

**SINUMERIK 840D sl**

CNC Software 4.8 SP3 HF1

Contents of the installation and upgrade instructions

1	SOFTWARE COMPONENTS .....	3
2	NEW FUNCTIONS.....	4
3	PREREQUISITES .....	5
3.1	Software .....	5
3.2	Hardware.....	5
3.3	Storage media.....	5
3.4	Tools .....	5
3.5	Other Requirements.....	5
4	INSTALLATION.....	6
4.1	Installing the CNC software .....	6
4.2	Boot code of the PLC .....	6
4.3	Installing the NCU service system on USB stick .....	6
5	Notes and restrictions.....	7
5.1	General notes .....	7
5.2	Notes regarding the CNC software .....	7
5.3	Notes regarding SINUMERIK STEP 7 Toolbox V15.....	13
5.4	Notes regarding Toolbox STEP 7 classic V04.08.08.....	13
5.5	Notes regarding the use of the Starter with SINUMERIK .....	15
5.6	Notes regarding SINUMERIK Operate.....	15
5.7	Note regarding the Siemens cycle packages .....	18

**SINUMERIK 840D sl**  
CNC Software 4.8 SP3 HF1

This document serves as a guide for the software update to SINUMERIK 840D sl CNC Software 4.8 SP3 HF1.

We kindly request you to carefully read through this document, as it comprises essential information for the installation and use of the software.

The following binding notes supersede the statements made in other documents. Additional information is provided in the following documents:

Name	File
Function improvements	Behobene_Funktionseinschraenkungen.pdf
Supplementary conditions	Randbedingungen.pdf (like SW 4.8 SP3)
Cycles	siemensd/e_appendix_cycles.pdf (like SW 4.8 SP3)
Emergency Boot System	Emergency Boot System.pdf (like SW 4.8 SP3)

**SINUMERIK 840D sl**  
CNC Software 4.8 SP3 HF1**1 SOFTWARE COMPONENTS**

The CNC software 4.8 SP3 HF1 (internal version 04.08.03.01.001) comprises the following components:

Component	Version
NCK (incl. NRK)	V128.01.02
SNCK	V04.08.00.00.001
SINAMICS	V05.10.40.02
PLC OpSys (317-3)	V32.84.03
PLC OpSys (319-3)	V32.84.03
FB15(sl)	V04.08.04
PLC_F_COPY	01.05.00
CP	V02.42.03
Diagnosis Data Collector	V02.03.02.00
NCK communication server	V01.03.00
DiagSrv application	V01.16.00
MCP_CLIENT	V01.06.13
SINUMERIK Operate	V04.08.03.01.001
Linux basic system	V04.71.42.00
NCK file system driver	V04.07.01.06.001
NCU-Link Config (SDBs for the configuration of NCU-Link)	V03.00.00
Profinet FW	14.01.06.04
Cycles	V04.08.03.00.97
HMIARC Tool	04.07.02.03.001
SinIntClient	03.00.12.00.012
OPC-UA Server	02.01.00.00.019

## **2 NEW FUNCTIONS**

New functions from SW4.8 SP3:

- Support for the 1FK2 motor series
- SINAMICS armature short-circuit braking for SINUMERIK can be activated
- Measuring cycles innovation: simultaneous measurement for double spindles
- SINUMERIK Operate Display Manager for the internal HMI
- Support for SIMATIC Thin Clients ITC V3
- Contains new OPC UA version V2.1
- New Protector V02.00.03
- New "Advanced Position Control ECO" option, 6FC5800-0AM12-0YB0

### **3 PREREQUISITES**

#### **3.1 Software**

- NCU Service System >= V04.71.24.01
- When using SINUMERIK Operate on PCU 50 / IPC / PC, version V4.8 SP3 HF1 must be used.

#### **3.2 Hardware**

- The CNC SW 4.8 SP3 may only be operated on SINUMERIK NCUs 710.3 PN / 710.3B PN, 720.3 PN / 720.3B PN and 730.3 PN / 730.3B PN.
- NCU 7x0.3 modules with boot code of PLC Version V2.5 (see also Chap. "Boot code ...")

#### **3.3 Storage media**

- 8 GB CF Card Smart Modular Xceed. - For the standard version 6FC5851-1XG45-\_YA\_.

#### **3.4 Tools**

- STEP 7 version as of V5.5 SP4;  
on Windows 10 with STEP 7 version as of V5.6 HF4
- Toolbox STEP7 classic V04.08.08.00
- SINUMERIK STEP 7 Toolbox V15 (TIA Portal) with the internal version 15.00.00.00\_26.02.0001
- Create MyConfig V4.8 or later
- NCU service system V04.71.24.01

#### **3.5 Other Requirements**

- For the internal use of SINUMERIK Operate on NCU, you require the option S00 SINUMERIK Operate on NCU in combination with real drives
- For the external use of SINUMERIK Operate on PCU or PC, you require the option P87/P88 SINUMERIK Operate on PCU / PC in combination with real drives

## 4 INSTALLATION

### 4.1 Installing the CNC software

The NCU Service System >= V04.70.47 on USB stick is required to install the CNC software. New installations can be performed via the NCU service system and TCU or with WinSCP / AMM / P2P from V4.6 SP1 HFx.

An upgrade or new installation can also be carried out using the tools of the Create MyConfig Version from V4.7 SP2. To do this, a CMC package must be generated that is stored and booted either on a PG, PC, a PCU or a USB stick at the NCU or PCU. The relevant instructions are included in the Create MyConfig manual.

The software may only be installed on a SINUMERIK 8 GB CF card - see requirements.

### 4.2 Boot code of the PLC

The Software Version 4.8 only starts with PLC boot code V2.5 (or later) internally coded on the NCU hardware. If this PLC boot code has not yet been installed, the control will not start (in this case, the 7-segment display shows "PLC" and SF and DP LED flash red).

In this state, the boot code upgrade can be initiated by actuating the PLC rotary switch. The boot code is flashed by turning the PLC rotary switch into position 5. This can be identified by the fact that a rotating wheel is displayed in the 7-segment display. This takes approx. 15 s. 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 turned back to position 3 (general reset) and the system continues to boot.

### 4.3 Installing the NCU service system on USB stick

The NCU service system is stored as a USB stick image in the directory emergency\_bootsys\_ncu on the supplied DVD. The copy program installdisk.exe is also included. Connect a USB stick >= 2 GB to your PG respectively PC and determine the relevant drive letters. Call installdisk to copy the NCU service system to the USB stick:

```
installdisk --verbose --blocksize 1m <Image file> <Drive letter>
```

It is best if the command specified above is executed in a DOS shell. For this, you require administrator rights on your PG/PC.

Sequence of screens for the NCU service system – Generate a separate file "Emergency Boot System.pdf"

Alternatively, the image can also be installed using the tool AccessMyMachine /P2P as of V4.6 SP1 HFx via the menu item "Write file system image".

Installation on the NCU:

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. Under Windows, you can now see an empty USB stick. Now copy the requested CNC software (file with extension .tgz) from the directory ncu\_sw on the DVD supplied. The USB stick can be connected to one of the two USB connectors of the NCU 7x0.3. The NCU will boot from this USB stick after power ON. The system is operated either via a TCU that is connected to the NCU, or via PG / PC using WinSCP under "Open Terminal". Instructions on how to use the NCU service system can be found in the documentation /IM7/ on the DVD supplied. We recommend that you use the USB stick "Service USB flash drive, 32 GB, USB 3.0", 6ES7648-6XA11-0YA.

## 5 NOTES AND RESTRICTIONS

### 5.1 General notes

Siemens observes standard data protection principles, in particular the principle of privacy by design. That means that this product does not process / store any personal data, only technical functional data (e.g. time stamps). If a user links this data with other data (e.g. a shift schedule) or stores personal data on the same storage medium (e.g. hard drive) and thus establishes a link to a person or persons, then the user is responsible for ensuring compliance with the relevant data protection regulations.

The HMI Pro sl software as of V8.0 HF1 can be used in combination with the CNC SW 4.8 SP3.

Language extensions V4.8.2, V4.8.3 and, where necessary, V4.7.3 can be used. New texts from SW 4.8 SP3 may be displayed in English.

The Vietnamese language extension for 4.7 SP3 can only be used if the new skin is NOT active.

The display MD 9112 must be set to 0 for Vietnamese.

The following functions / options are not released:

- HTL/TTL encoder in combination with HLA and safety
- The CBA function may only be used together with a declaration regarding machine-specific release
- Collision Avoidance Advanced option 6FC5800-0AS04-0YB0 has not been generally released

### 5.2 Notes regarding the CNC software

New password guidelines are applicable from SW 4.8 SP2.

When changing an NC password, the password for the associated Linux account is also changed.

This means that when changing the manufacturer's password, the password for the associated Linux account "manufact" is also changed.

IMPORTANT: After changing one of the existing NC passwords (e.g. SUNRISE), it is no longer possible to return to the old password; the reason for this is that it no longer conforms with the new guidelines (minimum length, 8 characters, uppercase letter, lowercase letter, number).

If an access level in NCK should be set from the PLC via FB4 (PI service \_N\_LOGIN), then a password valid for the NCK (and at the same time Linux) should only be precisely 8 characters long.

This case must be taken into account when assigning new passwords. Otherwise, passwords can have 8 - 32 characters.

As of SW 4.7 SP3 HF1, the option S17 Top Surface is generally available; the following notes must be taken into account:

- Operation only alternatively to Advanced Surface with moldmaking applications (moldmaking involves short blocks from CAD/CAM systems only a few millimeters long).
- 
- The setting instructions for Advanced Surface and Top Surface must be complied with – see <https://support.industry.siemens.com/cs/ww/de/view/109738423>

The current secondary conditions for Advanced Surface and Top Surface must be observed.

- Operation only with NCU 7x0.3B hardware, the NCU 710.3B hardware is only conditionally suited for Top Surface.

**SINUMERIK 840D sl**

CNC Software 4.8 SP3 HF1

- In terms of resources, 5(6)-axis simultaneous machining should only be implemented with 720/730.3B hardware.
- Top Surface may only be configured in the first and another channel and only active in a single channel.
- Collision Avoidance (ECO) or Collision Avoidance (Advanced) can be used (active) in the AUTOMATIC mode, but not the Collision Avoidance (Standard)
- NCU utilization: IPO/position controller and possibly PLC approx. 3 0% (visible under system utilization diagnostics)
- Operation only with 3D-TRC differential tools other than CUT3DFD or CUT3DCD

With Top Surface, the tolerance value from CYCLE832 is used for the contour – in contrast to Advanced Surface, where the tolerance value is used for the axis tolerance. When comparing Advanced Surface with Top Surface, this yields a difference of  $\sqrt{3}$  between the active contour tolerance and possibly surface and speed respectively machining time differences.

With Top Surface, an enhanced jerk consideration is performed, compared to Advanced Surface.

If the data throughput of data-intensive part programs leads to a reduction of the IPO buffer filling level, the FIFOCTRL function reduces the path velocity in order to prevent stopping of the system.

- The Collision Avoidance ECO function is generally available, taking into account the following information:

- Operation for 840D sl only with NCU720.3B or 730.3B hardware
- Collision Avoidance ECO may only be used in single-channel systems
- Machine modeling is only permitted using primitive basic solids (cubes, cylinders, spheres)
- A maximum of 34 primitives, 17 protection zones and 10 collision pairs is permitted

The recommended settings can be downloaded from Siemens Industry Online Support (SIOS).

- The function "Configuring static and dynamic transformations via kinematic chains" was completed with CNC SW 4.8 SP2.  
The launch should first be accompanied by the Siemens Technology and Application Center (TAC) (Mr. Claudio Jans, [claudio.jans@siemens.com](mailto:claudio.jans@siemens.com)).

- \$NT\_POLE\_TOL[transformer no.] previously had 1 as the default. This has been changed to 0.

- When upgrading the system from V4.5 SPx and having licensed all options, the options P87 and P88 (Operate on PC respectively PCU 50) may have to be licensed subsequently following the upgrade.

- If an upgrade is made from SW 4.7 within an active test license period (trial license), then the actual remaining time is shortened to 1/10. The next period then has the normal length.

- Auto Servo Tuning – AST

When upgrading the system from CNC SW < V4.5 SP3, the stored AST session files (Auto Servo Tuning) are not compatible and the AST settings (strategy, measurement parameters, etc.) get lost.

The standard directory for XML files has been modified from card/user/sinumerik/hmi/log/optimization (SW 4.5) to card/user/sinumerik/nck/data/optimization (as of SW 4.7SP2). You can perform an XML export. The files are stored under card/user/sinumerik/hmi/log/optimization. When using "Clear History", a backup is stored under card/user/sinumerik/nck/data/optimization/backup.

DDS0, and not the DDS currently selected via PLC/NC, is always pre-selected under "Active filter optimization". The DDS must be explicitly selected via "Filter group".

AST filter settings: Changes are discarded when switching to another axis, for example. This also applies to filter groups and DDS settings.

Commissioning wizard: Drive wizard does not function for axes on CU320-PN with plugged TB30 modules.



## SINUMERIK 840D sl

CNC Software 4.8 SP3 HF1

- SinuComARC  
Series commissioning archives can no longer be edited with SinuComArc.  
Alternatively, you can use Create MyConfig (for example, also CMC Diff) respectively AMM /P2P as of V4.7 SP2; does not claim to be a successful competitor.
- Restrictions with EES:  
When using a USB stick on the TCU in combination with a PCU, you can currently not edit any files on the USB stick with active EES mode. These files can, however, be executed.  
If a USB stick on a TCU is accessed by several HMIs / NCUs in parallel, the components are not coordinated. This entails the danger that a program being processed is modified / destroyed by another component.
- Problem solution for S5\_FETCH / S5\_WRITE in the CP:  
As of SW 4.7 SP1, the file cp\_param.ini containing the following contents is stored on the CF card in the directory /siemens/sinumerk/cp:  

```
[IniFile_ID]
Version = 100
Type   = 'CosCP'
Comment = 'Select fetch/write version'
#####
#### If you modify this file,                #####
#### - don't remove the section [IniFile_ID] above      #####
#### - check 'cp_param.log' to verify that your settings are accepted.  #####
#####
```

```
[FetchWrite]
# Selection of the S5 fetch/write implementation
# Version = 1 : the previous behavior
# Version = 2 : the new as far as possible CP 343 compatible behavior
Version = 1
```

  
The problem solution becomes active when recopying this file (possible as manufact) into the directory /user/sinumerik/cp and modifying the entry "Version = 1" in "Version = 2".
- Safety: Note on configuring Stop D delay times on axes that belong to a master-slave network:  
With this software version, the slave-side MD setting in SAFE\_STOP\_SWITCH\_TIME\_D is no longer taken into account in master-slave networks **when the slave axis is operated without Safety**.  
This may result in different braking times after activating a Stop D.  
This only applies if a higher Stop D time has erroneously been configured on the master axis than on the slave axis. If this is the case, the Stop D time for the master axis must be adjusted and confirmed.
- Safety: When traversing at low speeds, alarm 27011 is triggered during traversing movements in the negative direction.  
The following supplementary conditions must be considered a remedy for the described behavior:  
The minimum possible resolution (quantization) of the monitoring functions for Safety Integrated is one increment per monitoring cycle [incr/mcc].  
Two increments per monitoring cycle are necessary, so that no speed violation alarm is triggered when the axis is stationary.  
The minimum speed limit value that can be set for the machine data \$MA\_SAFE\_VELO\_LIMIT can be determined as follows:  
For a rotary axis:

**SINUMERIK 840D sl**

CNC Software 4.8 SP3 HF1

$$\text{\$MA\_SAFE\_VELO\_LIMIT[rpm]} = 2[\text{incr/mcc}] * 60[\text{s/min}] / 360[\text{deg/rev}] / 1000[\text{incr/deg}] / \text{MCC}[\text{s/mcc}]$$

Example 1: Monitoring cycle = 12 ms, LimitMin=2[incr/mcc]

$$\text{\$MA\_SAFE\_VELO\_LIMIT} = 2[\text{incr/mcc}] * 60[\text{s/min}] / 360[\text{deg/rev}] / 1000[\text{incr/deg}] / 0.012[\text{s/mcc}] = 0.02777.. [\text{rpm}]$$

=> rounded 0.028[rpm]

Example 2: Monitoring cycle = 8 ms, LimitMin=2[incr/mcc]

$$\text{\$MA\_SAFE\_VELO\_LIMIT} = 2[\text{incr/MC}] * 60[\text{s/min}] / 360[\text{deg/rev}] / 1000[\text{incr/deg}] / 0.008[\text{s/mcc}] = 0.04166.. [\text{rpm}]$$

=> rounded 0.042[rpm]

With decimal places, the value has to be rounded to be entered for the machine data.

For a linear axis:

$$\text{\$MA\_SAFE\_VELO\_LIMIT[mm/min]} = 2[\text{incr/mcc}] * 60[\text{s/min}] / 1000[\text{incr/mm}] / \text{MCC}[\text{s/mcc}]$$

Example 1: Monitoring clock cycle = 12 ms, LimitMin=2[incr/mcc]

$$\text{\$MA\_SAFE\_VELO\_LIMIT[mm/min]} = 2[\text{incr/mcc}] * 60[\text{s/min}] / 1000[\text{incr/mm}] / 0.012[\text{s/mcc}] = 10[\text{mm/min}]$$

Example 2: Monitoring clock cycle = 8 ms, LimitMin=2[incr/mcc]

$$\text{\$MA\_SAFE\_VELO\_LIMIT[mm/min]} = 2[\text{incr/mcc}] * 60[\text{s/min}] / 1000[\text{incr/mm}] / 0.008[\text{s/mcc}] = 15[\text{mm/min}]$$

With decimal places, the value has to be rounded to be entered for the machine data.

Actual speed value:

At very low speeds, the possible minimum resolution of the actual speed value must also be considered; this value depends on the encoder pulses per revolution, the gear ratio and the monitoring clock cycle!

SINAMICS display parameters for the speed resolution:

The drive provides a display parameter for the speed resolution, which is also visible with ncSI

r9732[0..1] SI Motion speed resolution

For Index 0:

Display of the safe speed resolution (load side). Specification of speed limits or parameter changes for speeds below this threshold have no effect.

For Index 1: Display of the safe speed accuracy based on the safe encoder accuracy. Unit: mm/min or rpm

Use with ncSI:

With a 1-encoder system, the above-mentioned display parameter r9732[0..1] "SI Motion speed resolution" can be used as a basis for the setting of the minimum limit values by the user. You must double the value displayed in r9732[0] for parameterization, so that no speed violation alarm is triggered when the axis is stationary.

With a 2-encoder system, the value displayed for r9732 is only valid for the encoder parameterized for SINAMICS. The speed resolution of the encoder for the NCK can vary from this value!

As already stated above, this parameter alone does not provide any information on the actual accuracy of the speed measurement. This depends on the type of actual value sensing, the gear ratios and the quality of the encoders used.

- Adjustment of the default machine data in the area "Saving of persistent data".  
With the values set, the number of write processes onto the CF card has been reduced.

a) Change of the memory default values for saving persistent data on the system CF card

	Old	New
N17610 \$MN_DEPTH_OF_LOGFILE_OPT_PF[0]=	10	100
N17610 \$MN_DEPTH_OF_LOGFILE_OPT_PF[1]=	0	10
N17610 \$MN_DEPTH_OF_LOGFILE_OPT_PF[2]=	0	30
N18232 \$MN_MM_ACTFILESYS_LOG_FILE_MEM[0]=	2	400
N18232 \$MN_MM_ACTFILESYS_LOG_FILE_MEM[1]=	3	50
N18232 \$MN_MM_ACTFILESYS_LOG_FILE_MEM[2]=	3	30

**SINUMERIK 840D sl**

CNC Software 4.8 SP3 HF1

- b) In order to keep the preprocess running during the Flush process, the system has been set to asynchronous flush by default.

N18234 \$MN\_MM\_MEMORY\_CONFIG\_MASK= 1

**Note:**

After loading the archive, the new default values remain only topical if the “old” default values are not changed and an archive has been created with

MD11210=FF

MD11212=1.

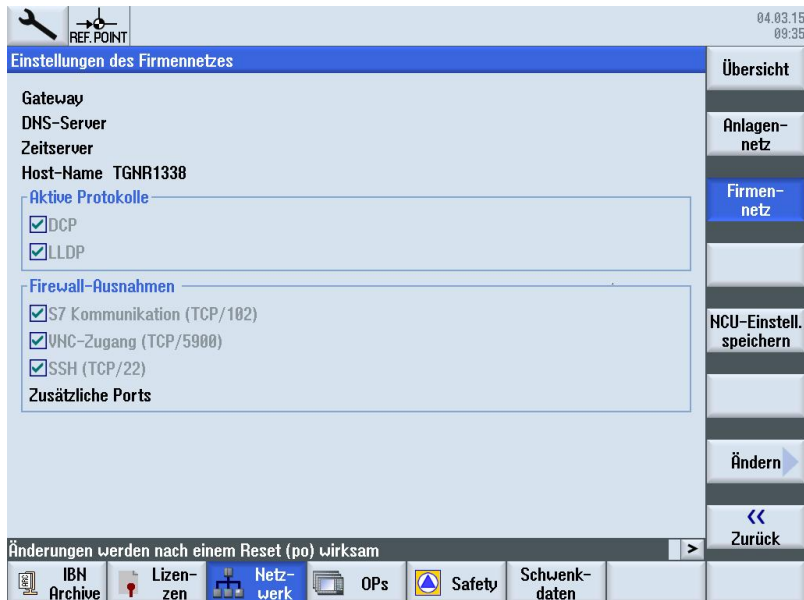
- When activating TRAILON using a synchronized action with active transformation (TRAORI), an incorrect coupling may occur if a RESET is triggered with active coupling. In case of a RESET, the transformation is deactivated for an IPO cycle. As a result, a setpoint is specified in the MCS instead of BCS. Thus, the coupling is made for the machine axis instead of the geometry axis, which leads to a position jump. TRAILON interpolates this jump and generates a movement to an incorrect coupling position (MCS), which leads to large and quick axis movements.  
Solution:  
The user must ensure that the transformation is active with activated TRAILON and the triggering / setting of RESET is suppressed in the PLC.
- TRANSMIT and Safety:  
When performing machining operations close to the pole, the feedrate is reduced. With the setting \$MC\_LOOKAH\_FUNKTION\_MASK Bit1=1, the feedrate is no longer reduced.
- 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 (approximate positioning 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.
- If different arithmetic resolutions have been set for rotary and linear axes using \$MN\_INT\_INCR\_PER\_MM and \$MN\_INT\_INCR\_PER\_DEG, 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.
- Safety/Diagnostics: No display of the SAFE.SPF checksum  
If, in SAFE.SPF, the line with the variable SAFE\_CHECKSUM is not displayed in line 1, no checksum is displayed for SAFE.SPF under Safety/DIAGNOSTICS/ checksums.  
Solution:  
; SAFE\_CHECKSUM = xxxxxx  
must be displayed in the first line.
- ET200-PN as NCK I/Os:  
The plausibility check in the NCK can be deactivated via MD 11415, Bit 24. Thus, the ET200SP PN modules can be used as clock-synchronous I/Os on the Sinumerik. However, the configuring engineer must ensure that his STEP 7 hardware configuration is consistent and conforming with Sinumerik (Ti, To and Tdp must always have the same values for drives / I/Os which must be served by the NCK).
- SSH port 22 on X130 interface:  
For security reasons, the SSH port 22 on the X130 interface is disabled in the firewall as of SW 4.7 SP1. As a consequence, the controller can no longer be accessed from external sources via this interface. This concerns, among

## SINUMERIK 840D sl

CNC Software 4.8 SP3 HF1

others, Operate on PCU/IPC/PC (which uses this for several functions), WinSCP and AMM.

The port can be permanently activated in Operate (only possible on the NCU) under Commissioning -> Network -> Company Network -> Change -> Checkbox "SSH (TCP/22)", see screenshot:



Alternatively, you can make a manual entry in the file `/user/system/etc/basesys.ini` (such as the ports 102 and 5900 are activated).

- The content of `/siemens/sinumerik/mcp_client/mcp_client.ini` has been changed to support the new "WarningTimeout" parameter. All users who have their own `mcp_client.ini` in `/user/sinumerik/mcp_client` must therefore take the new file from `/siemens/sinumerik/mcp_client` and import their changes there. Otherwise, an MCP will no longer work on X120.
- The MD13200 MEAS\_PROBE\_LOW\_ACTIVE now also works for simulated probes that are configured via MD13230 MEAS\_PROBE\_SOURCE > 0. The level states of such simulated probes are displayed just like they are for real probes as a function of MD13200 in DB10.DBB107, and are therefore checked with regard to the permissible deflection state before the measuring process begins. Depending on the deflection state, alarm 21700 may be triggered during this check. To avoid alarm 21700 being triggered, either the setting in MD13200 must be taken into account when specifying the simulated level states via digital outputs or the MD13200 must be adjusted to the required level states.
- In conjunction with jerk filter type MD32402=2, the programmed thread start position SF was not observed for G33 with SPCOF (position coupling of spindle actual value with axis setpoint).
- The thread start position is now observed independently of the programmed speed.
- Now, the passwords for NCK and the corresponding Linux accounts of the NCU basis system are changed using the Operate dialog "Change password".
- Tool manager
  - The number of adjacent locations was increased from 7 to 11

**SINUMERIK 840D sl**

CNC Software 4.8 SP3 HF1

- "Monifact" function:

Monifact has been taken out of the TOA context, and can be completely handled on a channel-for-channel basis. The active Monifact value can be read in the channel-specific status data (block C/S) "aMonifact".

Now, the tool lifetimes are not converted at the OPI (block T/TS).

This means, that in Operate the tool lifetimes are no longer evaluated with MONIFACT – i.e. they are shown in real time.

If the real remaining lifetime of a tool is to be specified, taking into account \$A\_MONIFACT, then the application must link the remaining lifetime (\$TC\_MOP2) with \$A\_MONIFACT

The tool lifetime monitoring is evaluated, unchanged using the Monifact value (i.e. part programs are or remain fully compatible).

- The "flat D numbers" function (N18102 \$MN\_MM\_TYPE\_OF\_CUTTING\_EDGE=1) is no longer available as of SW 4.8 SP3 and is no longer supported by Operate.

### 5.3 Notes regarding SINUMERIK STEP 7 Toolbox V15

SINUMERIK STEP 7 Toolbox is generally released with version V15.

This toolbox can thus be used for engineering SINUMERIK hardware.

The software is an option package for TIA Portal STEP 7 Professional V15.

The following hardware is supported:

- NCU 710.3 PN with PLC317F-3 PN/DP
- NCU 720.3 PN with PLC317F-3 PN/DP
- NCU 730.3 PN with PLC317F-3 PN/DP
- NCU 730.3 PN with PLC319F-3 PN/DP
- NX10.3
- NX15.3
- ADI4

Supported functions:

- Creating a hardware configuration
- Support of software versions 4.5, 4.7, 4.8
- Basic PLC program libraries for software versions 4.5, 4.7, 4.8
- Configuring the drive telegrams (telegram configuration)
- Migrating STEP7 Classic projects from STEP7 5.4 SP5
- Support of 'SINUMERIK Safety Integrated (SPL)' and 'SINUMERIK Safety Integrated plus'
- Exporting PLC symbols for SINUMERIK Operate
- Generating SINUMERIK PLC archives

Note:

It is not permissible that projects, which were created for SW 4.7 and for which "SINUMERIK Safety Integrated plus" is activated, are loaded to an NCU 7x0.3. These projects must first be upgraded to SW 4.8 using the "Replace device" function.

### 5.4 Notes regarding Toolbox STEP 7 classic V04.08.08

In conjunction with toolbox V04.08.08, we recommend the use of STEP 7 from V5.5 SP4 HF7 with Windows 7.

**SINUMERIK 840D sl**CNC Software 4.8 SP3 HF1

---

On Windows 10, STEP 7 from V5.6 HF4 should be used.

If no new functions from Software 4.8 SP3 are used, you need not upgrade the PLC user program to the basic program version 4.8. PLC basic user programs Version 4.5 (or later) are functional. PLC user programs with basic PLC program from Version 4.5 can function.

When configuring HLA hardware, you may only change the telegrams as of slot (object) 1 from 136 to 166.

S120 drives on Profinet isochronous:

Configuring via the NCU is supported, it is not supported via the IO device.

With an isochronous alarm of the NCU, select the setting for the Ti/To mode as “fixed” rather than “in IO device”.

Upgrading the system software version 4.8 SP2 using the existing PLC user program causes a problem to arise with the Toolbox >= V04.07.21 at the call interface of the FB1 in the OB100.

Inserting the new parameter “MCP\_IF\_TCS” results in a change to the call interface of the FB1. This causes the user to receive an error message from STEP 7 after inserting the blocks from the Toolbox >= V04.07.21.

This error message only occurs if the OB100 is opened.

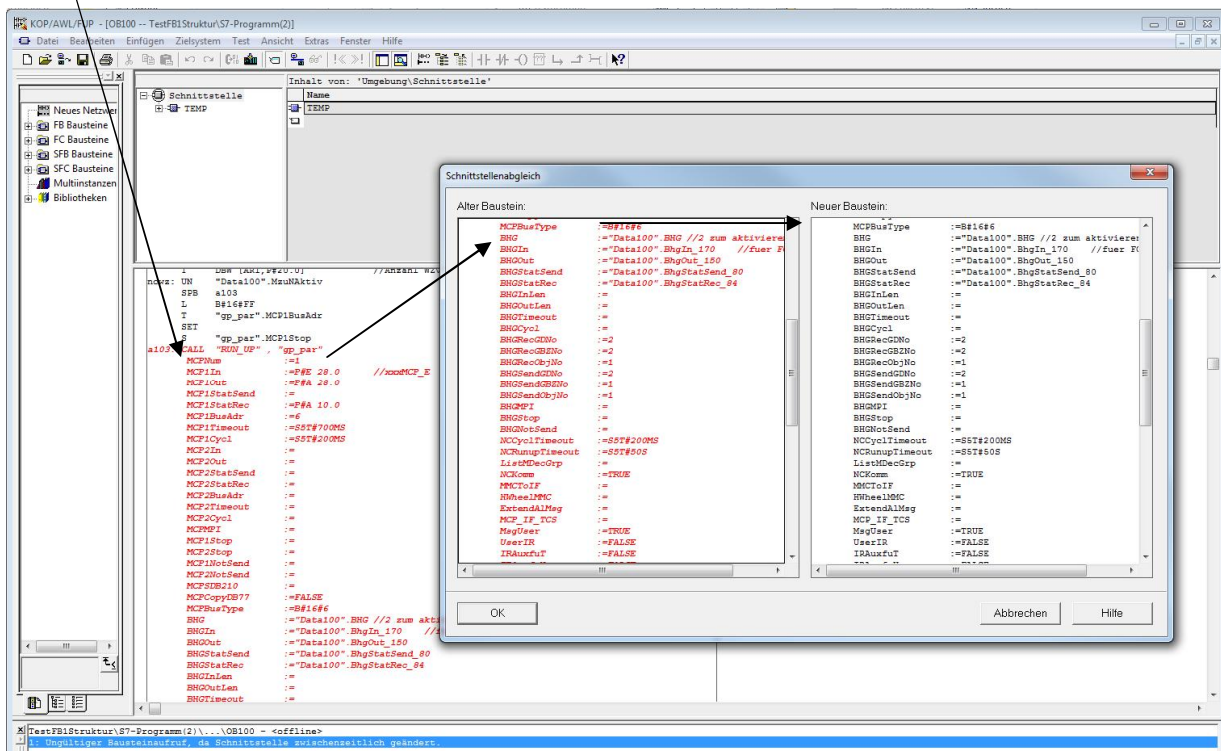


The block call FB1 “RUN\_UP” has to be updated once and then the call parameters are displayed without any time stamp conflict.

## SINUMERIK 840D sl

### CNC Software 4.8 SP3 HF1

Mark block,  
Menu Edit > Call > Update



## 5.5 Notes regarding the use of the Starter with SINUMERIK

Starter version V4.5 SP1 HF5 must be used with SSP SINUMERIK (int. vers. 05.10.10.00 / Release V05.10.18.00\_02.50.09.00).

Note: Standard drive commissioning and also the commissioning of SINUMERIK Safety Integrated plus must be performed via SINUMERIK Operate or SINUMERIK Commissioning, and are not supported by Starter.

Offline commissioning using Starter is likewise not supported.

## 5.6 Notes regarding SINUMERIK Operate

- Access using the VNC Viewer for Operate or Linux via the company network X130 is password-protected from SW 4.8 SP2.

A password with max. 8 characters can be set using service commands at the Linux level:

```
sc vncpwd set companynetwork <my_pass>
```

```
sc vncpwd reset companynetwork
```

Password protection can also be disabled in file user\system\etc\sinumerikvnc.ini using entry

**SINUMERIK 840D sl**CNC Software 4.8 SP3 HF1

---

Authrequired=0 (default setting =1).

- VNC operation in the plant/system network X120 can also be protected with a password that deviates from the "password" standard value. However, this is only possible with panels with TCU30.3; however, not for mixed operation with TCU20.2 or HT8. The password can be entered at the TCU30.3 using the TCU menu, and is saved on the TCU. The old password must be entered in order to be able to change the password. This is important if the TCU30.3 is to be used in a different plant/system with different password.  
At the present time it is not possible to reset the password. You must return the TCU if you have forgotten the password.
- Interface X127 responds just like the X120 interface, and in the basic setting, does not request that a password is entered.
- Simulation, display of "half cut":  
During the simulation, the half cut view allows to examine internal turning operations more exactly. This view has not been designed to view milling operations. In this context, the display of milling operations can lead to long simulation times.
- When using SIMATIC ITC panels, no touch softkey is available for the help function.
- The mold and die view cannot be used with POLY and G91 blocks.
- The mold and die view cannot be used with BSPLINE blocks.
- Only the elements 0 to max. 65534 can be displayed respectively changed in GUD arrays, even if the array is greater in the NC.
- When executing part programs from network drives or editing files on network drives, the user must provide for a stable, interference-free network connection to the network drives.
- OPC-UA:  
There are two possibilities to set up a connection:
  - Connection without security
  - Connection with the security policy "Basic128Rsa15" respectively "Basic256" and the security mode "SignAndEncrypt"Siemens recommends that you always establish a connection with security, as only in this way the confidentiality of the data transferred can be ensured.
- HMIFunctionMode:  
As from Software Version 4.8 SP2 HF3, it is possible to configure an HMI-Base installation on PCU 50 / IPC / PC such that no conflicts occur with a SINUMERIK Operate device being operated in parallel on the same NC/PLC. The configuration data "HMIFunctionMode" must be used for this. This is included in the systemconfiguration.ini file in the section [miscellaneous] and may assume the following three values:  
PanelMode or <empty>  
InterfaceMode  
TerminalMode



**SINUMERIK 840D sl**CNC Software 4.8 SP3 HF1

---

The values "PanelMode" respectively "<empty>" have the consequence that the HMI-Base installation has the same behavior as a regular operator station. In this case, conflicts may occur with a SINUMERIK Operate device being operated in parallel on the same NC/PLC.

If this data is assigned the value "InterfaceMode", the function of the HMI-Base installation is primarily reduced to the function of a passive interface to the NC/PLC. This setting prevents conflicts arising on a SINUMERIK Operate being operated in parallel on the same NC/PLC. The HMI-Base installation does no longer log onto the PLC as a regular operator station, does no longer monitor the PLC interface or supply data to the PLC interface. This HMI-Base installation

- does not make a writing access to the PLC interface,
- does not monitor PLC hard keys,
- does not monitor MMC commands,
- does not monitor language switchover via PLC,
- does not monitor PLC commands,
- does not monitor CTRL-Energy,
- does not monitor screen darkening via PLC,
- and does not set HMI-Ready in PLC.

The value "TerminalMode" is set when using the SINUMERIK Operate being operated in the NCU as tool loading station. In this case, it also shows primarily a passive behavior towards the PLC, such that no conflicts occur on the SINUMERIK Operate being operated on the main operator panel.

Example (systemconfiguration.ini file in the directoryaddon, oem or user):

[miscellaneous]

HMIFunctionMode=InterfaceMode

- Archiving/series commissioning:

Whether files with logon data should be backed up in an archive can be set via the new display machine data 9115

\$MM\_SAVE\_CREDENTIALS.

\$MM\_SAVE\_CREDENTIALS

0 = Dialog is displayed (default)

1 = No dialog, logon data is not backed up

2 = No dialog, logon data is always backed up

A message box is displayed for MD9115 = 0 and existing login data:

"Wollen Sie die Anmeldedaten für Netzlaufwerke und OPC UA in das Archiv aufnehmen?"

"Do you want to include the logon data for network drives and OPC UA in the archive?"

The prompt can be acknowledged with Yes or No.

If "No" or MD9115=1 is selected, the following files won't be backed up:

Logdrive.ini, Logdrive\_.ini.sav, Logdrive.oldstyle

Extdev.ini

UserDataBase.xml

- From SW 4.8 SP2, a VNC Viewer can be integrated as operating area in SINUMERIK Operate for access to a remote PC. The VNC Area configuration dialog can be found under Startup area / HMI / Operating area menu (VSK 5)  
You can move through all of the operating area softkey bars using softkey "Level>>". In every displayed bar, the first available operating area softkey is automatically selected.

Using the "Change" softkey, you can change the parameter of an empty or a VNC operating area softkey. The following parameters can be configured:

VNC connection

- Type: Only "VNC" (or <leer> for delete) possible.
- Computer name: IP address or DNS name of the host, on which the associated server runs.
- Port: TCP port that the server uses.
- Password (optional): Password for the appropriately protected access to the VNC server.
- Operate Header: Defines whether the Operate header should be visible while the VNC area is selected and is in the foreground.
- Position of the status display: Position of the display of the connection status of the VNC area.

Operating area softkey

- Access level: Access level from which the softkey is displayed.
- Softkey text: Either language-independent text for labeling the softkey – or text ID for a language-dependent foreign language text. The text ID can only be used when a text file is specified, along with the text context.
- Text file: Name of the text file containing the context and the foreign language text.
- Text context: Context of the foreign language text.
- Softkey icon: Name of an image file, which is used as icon for the softkey.

Note: As standard, function keys F1 to F12 are not transferred to the remote PC.

## 5.7 Note regarding the Siemens cycle packages

CNC SW 4.8 SP3 contains the following cycle packages on the CF card:

- Technological cycles (standard cycles)
- Measuring cycles
- ISO cycles
- ShopMill cycles
- ShopTurn cycles
- Grinding cycles
- AST cycles
- Adapting Cycles

All these Siemens cycles are automatically loaded into the NC during the run-up of the NCU. The corresponding variables are stored in the definition file PGUD.

Note:

Cycle calls in part programs for these cycles cannot be recompiled or processed with the cycle support in HMI Advanced 7.x.