bit 5: Block search with test run without approach
☐ Bit 6: Block search with test run with approach

2.2. Compatibility with grinding programs with 802D sl SW 1.4

802D sl grinding programs are not executable without making the relevant adjustments. This is due to the new tool concept (grinding wheels and dresser).

2.3. Adapting machine manufacturer cycles

2.3.1. Change Status of CUST

The table provides an overview of the changes in the machine manufacturer cycles compared to software versions as of 4.5 SP3 (this also applies to the following SPx and Hfx releases).

Cycle	Function, significance	Adaptations after the upgrade
CUST_TECHCYC	Manufacture cycle to adapt the functions of the technological cycles	no change since SW 6.22
CUST_800	Manufacturer cycle for adaptation to the function swivel plane and swivel tool (CYCLE800)	includes new features in SW 6.24
CUST_832	Manufacturer cycle for adaptation of the High Speed settings function (CYCLE832)	no change since in SW 6.23
CUST_MEACYC	Manufacturer cycle to adapt functions for measuring cycles	no change since in SW 6.23
CUST_T	Cycle is used to track T-preparation after SERUPRO	No change since SW 4.90
CUST_M6	Cycle is used to track tool change after SERUPRO	No change since SW 4.4
CUST_MULTICHAN	Manufacturer cycle for lathes with several channels	no change since SW 4.5
CUST_CLAMP	Manufacturer cycle for clamping elements	no change since SW 4.5
CUST_MEAPROT	Manufacturer cycle for logging during measuring cycles	no change since SW 6.21
CUST_782	Manufacturer cycle for "Adjust to load" (CYCLE782)	No change since SW 4.92

2.3.2. Special notes regarding the upgrade

For functional entry points and changed functions regarding the individually changed CUST cycles, see *Kap. 5 Notes on the application of this cycle package*

To use the new functions, the cycles must be copied and appropriately adapted by the machine manufacturer (OEM).

Note:

Generally, the machine manufacturer is advised to copy all CUST cycles with changed functionality with respect to the initial state before the upgrade and then readapt them.

3. Änderungen zum vorherigen Stand

3.1. Changes from SW 6.25

This chapter only describes the changes regarding the released SW version 6.25, i.e. the cycle packages ZYKL_06.24.00.00.015.

New features, functional changes/enhancements Technological cycles:

- Turning technology – PrimeTurning
Realization of plan processing front and rear

New Functions, Functional Changes/Enhancements Measurement Cycles:

- Correction of kinematics (measure according to kinematics) even without operator receipt
- The measurement cycles support the existing 4 connection options for touch probes to SINAMICS at SINUMERIK ONE.

4. Supplementary conditions

Supplementary conditions:

- Note on tool measurement, turning tools on lathes with Y axis (CYCLE982):
To avoid traversing motion of a Y axis when measuring tools, a target position must be entered for the Y axis in the
SD54629 \$SNS_MEA_TP_TRIG_MINUS_DIR_AX3[n] and
SD54630 \$SNS_MEA_TP_TRIG_PLUS_DIR_AX3[n]
setting data.
- Note on kinematic measurement (with CYCLE996) when the program is called up from a "local drive"-or "USB":
No report is generated; the results are only available in the _OVR[n] result parameters of the measuring cycle and can be viewed in the Parameter operating area. (For a description of the relevant values, see Measuring Cycles manual).
- Note regarding kinematic measurement with CYCLE9960 on the counterspindle of turning machines:
 - (1) Positioning with circumferential movement at the counterspindle is not supported.
 - (2) The automatic shaft bypass at the counterspindle does not yet work, this must be organized by the user in the measuring program.
 - (3) With kinematic type TRAFO_DYN, the measurement results are inaccurate as the spindle position is not correct in each case.

5. Notes on the application of this cycle package

5.1. Summary notes on the manufacturer cycle CUST_800

Compared to SW 6.23, the manufacturer cycle CUST_800 has been functionally extended for the following applications:

Brand for block search for automatic rotary axes:

_M34 Block search for automatic rotary axis 1 and/or rotary axis 2

Previously, automatic rotary axes were used as a general rule for a sentence search in the PROG_EVENT.

5.2. Notes on block search with G code programs with CYCLE800

If a rotary axis is traversed between a CYCLE800 call and the target block, problems occur when calculating the block search. The search target should be preferably located before performing such a rotary axis movement.

5.3. Notes on cycle processing in a handling channel of the 828D

5.3.1. General information on the function and test scope in step 3

A handling channel with limited scope of functions is introduced for 828D. Cycles shall not be executed in this channel. Otherwise, the Alarm 61075 is output (so far without alarm text).

Tests can be performed for all technological cycles, all grinding cycles, all measuring cycles, all ISO mode functions as well as for measuring in JOG!

In general, JobShop programs are not executed in a handling channel. Otherwise, the Alarm 61076 is output (so far without alarm text).

5.3.2. Additional notes on ISO mode and handling channel

In general, the check for processing in a handling channel (call CYCLE218) has been implemented in all ISO mode cycles being assigned to a G-code.

No check for processing in a handling channel has been implemented in the following cycles:

- CYCLE305 OEM cycle (milling only)
- CYCLE3106 OEM cycle
- CYCLE396 Call from M96

CYCLE305 is called up with G05.1. This is an OEM cycle which has no function (it has only been prepared as such a cycle) and which must be functionally adjusted by the OEM.

CYCLE3106 is called up with G10.6. Here, the same applies as for CYCLE305.

CYCLE396 is not called up by a G function, but by an M function, i.e. M96. This M function ensures that a subprogram to be programmed by the user via parameter P is called up for an interrupt signal (from a high-speed I/O).

This cycle is not excluded with the handling channel! It comprises ASUP functionality which is not excluded for handling channels.

5.4. Note on the reverse translation of "old" measurement cycle calls without interface

Old measurement cycle calls, i.e. measurement cycle calls without an interface, can no longer be translated back. With regard to reverse translation, the cycle call must be reprogrammed.

5.5. Note on technology Turning

Only Y-turning tools of tool type 5x5 are allowed for the "Y-turning" function, i.e. this tool type may only be used for "Y-turning". The CYCLE97, 98, 99, 95 and 950 cycles do not support "Y-turning".

5.6. Notes for Measuring in the JOG for the Turning Technology

In the measurement cycles, spindle commands always refer to the active master spindle of the controller. When using the measurement cycles on machines with several spindles, the spindle in question must be defined as the master spindle before the cycle is called..

The range of measurement variants offered differs according to the discontinued technology (MD 52200 \$MCS_TECHNOLOGY), extension technology (MD 52201 \$MCS_TECHNOLOGY_EXTENSION) and MD 52740 \$MCS_MEA_FUNCTION_MASK Bit 3

Simple lathe (MD 52200 = 1, 52201 = 0):

Only rotary measurement cycles available

- Measuring leading edge
- Measuring inner diameter
- Measuring outer diameter

Turning-milling machine (MD 52200 = 1, 52201 = 2) without Y-axis:

Turning measurement cycles and milling measurement cycles that do not require a Y-axis

- Measuring leading edge
- Measuring inner diameter
- Measuring outer diameter
- Measuring edge
- Measuring distance 2 edges

Turning-milling machine (MD 52200 = 1, 52201 = 2) with Y-axis and MD 52740 Bit 3 = 0:

Turning measurement cycles and milling measurement cycles that do not require a Y-axis

- Measuring leading edge
- Measuring inner diameter
- Measuring outer diameter
- Measuring edge
- Measuring distance 2 edges

Turning-milling machine (MD 52200 = 1, 52201 = 2) with Y-axis and MD 52740 Bit 3 = 1:

Complete scope, rotary measurement cycles and milling measurement cycles

SINUMERIK ONE / 828D / MC

Appendix Siemens Cycle Packages for SW 6.25, SW 5.25 und SW 1.25

There are lathes that have a Y-axis with only a very small travel (of a few millimeters). For these machines, it does not make sense to offer measurement variants that require movement in the Y-axis. Therefore, MD 52740 \$MCS_MEA_FUNCTION_MASK bit 3 was introduced. Bit 3 can be used to control whether or not the measuring variants that require a Y-axis are displayed on a turning-milling machine with a Y-axis.

5.7. Note for CYCLE9960

Extension of the error message CYCLE9960::

Error message 61425: "Parameter for measuring axis 1 or 2 incorrect – error code: B(RA) minANG=<n>"

CYCLE9960 measuring range: An angle in the measuring range is very small, which leads to errors in the calculation. For the calculation of the axis to be measured, the basic position is necessary. This is not automatically a position in the selected measuring range. If the home position is missing in the measurement range, it is added. If the start or end point is unfavorably chosen, there may be too small an angle between 2 measuring points.

In the event of an error, the calculated rotary axis positions of the SGUD _OVR[0] to _OVR[12] can be viewed. A remedy can be to change the measuring range or to vary the number of measuring points.

Example1: Measuring range from -5° to 180°, 4 measuring points => -5°,0°,87.5°,180° => minANG=5.

Removal: Measuring range 0° to 180°, 4 measuring points => 0,60°,120°,180°

Example2: Measuring range from -110°..100°, 6 measuring points => -110°,-57.5°,-5°,0°,47.5°,100° => minANG=5.

Removal: -110°..100°, 5 measuring points => -110°,-40°,0°,30°,100°