

Installation and Operating Notes

The CNC Software 4.5 SP2 (internal Version 04.05.02.00.015) comprises the following components:

Component	Version
NCK	V87.09.00
SNCK	V02.06.00.00.001
SINAMICS	V04.50.30.19
PLC Op. Sys.	V32.81.01
FB15(sl)	V04.05.14
CP	V02.22.00
MCP CLIENT	V01.06.01
SINUMERIK Operate	V04.05.02.00.030
Linux basic system	V04.50.33
NCK file system driver	V04.04.00.00.004
Profinet FW	V04.01.33.52
Cycles	V04.05.47.03
SNC	V01.05.00.00.004

Requirements:

- The CNC SW 4.5 can only be operated on SINUMERIK NCUs 7x0.3 PN.
- CF Card Smart Modular Xceed 8GB. A specific CF card is required for the CNC software:
 - For the export version 6FC5851-1YG41-2YA8 or
 - For the standard version 6FC5851-1XG41-2YA8.
- No standard versions may be operated on CF cards which are licensed for the export version because, in this case, NC-Start is disabled.
- Step 7 Version V5.5 SP2 HF4 (or later)
- Current toolbox =V 04.05.16.00
- NCU Service System V04.60.23.00 (or later)
- If required, startup tool resp. HMI-Advanced V7.6 SP2 HF7 (or later) for drive commissioning, Actual startup tool V7.7
- If required, SINUMERIK Operate V4.5 SP2 for PCU 50 resp. PC / PG
- NCU 7x0.3 modules with boot code of PLC version V2.3 (see also the point "Boot code..")
-

Functional improvements in NCK SW 4.5 SP2 compared to SW 4.5 SP1:

PR number:

AP01509089	AP01508185	AP01504398	AP01504398	AP01502272
AP01499085	AP01497953	AP01496254	AP01495027	AP01491625
AP01484567	AP01484538	AP01483259	AP01483144	AP01483144
AP01482918	AP01478030	AP01477434	AP01474655	AP01473879
AP01470977	AP01470977	AP01469611	AP01469513	AP01468423
AP01465990	AP01465546	AP01463191	AP01461812	AP01459861
AP01456040	AP01455032	AP01453708	AP01451444	AP01448424
AP01430855	AP01424290	AP01422327	AP01421485	AP01419899
AP01418461	AP01416445	AP01412229	AP01411484	AP01408667

AP01406483	AP01406376	AP01404378	AP01403464	AP01402975
AP01402412	AP01400974	AP01398437	AP01397880	AP01392548
AP01391082	AP01391082	AP01389892	AP01384149	AP01384097
AP01382332	AP01382029	AP01381012	AP01380998	AP01379605
AP01375428	AP01367979	AP01366428	AP01362953	AP01359493
AP01352396	AP01351992	AP01351886	AP01351731	AP01350660
AP01349169	AP01329714	AP01317506	AP01316900	AP01309341
AP01298324	AP01295545	AP01286941	AP01265300	AP01233063
AP01180753	AP01177584	AP01099409	AP01023663	AP00855962

Functional improvements in Operate SW 4.5 SP2:

PR-Nummern	AP00973816	AP01017173	AP01131448	AP01132979	AP01210431
AP01235921	AP01259200	AP01282766	AP01290124	AP01290755	AP01296388
AP01301859	AP01312986	AP01331837	AP01333896	AP01335720	AP01339644
AP01340710	AP01342553	AP01348274	AP01352296	AP01355047	AP01355087
AP01355287	AP01356719	AP01361037	AP01368723	AP01372420	AP01373162
AP01376057	AP01377827	AP01378251	AP01379054	AP01379602	AP01381942
AP01381999	AP01382640	AP01384106	AP01387363	AP01389148	AP01390430
AP01391655	AP01392878	AP01393456	AP01395363	AP01395641	AP01395652
AP01395884	AP01396676	AP01398021	AP01398901	AP01399391	AP01399646
AP01399685	AP01403616	AP01404034	AP01406575	AP01406581	AP01406620
AP01412254	AP01413894	AP01413899	AP01414641	AP01415251	AP01416032
AP01416262	AP01418335	AP01419349	AP01420038	AP01422073	AP01422925
AP01423364	AP01424275	AP01424298	AP01424938	AP01426260	AP01427876
AP01428599	AP01429869	AP01430133	AP01430763	AP01433805	AP01435945
AP01436989	AP01438081	AP01438081	AP01438513	AP01439969	AP01440597
AP01441055	AP01441756	AP01442437	AP01443168	AP01443207	AP01443346
AP01443537	AP01444550	AP01445037	AP01445120	AP01445940	AP01448706
AP01448913	AP01452695	AP01455461	AP01456964	AP01457397	AP01462718
AP01463019	AP01466166	AP01472124	AP01473593	AP01477764	AP01478336
AP01479411	AP01479954	AP01480925	AP01481767	AP01481872	AP01482909
AP01483191	AP01483222	AP01484018	AP01484023	AP01484063	AP01484094
AP01486101	AP01486740	AP01487273	AP01488611	AP01488776	AP01489720
AP01489934	AP01490136	AP01492958	AP01493185	AP01493298	AP01493638
AP01494273	AP01494493	AP01495165	AP01495952	AP01496153	AP01496383
AP01498241	AP01498541	AP01498542	AP01498560	AP01499051	AP01499168
AP01499347	AP01499495	AP01500280	AP01500285	AP01500285	AP01500289
AP01500985	AP01501342	AP01501360	AP01501430	AP01501500	AP01502203
AP01502386	AP01502425	AP01502517	AP01505123		AP01507646
AP01507716	AP01508350	AP01509984	AP01510202	AP01517266	AP01517608
AP01517641	AP01519735				

Functional improvements in the Siemens cycles SW 4.5 SP2:

PR-Nummern	AP00936672	AP01265300	AP01281881	AP01289230	AP01333896
AP01338091	AP01348261	AP01351540	AP01352358	AP01361336	AP01362004
AP01364849	AP01367709	AP01367709	AP01374123	AP01378001	AP01378114
AP01380406	AP01381563	AP01383011	AP01385781	AP01386373	AP01389398
AP01393149	AP01394774	AP01398137	AP01399607	AP01401884	AP01403578
AP01404606	AP01404629	AP01407709	AP01413449	AP01413715	AP01421787

AP01422141	AP01424938	AP01425568	AP01430664	AP01435945	AP01436533
AP01436596	AP01436821	AP01439916	AP01440889	AP01441375	AP01447357
AP01448642	AP01449761	AP01454766	AP01457010	AP01458587	AP01463636
AP01465509	AP01466847	AP01467700	AP01469669	AP01469752	AP01470985
AP01472316	AP01473789	AP01476147	AP01481980	AP01484700	AP01488211
AP01492860	AP01495351	AP01496556	AP01497272	AP01498805	AP01500285
AP01500300	AP01500544	AP01500908	AP01502891	AP01504479	AP01504555
AP01505892	AP01507104	AP01508242	AP01508783	AP01509296	AP01510101
AP01527510	AP01528476				

Please observe the "Important notes on the Siemens cycles" on Page 14 !

Installing the CNC software:

The NCU service system V04.50.15 (or later) is required on USB stick to install the CNC software. New installations can be performed via the NCU service system and TCU or with WinSCP / RCS Commander. The software may only be installed on a Smart Modular XceedCF 8GB card – see prerequisites.

Boot code of the PLC:

Software release 4.5 SP2 only starts with PLC boot code V2.3 internally coded on the NCU-HW.

All NCUs 7x0.3 that were supplied from the middle of 2012, have this new PLC boot code V2.3:

- MLFB 6FC537x-0AA30-0AA0 from release G
- MLFB 6FC537x-0AA30-0AA1

To do this, nothing else has to be taken into account.

The version can also be determined with Step7 online under target system/diagnostics/module state.

For all NCUs 7x0.3 that were supplied before the middle of 2012:

MLFB 6FC537x-0AA30-0AA0 up to release F, the PLC boot code must be upgraded once.

To do this, proceed as follows:

If this PLC boot code has still not been installed, then the control will not start (in the 7-segment display there is "PLC" and SF and DP LEDs flash red).

In this state, the boot code upgrade can be initiated via the PLC rotary switch. The boot code is flashed when turning the PLC rotary switch to switch position 5. This can be identified by the fact that a rotating wheel is displayed in the 7-segment display. This lasts approx. 15 seconds. With this procedure, under no circumstances is it permissible that the NCU is switched off! The 7-segment display goes dark after the boot code upgrade has been completed. The PLC rotary switch can now be rotated back to position 3 (memory reset) and system booting (powering up) is continued.

Installing the NCU service system on USB stick:

The NCU service system is stored as USB stick image in the directory emergency_bootsys_ncu of the DVD delivered. The copy program installdisk.exe is also included. Connect a USB stick >= 2 GB to your PG resp. PC (with Windows XP) and determine the relevant drive letter. Call up installdisk to copy the NCU service system onto the USB stick:

```
installdisk --verbose --blocksize 1m <image file> <drive letter:>
```

We recommend that you execute this command in a DOS shell.

For this, you require administrator rights on your PG/PC.

Upon completion of installdisk, remove the USB stick from the PG / PC, boot once an NCU 7x0 from this stick (this boot process takes a little bit longer; a "P" in the 7-segment display of the NCU indicates that the FAT

partition of the USB stick is partitioned) and reinsert the stick in the PG / PC. Windows now displays an empty USB stick. Copy the requested CNC software (file with the extension .tgz) from the DVD delivered, the directory ncu_sw.

The USB stick can be inserted in one of the two USB plugs of the NCU 7x0.3. After being switched on, the NCU is booted from this USB stick. The system is operated either via a TCU which is connected to the NCU or via PG / PC using WinSCP under "Open Terminal".

The use of the NCU service system is described in the documentation /IM7/ on the DVD delivered.

We recommend that you use the USB stick "SIMATIC IPC USB Flashdrive, 6ES7648-0DC50-0AA0".

Alternatively, the image can be installed with the RCS Commander (6FC5860-7YC10-1YA0) via the menu item "Write file system image".

Notes:

- The contour handwheel and velocity override with handwheel have only been released for Profibus MCP.
- The reset button on the NCU allows to restart the NCU module. A restart of the entire system cannot be ensured in all configurations. This also applies to the PI service of PLC with FB4.
- As standard, the port 102 on X130 is disabled. If you wish to access the controller via X130 with Step 7 or SINUMERIK Operate, you must activate the port 102.
This can be achieved by making the following entry in the file /user/system/etc/basesys.ini:
[LinuxBase]
FirewallOpenPorts=TCP/102
If required, you can also activate the port 5900 for an external VNC Viewer.
- Spindle traversing with limited torque (Focon) or spindle traversing to fixed stop:
The performance indication on the SINUMERIK Operate is calculated as 100% of the limited torque and displayed.
- As before, the DBS11 is not released in combination with SINUMERIK 840D sl and integrated drive control.
- The Starter, in combination with 840 D sl, is still only released for diagnostic purposes.
- SPI and PW have always been keywords and may not be used as axis identifiers.
- The function G643 (block-internal approximate positioning) has been released for tool change applications (e.g. optimizations when approaching the tool change position). It has not been released for machining applications.
- The function G644 (corner rounding with maximally possible dynamics) has been released for tool change applications (e.g. optimizations when approaching the tool change position). It has not been released for machining applications.
- The function "extended measurement" with distributed measuring input at the SINAMICS modules is not functional (MEAC, MEASA, MEAWA).
- Access to arbitrary drive data via system variable \$nn_nn has not been released. Only the system variables transferred in the telegram 116/136 can be accessed.
- As from 4.4 SP1, the drive-autonomous ESR has also been released with Safety Integrated.
The machine data MD10089 \$MN_SAFE_PULSE_DIS_TIME_BUSFAIL (corresponding to p9580 in the relevant drive) allows to configure the time during which the NCK-SGA are still left in the old version (prior to communication failure) in case of a communication failure between NCK and drive. Afterwards, they are initialized.
The time provided for the safety drives for retraction is defined in the drive parameters p9697 and p9897.
Notice: These parameters are not included in the safety copy procedure. Proceed as follows to set these times:
 - Go to the safety drive commissioning mode (p10=95); for this, use the HMI functionality (softkey "Activate drive commissioning")
 - Set the value p9697 to the required value (should be identical to MD10089); Unit: ms
 - Set the value p9897 to the required value (should be identical to MD10089); Notice: Unit: us
 - Quit the commissioning mode via HMI; as a result, the checksums are confirmed and the data stored
 - When working without HMI, you have to confirm the checksums manually (copy 9798 to p9898, p010=0, p0971=1 resp. p977=1)

- On account of the interrupted communication, data cross-check errors occur, StopF. If the StopF-StopB time is not sufficient, the pulses are deleted by the StopB-StopA, retraction is not started resp. aborted too early (depending on the values for the StopF-StopB time resp. the StopB-StopA time).
- The MD 10062 \$MN_POSCTRL_CYCLE_DELAY must be zero. Check the existing data backup.
 - The machine data 32250 \$MA_RATED_OUTVAL[] must have the value zero.
 - After performing a block search, the auxiliary function M6 is no longer output by default.
- Remedy:
- Change the auxiliary function group in MD 22040 \$MC_AUXFU_PREDEF_GROUP[5] into a free group.
- If the value of the MD18210 deviates from the default, increase the MD18210 by at least 21 MB DRAM.
 - Deleting the PLC 317-3PN/DP:
After deleting the PLC via an operator sequence at the PLC rotary switch or from Step 7, the PLC program is automatically reloaded.
When starting the PLC initialization via the operator handling PLC switch into position 3 and power off/on, the PLC program is not loaded automatically. This is then neither possible via the above-mentioned operator sequence.
 - The PLC series commissioning archive must be created in the stop status of the PLC. Otherwise, it is not ensured that the PLC goes to cyclic mode after loading the data backup.
 - The service interface X127 of the NCU may only be used for commissioning and service.
 - The Profibus 1 must be configured to ensure that the softkeys CU, Infeed, Drives are displayed on the SINUMERIK Operate.
 - No parallel use of measuring functions for i-/n-measurement via the Starter and HMI-Advanced. If Starter measurement is required, this requires a Power OFF=>ON and direct measurement with the Starter. Measurements with HMI-Advanced may not be performed prior to Starter measurements.
 - When replacing a SINAMICS component resp. performing an upgrade, check the firmware version and restore the original version, if required.
(Keyword: Macro 150399)
The firmware versions can be checked in the following parameters:
Control Unit R18 → Firmware Version of the CU
Infeed unit R128 → Firmware Version of the infeed unit
Drive MD R128 → Firmware Version of the power unit
R148 → Firmware Version of the sensor module
 - The Starter is not yet required to determine the motor codes.
 - Line voltages: To ensure smooth operation in the target country, the following parameters must be set in the stated order:
P010 = 1
P210 = Rated line voltage in the target country
P211 = Rated line frequency in the target country
P340 = 1
P3410 = 4
P3900 = 3
Save RAM to ROM
Switch off the system, only switch it on in the target network.
With the next ON command on the ALM, a line identification is performed and the values determined are stored in a power fail-proof manner.
 - With the SINUMERIK Software Versions 1.3 and 1.4 / 2.4, you may combine different firmware versions of the components involved in the **safety functions** (NCU, NX, Motor Modules, DRIVE-CliQ motors) without adjusting the firmware versions.
The following applies as from SINUMERIK Software Version 1.5 / 2.5:
The firmware versions of the Motor Modules, Sensor Modules and DRIVE-CliQ motors (comprising integrated Sensor Modules) involved in the safety functions must be adjusted to the SINAMICS firmware version of the NCU.
This is automatically performed during the startup if the parameter p7826 (firmware automatic) = 1 is set (standard setting). When using **Safety Integrated**, the parameter p9826 (firmware automatic) = 1 must be set and may not be reparameterized. When performing an acceptance test of **Safety Integrated**, you

must read out and record the safety firmware versions of all Motor Modules, Sensor Modules and DRIVE-CliQ motors involved in the safety functions, and check them against the following list:

<http://support.automation.siemens.com/WW/view/de/28554461>

Each line in the tables displays a permissible combination of safety firmware versions.

- Drive system utilization of CU or NX:
Setpoint values are displayed in V4.4 (or later). The values do not fluctuate any more, in contrast to former times. Now, the statement applies that r9976[1] and r9976[5] must both be less than or equal to 100% to ensure that the utilization is permissible. If one of these two indexes exceeds 100 %, the fault F1054 "CU: System limit exceeded" is output.
- In order to have sufficient reserves for program processing, the maximum load with typical user operations should not exceed 75%. The current average load should not exceed 50%. The current load can be checked under Diagnosis / System resources.
- Cam output (position switching signals) with modulo rotary axes is not possible in a time frame comprising less than 4 IPO cycles and is rejected with Alarm 21751.
- The machine data used to describe the spindle dynamics must be set such that they approximately correspond to the real spindle dynamics. If the values are unnecessarily exceeded, alarms may occur upon the transition from spindle to positioning mode.
- NCU 7xx.3 PN: The "DIAG" button has no function with 840D sl.
- The initial address of the NCK (slot 4 in the S7 hardware configuration) changes from 256 to 4096 (outside the peripheral image). For this reason, the hardware configuration must be newly created. Here, you must ensure that all properties of the PLC CPU (e.g. remanence ranges, cycles, cycle time monitors, etc.) that deviate from the standard are set as before.
- Alarm 411503: When using existing configurations, the alarm 411503 (incorrect configuring) is output and the PLC set to stop.
- With SMI encoder, the function "Parking axis" is not supported without voltage ON/OFF.
- Alarm 7300: When using NC archives of Software Versions lower than NCK 75.00.00, the alarm 7300 is output if the MD18235 is 0. The standard setting is MD18235=20000.
- The function "Fixed point approach G75" is not permissible with active radius offset G41 / G42 (Alarm 14091 Index 9).
- With fixed point approach with G75, the fixed point is now approached with non-linear interpolation (RTLIOF). As a result, the axes are traversed at the jerk set for positioning axes (\$MA_JOG_AND_POS_MAX_JERK).
- The machine data 11295[0..9] is now memory-configuring. Values deviating from the standard must be removed from existing archives before performing the upgrade.
- If you wish to use the service interface X127 with PG/PC on an NCU7x0.x with SW V2.6 (or later), please observe that the PG/PC network interface is operated in the mode "DHCP Client", that means "Automatically obtain IP address".
If you have set the PG/PC network interface to a permanent IP (192.168.215.xx), a ping is sent to X127, but you cannot go online neither with Step7 nor with SINUMERIK Operate. The firewall is activated by default setting.
- If you wish to operate more than six axes, use the modules NX10.3 or NX15.3.
- Archives with MD 19730.11 = false are incompatible. The machine data 19730.11 (hmi_function_mask) must always be set.
- Archives with MD18150=150 (old default value / default is 196) may be incompatible.
- Archives with MD 32250 > 0 are incompatible. With MD32250 > 0, neither speed setpoint value nor torque setpoint value standardization are taken from the drive.
- The interface signal "Invert M3/M4" is now also active when tapping with G331/G332.
In applications where the interface signal is always set to "1", the spindle now rotates in the incorrect direction with G331/G332. To achieve a behavior which is compatible with earlier versions, set the bit 22 in the MD35035 to "1" (default is 0).
- Archives with MD28253=100 (old default value) may be incompatible. Change the MD to 200 (default new).
- Archives with MD19730[0]=0 are incompatible. Missing softkeys in MDA. Change the MD to 804hex.
- The function TANGON with additional parameter "P" has not been released. Remedy: Use the standard setting "S".

- The SD43235 has a speed limit of 10,000 rpm as default value. When upgrading to Software Version NCK 78.00 or later and with spindle speeds exceeding 10,000 rpm, the SD43235 must be increased to an appropriate value. The speed is limited by SD43235 if the system variable \$AC_SMAXVELO_INFO[n] has the value 21.
- Multitool as manual tool:
With repeated selection (of tools within this manual multitool), in the sequence T="x" M06 --> T0 M06 --> T="Y" M06 --> T0 M06, the identifier "manual tool" is not set upon the second tool selection. As a result, the system tries upon the next T0 M06 to store the multitool in the magazine.
- Safety:

Alarm 27071 checksum error SPL parameterization

To facilitate handling when testing the valid scope of languages of the program SAFE.SPF, bit 2 of the machine data \$MN_SAFE_MODE_MASK has not been included in the checksum calculation. This allows to perform tests without confirming the checksum.

Due to the recalculation of the checksum without this bit, the relevant checksum must be confirmed once.

- Safety:
If safe operating stop is active for an axis and the handwheel is actuated for this axis in this state, the pulses are collected and traversed after deselecting the safe operating stop.
Proceed as follows to avoid this behavior:
In MD MD 20624 \$MC_HANDWH_CHAN_STOP_COND, set bit 7 to "1" (feed stop)
In the PLC, select the link "Select feed stop with active handwheel and active SBH".
Upon deselection of SBH, deselect the feed stop as well.
- Due to the F-CPU, the I/O addresses of F-DI/F-DO modules must now be included in the PLC process image.
- Multitool:
A multitool may not be a manual tool
 - a. With certain change procedures, the target position for the old tools is not output to the user interface (DB72)
Tool management: Multitool, incomplete data for the old tool to VDI in case of MT change.
 - b. With the end acknowledgement of the T preparation, the multitool position (\$TC_MTP_POS) is already set to the new location.
Thus, the system cannot detect during a following change cycle that the multitool must be positioned.
Tool management: Multitool \$TC_MTP_POS is incorrectly set with MT==manual_tool.
Tool provision:
Programming sequence
...
M06
T="drill";Provision
...
T="drill";repeated tool programming
M06
...
Can only be used if the Bit11 is set to 0 in \$MC_Tool_Management_Mask.
Otherwise, this programming leads to the Alarm 6402 upon the 2nd tool call.
Tool management: Multitool, Alarm 6402 in combination with MD20310 Bit11=1
That means the setting \$SCS_FUNCTION_MASK_Tech_Set Bit0=1 is not supported (tool preselection)
- When working with multitool and multitool location OEM data, you must change the data types (\$MN_MM_TYPE_CC_MULTITool_PARAM and \$MN_MM_TYPE_CC_MTLOC_PARAM) from default=3 (integer) to the value=4 (real).
- When taking data backups from NCU versions lower than Software Version 4.4, the machine data 10185 NCK_PCOS_TIME_RATIO should be checked. The setting of NCK_PCOS_TIME_RATIO should be as follows:
 - With NCU 7x0.1 and 7x0.2: 65% with internal HMI and 90% with external HMI

- With NCU 7x0.3: 90%

○ **Fading out the tool management**

You can deactivate, that means fade out, the tool management in the Parameter operator area. For this, change the entry DLG002 in the file systemconfiguration.ini in the paragraph [dialogs] as follows:
[dialogs]

```
...
DLG002= name:=SIParameter, implementation:=slpdialog.SIPaDialog, process:=SIHmiHost1,
preload:=true, terminate:=false, cmdline:="-conf slpdialog.hmi"
...
```

In this case, the softkey "Zero offset" is selected when selecting the Parameter operating area for the first time.

The modified file system configuration.ini must be stored in the directory /oem/sinumerik/hmi/cfg or /user/sinumerik/hmi/cfg.

- The existing Siemens system password is no longer valid.
- The machine data 28070 \$mc_mm_num_blocks_in_prep should be set to a value greater than or equal to 80.
- With the new channel-specific machine data MD20115 \$MC_IGNORE_REFP_LOCK_ASUP and MD20105 \$MC_PROG_EVENT_IGN_REFP_LOCK, the customer can set separately for each ASUP channel resp. each Prog-Event whether the axes must be referenced for the start.
If the bit 1 of \$MN_ASUP_START_MASK is set prior to Software Version 4.5, the Version 4.5
N20115 \$MC_IGNORE_REFP_LOCK_ASUP='HFFFFFFF'
N20105 \$MC_PROG_EVENT_IGN_REFP_LOCK='H3F'
must be set in all channels to enforce an identical behavior.
If the bits in \$MC_IGNORE_REFP_LOCK_ASUP or \$MC_PROG_EVENT_IGN_REFP_LOCK are not set, but not all axes are referenced in ASUP resp. Prog-Event-Start, the start is rejected and a new Alarm 10204 output
- In Software Version 4.5, the compile cycle interface has changed. For this reason, new compile cycles are required.
- In Software Version 4.5, the compile cycle CC-MCSC is no longer provided **as standard** and replaced by the generic couplings (CPSETTYPE="TRAIL" CPFERS="MCS" / see also Function Manual Special Functions -M3).
- After a software upgrade, the NC MD 18150 must be increased at least to the standard value 196.
- Safety data must be confirmed when upgrading from 4.4 to 4.5.
- The NCK can only access PLC I/Os on the DP1 bus.
- **Safety:**
Alarm 1671 with identifier 9514 after upgrading and use of linear motors
Sinamics 4.50 provides a new safety drive parameter: p9514, absolute encoder linear measuring steps (CU).
This parameter corresponds to the parameter p422.
The Alarm 1671 indicates that the values in p422 and p9514 are not identical.
When upgrading SW <4.5 with commissioning archive, this alarm may occur, when using linear motors, if the parameter p422 does not show the value "100" (default value of p9514).
To correct the error, enter the value of p422 in p9514 and then confirm the checksum for this drive.
When commissioning a new system, this status does not occur because the values are automatically copied as long as safety is not active (p9501=0)
- The Sinamics function "know-how protection" has not been released for Sinumerik.
- Alarm 12080: In the part program, all ASCII characters > 127 (0x7F) are rejected and Alarm 12080 is output. The existing behavior stating that all impermissible ASCII characters are internally converted into blanks can be set anew with the machine data \$MN_PROG_FUNCTION_MASK BIT3 = 1. The presetting is BIT3 = 0.
- The Installdisk for creating a boot-capable USB stick must be installed with installdiskCmd.exe.
- The bus systems at DP1 (X126) and DP2 (X136) and PN (X150) can be simultaneously operated with clock cycle synchronism (isochronous operation).
- PLC FC24 key-operated switch position:

From SW 4.5 SP2 the FC24 also transfers the key-operated switch signals to the user interface (DB10.DBX56.4..7). This transfer takes place, irrespective of whether a key-operated switch is mounted at the MCP. If previously, the key-operated switch information was entered into the user interface by the user program, then this must possibly be adapted.

- ADI4:
To avoid access conflicts between PROFIBUS drives and I/O modules, the I/O addresses of DP slave ADI4 must lie completely outside the process image.
- Alarms 400551 - 400553:
For module faults at the DP or PN bus (PLC fault LED bright), then alarms (400551..400553) are output. The error event/alarm can be suppressed using SFC12 (deactivating and activating DP slaves).
- It is not permissible that PN drives are operated at X150.
- MCP PN: From software release 4.5 SP2 and higher, machine control panels and the connected handwheels can be operated in the Profinet mode.

Example:

The first handwheel is connected at the first MCP X60 with MCP address 2.

- DIP switch S3 of the MCP
9 &10 = on
2 = on

- FB1 parameters in the OB100

```

CALL "RUN_UP" , "gp_par"
MCPNum      :=1
MCP1In      :=P#E 0.0
MCP1Out     :=P#A 0.0
MCP1StatSend :=P#A 8.0
MCP1StatRec :=P#A 12.0
MCP1BusAdr  :=2
*
*
*
MCPBusType :=B#16#66 //für MCP1&2 PN B#16#66

```

- Machine data
11350[0] = 5
11351[0] = 1
11352[0] = 1
11353[0] =10000

- Step 7 hardware configuration

The screenshot displays the 'HW Konfig' software interface for configuring an NCU 720.3 PN. The main window is titled '[0] NCU 720.3 PN [V3.2+]'. On the left, a tree view shows the hardware components, with 'CPU 317F-3 PN/DP' selected. The right pane shows a network diagram labeled 'Ethernet(1): PROFINET-IO-System (100)' with two MCs connected. The bottom pane shows a detailed table for the selected component 'MCP-pn2'.

Steckplatz	Baugruppe	Bestellnummer	E-Adresse	A-Adresse	Diagnoseadresse	Kommentar
0	mcp-pn2	6FC5303-XAFXX-XXXX			16342*	
Interface	MCP				16341*	
X1	port-001				16340*	
X2	port-002				16339*	
1	MCP 483C PN		0...13	0...7		
2	Option handwheels		10000...10003			

- Incremental jogging for G700 :
If, for rotary and linear axes, different computation resolutions have been set via \$MN_INT_INCR_PER_MM and \$MN_INT_INCR_PER_DEG, then this must also be taken into account in the new machine data 31092 \$MA_JOG_INCR_WEIGHT_TRAFO. This applies if the machine axes assigned to the geometry axes are rotary axes.
- Collision avoidance has only been released for the field test.
The following applies:
The indices for kinematic chains, protection areas, protection area elements and collision pairs must be allocated starting at 0 and must be seamless (no gaps).

Notes regarding SINUMERIK Operate:

- When operating an NCU without TCU (i.e. with SINUMERIK Operate on a PCU 50), then the internal SINUMERIK Operate must be deactivated. This is realized using the service command "sc disable hmi".
- Only one SINUMERIK Operate may be active on one NCU, either the internal SINUMERIK Operate or SINUMERIK Operate for PCU 50 / PC.
Exceptions: Commissioning / service; SINUMERIK Operate on PCU 50 for the main operator station with permanently assigned MCP and one tool loading station with internal SINUMERIK Operate without

MCP.

- Path names in SELECT instructions of job lists referring to part programs on the CF card or USB data carrier are case-sensitive.
- Handling of special characters in the editor:
The SINUMERIK Operate editor allows to edit text files which use the character LF (0aH) or the string CRLF (0d0aH) as line resp. end-of-block identifier. Binary files cannot be opened with the editor. Files with the extensions .EXE, .LIB, .ELF, .ARC, .TS, .ZIP, .SO, .PNG, .BMP, .ICO, .CFS, .BIN, .QM, .HMI, .CFG, .ACX, .EMF, .ALM, .ARD, .TGZ, .PTE, .CYC cannot be opened either.
Files which are newly created with the SINUMERIK Operate editor have been coded in UTF8 and use the character LF as end-of-block identifier. With UTF8-coded files, all special characters are displayed correctly. When opening files, the SINUMERIK Operate editor assumes that the files have been coded in UTF8. When opening files with another coding, e.g. with Windows code-page coding, special characters are only displayed correctly if the SINUMERIK Operate has been switched over to the corresponding system language. This applies, for example, to files which have been created with the HMI-Advanced editor. The coding of such files is not modified when being opened with the SINUMERIK Operate editor. They are not automatically converted into the UTF-8 coding.
If files are created or edited with an external editor (e.g. Notepad under Windows) rather than the SINUMERIK Operate editor, please ensure that the file is saved in UTF8 coding. For this, when using Notepad, enter "UTF-8" as encoding when storing the file in the dialog "Save As". If no special characters are used, you can also state "ANSI" as encoding.
For the V24 data transfer, install a V24 module in the NCU.
Notice: Different V24 modules are required for the NCUs 7x0.2 and 7x0.3.
The V24 baudrate is limited to max. 19200.
- In the Machine area, the "Retract" softkey is no longer available as standard. It can be reactivated by creating the slmaconfig.xml in /oem/sinumerik/hmi/cfg file with the following content:

```
<?xml version="1.0" encoding="UTF-8" standalone="yes" ?>
<CONFIGURATION>
<Settings>
<ShowRetractSoftkey type="bool" value="true" />
</Settings></CONFIGURATION>
```

Networking:

- Important system and network settings of the NCU Base software are specified in the file **basesys.ini** in the directory /card/user/system/etc and may be modified. The original basesys.ini is called "template-basesys.ini" and is included in /card/siemens/system/etc ".
Each NCU in the plant network should be assigned an unambiguous ("talking") computer name, with the entry "Hostname=..." in the basesys.ini. Upper and lower cases, digits and minus signs are allowed.
- Always switch in the DHCP server of the NCU.
- When changing IP addresses of NCUs / PCUs, execute the service command "sc clear dhcp" to activate the change.

Configuration with 1 NCU with TCUs and MCPs

In this configuration, no specific settings must be made in the basesys.ini. The DHCP server and internal SINUMERIK Operate remain switched on.

Configuration with 1 NCU with 1 PCU 50, MCPs and possibly TCUs

In this configuration, the following settings must be made in the basesys.ini of the NCU:

Hostname =

SyncModeDHCPD_SysNet = ON_MASTER

Generally, the internal Operate should be switched off, as operating two HMIs connected to one NCU is only permissible in special cases.

If the system has a TCU (HT8) which shall be connected to the PCU 50 as standard, the following entries must be made in the file /card/user/common/tcu/<TCU name>/common/tcu/config.ini on the NCU:

```
MaxHostIndex = 1
[host_1]
Address = <IP address of the PCU>.
```

Configuration with several NCUs and possibly 1 or several PCUs, TCUs, MCPs

In this configuration, the following settings must be made in the basesys.ini of the NCU:

```
Unambiguous Hostname = .... for each NCU
Unambiguous InternalIP= ....
Same InternalNetMask= ....
Exactly one NCU with SyncModeDHCPD_SysNet = ON_MASTER
```

During the first startup after networking, the DHCP master shall run up first, followed by all other stations. All operator stations in the plant network (TCUs, HT8, PCU 50) are managed on the NCU with "ON_MASTER", that means the config.ini files relevant during the runtime and the .leases file comprising all IP addresses assigned in the system are located here.

The config.ini files are distributed by the master NCU to all other NCUs / PCUs via the service command "sc distribute tcudata".

With the service command "sc clear dhcp" and subsequent switching off / on of the overall system, the NCUs / PCUs are assigned their preset IP addresses, which are newly assigned for TCUs and MCPs, the .leases file is the distributed to all other NCUs / PCUs.

7. Cycle packages:

With CNC SW 4.5 SP2, the following cycle packages are stored on the CF card:

- Standard cycles (technological cycles)
- Measuring cycles
- ISO cycles
- ShopMill cycles
- ShopTurn cycles

All these Siemens cycles are automatically loaded onto the NC during startup of the NCU. The corresponding variables are stored in the definition file PGUD.

Important note:

The "programGUIDE" (previously cycle support) in SINUMERIK Operate is based on the cycle packages of the CNC SW 4.5 SP2. Cycle calls in part programs for these cycles cannot be recompiled or processed with cycle support in HMI-Advanced 7.x.

1 Reference documents

1.1 List of reference documents and editions

- /1/ SINUMERIK 840D sl / 828D
Measuring cycles–Programming Manual 03/2013 (SW 4.5 SP2)
- /2/ SINUMERIK 840D sl / SINUMERIK Operate (IM9)
Commissioning Manual 03/2013 (SW 4.5 SP2)

- /3/ SINUMERIK 840D sl / 840D / 840Di
Cycles – Programming Manual 01/2008 (SW 1.5 840D sl or SW 7.5 840D)
- /4/ SINUMERIK 840D sl / 828D
Milling – Operating Manual 03/2013 (SW 4.5 SP2)
- /5/ SINUMERIK 840D sl / 828D
Turning – Operating Manual 03/2013 (SW 4.5 SP2)
- /6/ SINUMERIK 840D sl / 828D
Production planning – Programming Manual 03/2013 (SW 4.5 SP2)

1.2 Documentation overview –where are cycle topics described

Document	Contents
SINUMERIK 840D sl/828D Milling Operating Manual	In Chapter 2 – All functions for measuring in JOG - workpiece and tool on milling machines Chapter 6 – Generating G code program Chapter 7 – Generating ShopMill program Chapter 8 – programming cycles (in these, all drilling, milling and turning cycles)
SINUMERIK 840D sl/828D Turning Operating Manual	In Chapter 2 – Measuring tools in JOG on a lathe Chapter 6 – Generating a G -code program Chapter 7 – Generating a ShopTurn -program Chapter 8 – Programming cycles (in these, all drilling, milling and turning cycles)
SINUMERIK 840D sl/828D Measuring cycles Programming Manual	Chapter 1 – Basics/measuring principle etc. (as before) Chapter 2 – All measuring versions, workpiece and tool measuring, turning and milling– SK oriented Chapter 3 – Overview parameter list of the measuring cycles (for external programming): contains the call interfaces for all measuring cycles in the G-code and detailed description of the transfer parameters Chapter 4 – Attachments, list for the changeover to SW from 2.7/4.4
SINUMERIK 840D sl/828D Programming Manual - Production Planning	Chapter 16 - Externally programming cycles: contains the call interfaces of all technological cycles in the G-code and the detailed description of the transfer parameters
SINUMERIK 840D sl SINUMERIK Operate (IM9) Commissioning Manual	Chapter 9 – Settings for simulation and simultaneous recording Chapter 20 – Configuring cycles Contains a description of the machine and setting data for cycles, swiveling, JobShop, adapting manufacturer cycles (CUST_TECHCYC, CUST_MEACYC etc.)

2 Cycle packages– Functional scope

2.1 Technological cycles

The functional scope of the technological cycles is described in documents /4/ and /5/.

Difference list cycles/cycle functions for 840D sl SW 1.5 (cycles SW 7.5):

The following cycles/cycle functions are not included in this cycle package:

Cycle	Function and entry softkeys
POCKET1 POCKET2	Rectangular pocket/circular pocket ¹

An assignment list of the earlier adjustable parameters (GUD-variable) to the machine and setting data is provided in Attachment_1.

2.2 Measuring cycles

The functional scope of the measuring cycles is described in document/1/.

Difference list measuring cycles/cycle functions for 840D sl SW 1.5 (measuring cycles SW 7.5):

The following cycles/cycle functions are not included in this cycle package:

- Logging measuring results
- For workpiece measurement with automatic tool correction, the tool to be corrected can still only be specified using the tool name – the alternative by specifying a tool number is no longer possible. Further, it is no longer possible to select a tool environment TENV. This function is an absolute special case and was no longer realized from SW 4.4.

2.3 Cycles for ISO compatibility

The setting data for cycles for ISO compatibility (GUD fields _ZSFR, _ZSFI) has been changed over to configured machine and setting data.

An assignment list of the previous adjustable parameters (GUDvariable) to the machine and setting data is provided in Attachment_2.

¹ POCKET1/POCKET2 functions are covered by the newer pocket milling cycles POCKET3/POCKET4

3 Compatibility

3.1 Compatibility to the cycle packages 840D sl up to SW 1.5

In the following points, the behavior of the cycles has changed as compared to Software Versions up to and including SW 1.5:

Technological cycles:

1. Milling cycles - Calculation of the infeed depth with / without consideration of the safety clearance (could so far be set in the GUD variables `_ZSD[1]`):
In the setting data `$SCS_FUNCTION_MASK_MILL_SET Bit2`, you can set whether the safety clearance is considered in the depth calculation or not. So far, "with consideration of the safety clearance" was set as default – now "without consideration of the safety clearance" is set as default.
Thus, the individual depth infeeds may change in the existing program.
The setting of this setting data has the same effect in G-code programs and ShopMill resp. ShopTurn programs.
2. Milling cycles when calling with MCALL:
Milling cycles in G-code programs can optionally also be called up on position patterns with MCALL.
3. Thread drilling cycles `CYCLE84`, `CYCLE840` – technological parameters (could so far be set in the GUD variables):
The machine manufacturer must enter the required settings of the technological parameters in the corresponding setting data. These settings are neither changed by the cycle nor by the input screen form
4. Thread drilling cycle `CYCLE840` – parameter for reversal of rotating direction:
The parameter "rotating direction for retraction" is no longer supported in the cycle input screen. This makes sense as the cycle now assumes this function for both thread types and only functions if a spindle direction has been programmed before the call-up.
5. Pocket milling cycles `POCKET3` and `POCKET4` – Parameter for plunge feed:
The programmed feedrate for depth infeed is only active with "pre-drilled" and "vertical". "Helical" plunging and "oscillating" are performed with the infeed for machining in the plane.
6. Milling cycle rectangular pocket `POCKET3` – Reference to the pocket angle of rotation:
The angle of rotation now always acts at the point of reference. The programming for rectangular pocket "reference point is the corner" and "angle of rotation refers to the center" (with `ZSD[2]=1` and `ZSD[9]=1`) is no longer supported, but leads to the Alarm 61109 "Parameter `_STA` incorrectly defined".
7. Milling cycles – Parameter for infeed width:
A value > 0 must always be entered for the infeed width. In the screen, the field is marked as incorrect and the cycle outputs an alarm.
8. Drilling and milling cycles – parameter for safety clearance:
A value > 0 must always be entered for the safety clearance as this value also affects the switchover from G0 to G1 when approaching for machining in the tool axis. In the screen form, this field is marked as incorrect.
9. High-speed settings `CYCLE832`: The differences are explained on the basis of the documentation:
Documentation up to and including SW 7.5: Cycles – Programming Manual /3/
Documentation as from SW 2.6: Commissioning Manual IHsl IM9 /2/
Milling Operator Manual /4/ resp. Turning Operator Manual /5/
 - As from Software Version 2.6, all functions documented in /3/ "Point 3.17" which are coded in the parameter `_TOLM`, except the machining type (`_TOLM` unit position), are only supported in the compatibility mode.
 - All GUDs described in /3/ Point "3.17.4.3 Adjustment by the machine manufacturer" and the documented functionality no longer apply as from Software Version 2.6.
 - Note regarding /3/ "Point 3.17.4.4 Adjustment of additional program parameters `CYC_832T`"
As from Software Version 2.6, the user cycle `CYC_832T.SPF` is replaced by the cycle `CUST_832.SPF`. For reasons of compatibility, the markers `_M0` to `_M4` in the cycle `CUST_832` are still provided. The transfer parameter `_OVL_on` of the `CYC_832T` is no longer used because the documented machine data in `CYCLE832` are not rewritten.
 - Note regarding /3/ Point "3.17.5 Interfaces"
All documented machine and setting data are not rewritten in `CYCLE832` as from Software Version 2.6. All documented channel-specific variables `GUD7` are omitted as from Software Version 2.6.

- When using the CYCLE832, the option Advanced Surface (AS) must always be active. Otherwise, fault 8025 “Option AS not set” is output. According to the application example AS in the Commissioning Manual HMI Operate (03/2010), AS requires specific G commands. These G commands are not necessarily compatible with the settings of the "old" CYCLE832 resp. CYC_832T. These new G commands are entered as a suggestion (commented out with ;) in the individual markers of the compatibility branch (as from the marker _M1). The SOFT command is always activated because BRISK does not fit with AS. The old calls CYCLE832 should only be used in exceptional cases. When upgrading the machine, the old calls should preferably be replaced by new calls with CYCLE832. The tolerance settings and machining type (roughing, finishing, ..) settings can be transferred. The parameter _FACTOR should no longer be used. Factor settings see Commissioning Manual HMI Operate (03/2010) -> SD55441 to SD55443. If the parameter _FACTOR in CUST_832.SPF is assigned in the compatibility branch, this value is active..
- 10. Circular pocket cycle POCKET4 – Plunging: With depth infeed with G0 (pre-drilled), the infeed is always performed with G0 independently of whether raw dimension is programmed or not. Sufficient space up to (DP+SDIS) must be provided for this. With infeed into the material, the programmed helix radius is active if it is smaller than the tool radius. Up until now, the tool radius was active in this case.
- 11. Pocket milling cycles POCKET3, POCKET4 – special case 1 depth infeed: Only one depth infeed is performed when roughing / finishing if the infeed depth is >= max. material depth to be removed.

Possibly existing programs must be adjusted for these changes

Measuring cycles:

1. Due to the introduction of new machine and setting data for cycles, the data concept for measuring cycles has been revised. Setting data which have so far been stored in GUD variables have become machine and setting data. The Document /1/ comprises tables with the corresponding information in Attachment A:
 - Comparison of GUD parameters ⇔ Machine and setting data
 - GUD variables which are no longer used
 - Changed names of cycles and GUD blocks
2. When using measuring cycles with different systems of units (basic system <> programmed system) and programming of G commands G70 or G71, the system of units of the following tolerance parameters has changed – they now refer to the programmed system (formerly basic system): Confidence range (_TSA), zero offset range (_TZL), averaging with offset (_TMV), measure difference check (_TDIF).).
3. Notes regarding DFA with measurement programs in inch: The measurement path DFA is now also calculated as an inch value in the programmed system inch. For programs in inch which have been newly created as from SW 2.7/4.4, the DFA parameter must be adjusted accordingly.

Possibly existing programs must be adjusted accordingly for these changes using measuring cycles.

ShopMill cycles:

1. Up to SW 1.5, the reference point (X0, Y0, Z0) for ShopMill could also be indicated incrementally. In newly opened screens, the reference point can only be entered in absolute values. A toggle field abs/inc is offered in recompiled screens comprising an incremental reference point. With inc, the check outputs an error message “Reference point inc no longer supported, please convert to abs”.

3.2 Adapting machine manufacturer cycles

Note:

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

If an upgrade is only made to resolve errors (debugs) and no new functions are to be used from this software release, then the compatibility can be established as described in the following.

3.2.1 Status CUST cycles in SW 4.5 SP2

The table provides an overview of the changes in the machine manufacturer cycles from software 2.7/4.4 (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	Contains new functions and changes with respect to SW 2.7/4.4
CUST_800	Manufacturer cycle for adaptation to the function swivel plane and swivel tool (CYCLE800)	Contains new functions and changes with respect to SW 2.7/4.4
CUST_832	Manufacturer cycle to adapt the High Speed Settings function (CYCLE832)	No change with respect to SW 2.7/4.4
CUST_MEACYC	Manufacturer cycle to adapt functions for measuring cycles	Contains new functions and changes with respect to SW 2.7/4.4
CUST_T	Cycle is used to track T -preparation after SERUPRO	From SW 4.5, bit 1 in MD 20310 \$MC_TOOL_MANAGEMENT_MASK must be set
CUST_M6	Cycle is used to track the tool change after SERUPRO	No change with respect to SW 2.7/4.4
CUST_MULTICHAN	Manufacturer cycle for lathes with several channels	No change with respect to all previous releases
CUST_CLAMP	Manufacturer cycle for clamping elements	No change with respect to all previous releases

3.2.2 Special note regarding the upgrade

When upgrading from a previous release SW 2.6 or SW 2.6 SP1, all CUST cycles with the exception of CUST_MULTICHAN from SW 4.5 SP2 must be accepted and readapted!

When upgrading from previous releases from SW 2.7/4.4 and higher, compatibility can be established as described below.

3.2.3 Establishing compatibility

The machine manufacturer cycles CUST_<Name> have been function ally expanded so that after a software upgrade, existing CUST cycles from SW 2.7/4.4 are still able to run after a small **one-off (single) adaptation** (compatibility). The adaptation means that when calling new functions (new transfer values in the MODE parameter), a branch is always made to the actual CUST cycle in the standard cycle directory, which belongs to the cycle package.

Adapting means the following:

- Copying blocks from the CUST cycle, supplied in the CST.DIR and inserting in the existing cycle in the CMA.DIR
- This procedure is just once, if the cycle had a previous version < 4.5 SP2;
- All of the blocks to be copied are bracketed with


```
;BEGIN_DO_UPGRADE
;END_DO_UPGRADE
```

CUST_TECHCYC – copy 2 blocks:

- (1) Copy the first designated block and insert immediately before the instruction


```
CASE _MODE OF 1 GOTOF _M1 2 GOTOF _M2 ...
```

- (2) Copy the second designated block, and use it to replace
 SETAL(61019,"_MODE="<<_MODE) ; Parameter _MODE=n ...

CUST_800 – copy 2 blocks:

- (1) Copy the first designated block and insert directly before the 1st executable instruction
 IF ((_MODE>=21)AND(_MODE<=31)) ...
- (2) Copy the second designated block, and use it to replace
 SETAL(61019,"_MODE="<<_MODE) ; Parameter _MODE=n ...

CUST_MEACYC – copy 4 blocks:

- (1) Copy the first designated block, and insert immediately before the instruction
 IF _MODE==2 GOTOF _MEACYC_END ...
- (2) Using the three additional blocks, the blocks for alarm outputs are replaced.

```
...
N631001 SETAL(61099," Parameter _OVI[2] ")
...
N631002 SETAL(61099," Parameter _OVI[2] ")
...
N631003 SETAL(61099," Parameter _OVI[14] ")
...
```

Copy the designated block and use it to replace the instruction
 N631xxx SETAL(61019,...

4 Changes with respect to the previous release

4.1 Changes in SW 4.5 SP2 with respect to SW 4.5 SP1 HF3

New functions, functional changes / expansions, technological cycles:

- CYCLE95 at SK contour turning in the G code editor
 Can be switched with MD 52218 \$MCS_FUNCTION_MASK_TURN Bit 13 =1
- Complete machining for stock removal along the contour
- Milling cycles pockets, grooves, spigots, path milling: now monitor the TRACYL type and for active slot wall correction (type 513 or type 514 with appropriate parameters) output an alarm
- Contour milling and pre-drilling (CYCLE63, CYCLE64) – only valid in the G code: After the end of the cycle, the tool is always retracted to the retraction plane (parameter RP).
- Stock removal along the contour/contour pocket (CYCLE952, CYCLE63, CYCLE64): When calling cycles from programs that are located on an external storage medium (program called with EXTCALL) the generated programs are always saved in the TEMP.WPD – also applicable in the G code.
- Milling cycles: In all milling cycles, which are called with active COMPCAD and internally activate the milling radius correction, then COMPCAD is switched out in the cycle. It is active again at the end of the cycle.

- Monitoring tool for tapping:
In cycles CYCLE84 and CYCLE840 the pitch of the thread drill (type 240) is only monitored if a value for the pitch was entered in the tool table.
- Monitoring the tool for groove and cut-off:
In the cycles CYCLE930, CYCLE93 and CYCLE92 all turning tool types are permitted 5xx. If the tool is not a plunging tool (types 520 and 530), then the plate width is calculated from the length difference between two D numbers.

Functional changes/expansions configurable machine and setting data:

i. minimum value changed:

SD 5558100 \$SCS_TURN_CONT_RELEASE_DIST from 0 to 0.01

Secondary conditions:

Cycle packages, general

- o The cycles require the following machine data setting:
MD 20360 \$MC_TOOL_PARAMETER_DEF_MASK bit 3 = 0
(this means that in NVP in frames, it is not permissible that the face axis is set as diameter)

Technology cycles

- Manufacturer cycle CUST_800 for marker M40: a STOPRE has been incorporated.
The machine manufacturer can test whether this STOPRE for marker M40 in the CUST_800.spf is in line with his functional requirements. This case, it must be weighed up between speed (without STOPRE) and functionality for interrupt conditions (RESET) for swiveled planes). This also depends on the type of machine kinematics.

Measuring cycles

- ii. Measuring in JOG /measuring cycles with TRAORI: When measuring a workpiece with active TRAORI, the probe must be calibrated to the center of the ball (MD \$MNS_MEA_FUNCTION_MASK bit 1 = 0).
- iii. When measuring in JOG, the coupling spindle is always effective with coordinate rotation. The setting data bit SD \$SCS_MEA_FUNCTION_MASK bit 14 is ineffective.
- iv. The new setting data bits have no big description and no online help in the detail view. They can be displayed by a previous selection "display all bits".
- v. The measuring cycles always require an SPOS-capable spindle. However, when taking into account the following notes when using a workpiece probe, they can be used in a non-SPOS-capable spindle with restrictions.
- vi. (1) For this case, in order that no spindle position takes place in the measuring cycles, in machine data MD 52207 \$MCS_AXIS_USAGE_ATTRIB[Spindle] bit 9 must be set to 1
.
(2) At the instant of calibration and measuring, the user must guarantee an identical orientation (spindle position) of the probe, for example, by clamping or indexing.
(3) The CUST_MEACYC from SW 4.5 must be executed! If an older version is in the manufacturer or user directory, then this cycle should again be taken from the standard directory and if required, adapted.
(4) The following measuring versions cannot be executed for a non-SPOS-capable spindle, they result in incorrect measurement results:
 - o Calibration with unknown center point in the ring
 - o Calibration at the ball
 - o Measuring with spindle reversal
 - o Spindle coupling with a coordinate rotation around the feed axis

4.2 Notes for using this cycle package

4.2.1 Notes regarding the boring cycle (CYCLE86)

If the "adapting the spindle position to the tool orientation" function is activated in CYCLE86 (boring) (SD 55216 \$SCS_FUNCTION_MASK_DRILL_SET Bit6=1), then internally, a correction value is calculated, and saved in the GUD variable `_MEA_CORR_ANGLE[1]`.

As it may be necessary, for selected machine kinematics, to additionally adapt this correction angle for spindle positioning, after this calculation, the `CUST_TECHCYC` is called at Marker `_M300`. There, you have the possibility of describing the correction angle `_MEA_CORR_ANGLE[0]` (for the manufacturer) and `_MEA_CORR_ANGLE[1]` (system). For subsequent spindle positioning operations, both angles act additively in CYCLE86.

5 Attachment_1: Compatibility list, machine and setting data for technological cycles

Validity:

This list describes the assignment of setting parameters in the GUDs up to cycles SW 7.5 (complete status SW 1.5 of 840Dsl) to the machine and setting data from SW 2.6 (This is also applicable for SW 2.7/4.4 and SW 4.5)

5.1 Setting data that are no longer applicable

GUD name up to SW 7.5	Function, significance
<code>_ZSD[4]</code>	=1: CYCLE93 retraction after the 1st plunge with G0 =0: Retraction with G1
<code>_ZSD[5]</code>	=1: CYCLE88 is only executed with M5 =0: CYCLE88 is executed with M5 M0
<code>_ZSD[6]</code>	=1: CYCLE93 In the cycle with active mirroring, tool offsets are not exchanged =0: Tool offsets are exchanged
<code>_ZSD[9]</code>	=1: POCKET3 When measuring the square pocket above the corner, the angle of rotation refers to the center of the pocket =0: When measuring the rectangular pocket above the corner, the angle of rotation refers to this reference point

5.2 New setting data

MD/SD number	MD/SD name and bit	Function, significance	GUD name up to SW 7.5
General cycle settings			
	No longer applicable, is now programmed	=2: CYCLE95 infeed MID is the diameter value =1: Infeed MID is the radius value =0: Infeed is active corresponding to the active radius- /diameter programming before the call	<code>_ZSD[0]</code>

SD 55214	\$SCS_FUNCTION_MASK_MILL_SET, Bit 2	=1: Depth calculation without taking into account the safety clearance =0: Depth calculation between the reference plane + safety clearance and final depth	_ZSD[1]
	No longer applicable, is now programmed	=1: Measuring a rectangular pocket/rectangular spigot from a corner =0: Measuring a rectangular pocket /rectangular spigot from the center point	_ZSD[2]
SD 55216	\$SCS_FUNCTION_MASK_DRILL_SET, Bit 0	=1: CYCLE84 The programmed sign of the incline is reversed in the cycle. =0: The programmed sign of the incline is kept in the cycle.	_ZSD[7]
Thread table for thread cutting (CYCLE97)			
SD 55218	\$SCS_FUNCTION_MASK_TURN_SET, Bit 0=1	CYCLE97: New thread table according to DIN 13-1 corresponding to edition 11.1999	_SC_MASK[2]
Tapping (CYCLE84/CYCLE840)			
SD 55481	\$SCS_DRILL_TAPPING_SET_GG12[0]	Setting for tapping: G-group 12: Block change response for exact stop	_SC_MASK[0] _SC_MASK[1]
SD 55482	\$SCS_DRILL_TAPPING_SET_GG21[0]	Setting for tapping: G group 21: Acceleration profile	_SC_MASK[0] _SC_MASK[1]
SD 55483	\$SCS_DRILL_TAPPING_SET_GG24[0]	Setting for tapping: G group 24: Precontrol	_SC_MASK[0] _SC_MASK[1]
SD 55484	\$SCS_DRILL_TAPPING_SET_MC[0]	Setting for tapping spindle operation for MCALL	_SC_MASK[0] _SC_MASK[1]

Notes:

(1) SD 55481 ... SD 55484 are only effective in conjunction with MD 52216 \$MCS_FUNCTION_MASK_DRILL bit0 and bit1; description of the function: IM9 Chapter 20

6 Attachment_2: assignment list GUD parameter ↔ machine data for ISO cycles

Validity:

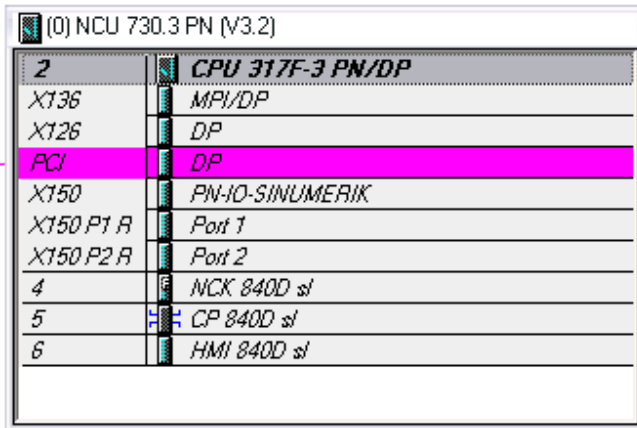
This list describes the assignment of setting parameters in the GUDs up to SW 6.5 (complete release SW 1.5 of 840Dsl) to the machine and setting data from SW 2.6 (also valid for SW 2.7/4.4 and SW 4.5)

GUD up to version 6.5	MD/SD number	MD/SD name
_ZSFI[0]	SD 55800	\$SCS_ISO_M_DRILLING_AXIS_IS_Z
_ZSFI[1]	SD 55802	\$SCS_ISO_M_DRILLING_TYPE
_ZSFI[2]	SD 55804	\$SCS_ISO_M_RETRACTION_FACTOR
_ZSFI[3]	MD 52800	\$MCS_ISO_M_ENABLE_POLAR_COORD
_ZSFI[4]	MD 52802	\$MCS_ISO_ENABLE_INTERRUPTS
_ZSFI[5]	SD 55806	\$SCS_ISO_M_RETRACTION_DIR
_ZSFI[6]	MD 52804	\$MCS_ISO_ENABLE_DRYRUN
_ZSFI[7]		
_ZSFI[8]	MD 52808	\$MCS_ISO_SIMULTAN_AXES_START
_ZSFI[9] ... _ZSFI[19]		
_ZSFI[20]	MD 52810	\$MCS_ISO_T_DEEPPHOLE_DRILL_MODE

_ZSFI[21]		
_ZSFI[22]	SD 55808	\$SCS_ISO_T_RETRACTION_FACTOR
_ZSFI[23]	SD 55810	\$SCS_ISO_T_DWELL_TIME_UNIT
_ZSFI[24] ... _ZSFI[38]		
_ZSFI[39]	MD 10881	\$MN_MM_EXTERN_GCODE_SYSTEM

Notes regarding the integrated SIMATIC CPU 317 F-3 PN/DP

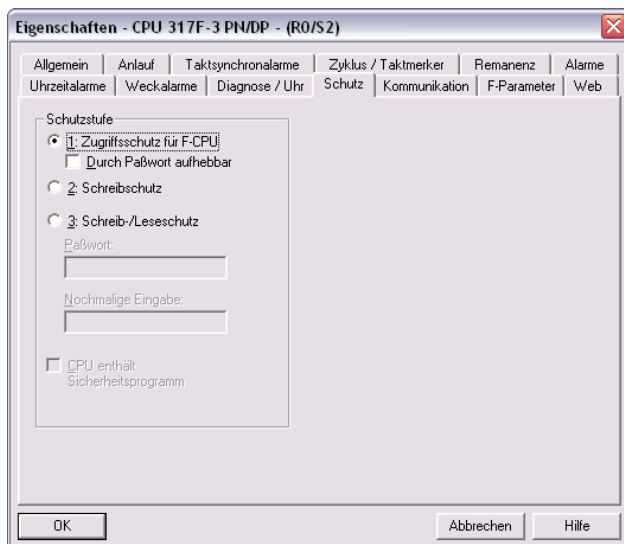
A failsafe **CPU 317 F** is integrated in the SINUMERIK 840D sl as from the hardware platform **NCU 7x0.3** (Software Version 4.x).



The safety Integrated functions known from the SIMATIC F-CPU **cannot** be used in the SINUMERIK environment.

If the software module “S7 Distributed Safety Programming” is installed in Step 7, you can generate and program a F-Program, as known from the SIMATIC. However, you cannot load such a program into a SINUMERIK.

To obtain an identical system environment as in the hardware and software versions lower than Software Version 4.x, we recommend that you leave the “**Protection**” tab in the properties of the CPU 317 F on the “**Standard setting**”.



Creation of a hardware configuration for F-Modules

You still have to install the “S7 F Configuration Pack” as a minimum requirement for creating a hardware configuration with F-Modules.

Depending on an additionally installed software package “S7 Distributed Safety Programming”, please observe the following.

S7 Distributed Safety Programming installed on the PC/PG

If the software module “S7 Distributed Safety Programming” is installed in Step 7, the programmer is requested to enter a password which entitles him to set up a safety program, as soon as he wants to enter, display or modify the safety settings in the HW Config.

Password assignment prompt.

Press the “Abort” softkey to confirm this prompt.

You are always prompted to enter a password when creating a new project resp. modifying an existing project and **no** password has been assigned before.

Note

You need not enter a password when using the Safety Integrated functions of the SINUMERIK 840D sl!

If you have assigned a password here, this cannot be undone!

If you have assigned a password, you must always enter this password when subsequently modifying the HW Config. Without entering the password, you have only read-only access to the parameter setting of the F-Modules.

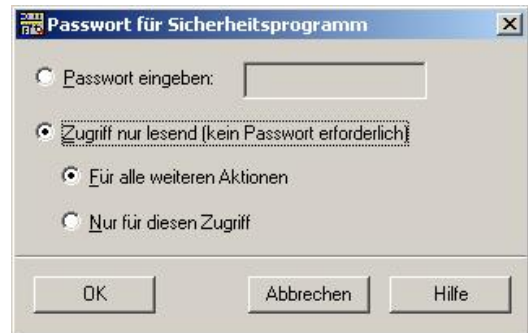
The “S7 Distributed Safety Programming” is required to modify the HW Config. The “S7 F Configuration Pack” no longer suffices.

If “S7 Distributed Safety Programming” is installed on the PC/PG and a password assigned in the screen “Authorization for setting up safety program” (see Page 2), the following prompt is output when subsequently modifying the HW Config:

Enter password:



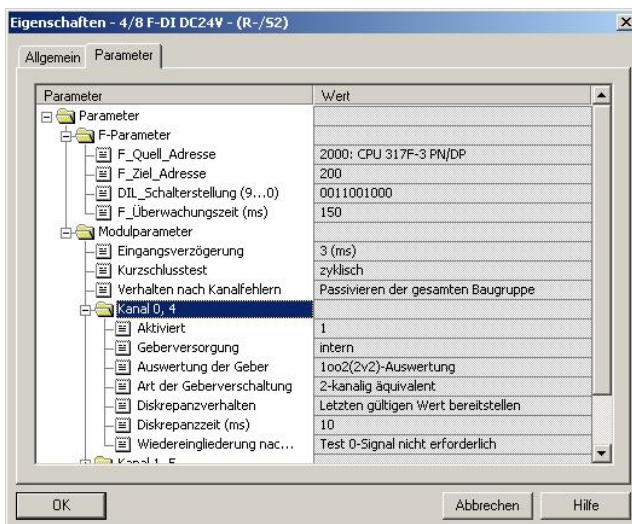
Read-only access



Note:

If a password has been assigned for the HW Config, which is not known, or when opening a project with password in a Step 7 environment in which only the software module "F Configuration Pack" is installed, you have solely read-only access to the F-Module parameters.

Example: Solely read-only access to the fields highlighted in gray



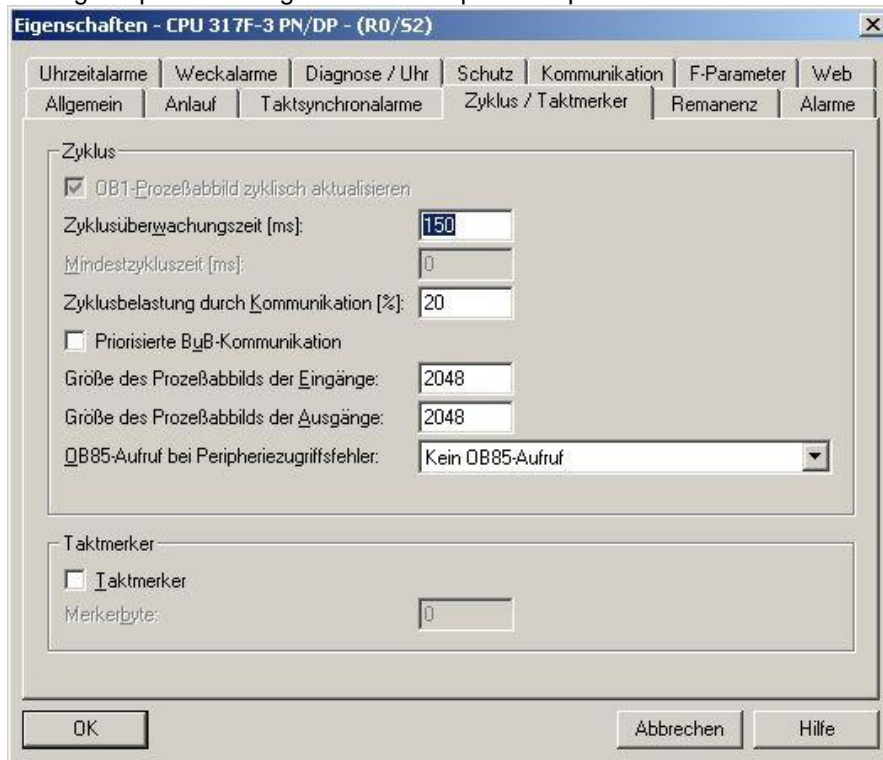
Only the Software Module "S7 F Configuration Pack" is installed on the PC/PG (without S7 Distributed Safety Programming)

In this case, you are **not** prompted to enter a password when creating the hardware configuration.

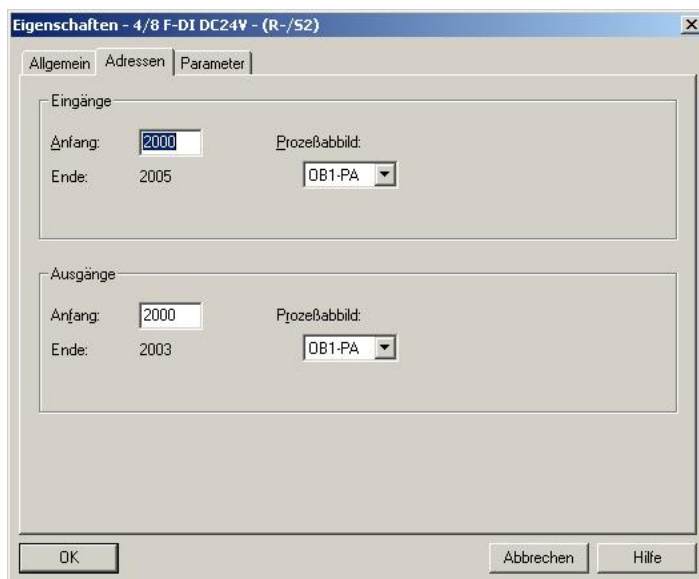
Entering the address areas of the F-Modules

On the CPU 317 F, the address areas of the F-Modules must be within the process image. The available address area can be set in the properties of the CPU 317 F. The default setting for the address area is 1024, which can be extended to 4079.

Setting the process image size of the inputs / outputs in the CPU 317 F



Setting the initial module addresses in a F-Module



When parameterizing the address areas of the F-Modules outside the process image, the following message is output upon storage and compilation:

