SIEMENS SINUMERIK ONE CNC Software User Guide

August 15, 2025

Contents [hide]

1 SINUMERIK ONE CNC Software
2 Specifications:
3 Product Information:
3.1 Software Components:
3.2 New Functions and Options:
4 Product Usage Instructions:
4.1 Precondition:
4.2 Installation:
5 FAQ:
5.1 Q: How do I ensure data protection compliance when using the product?
5.2 Q: What are the new functions introduced in SW 6.13 HF5?
5.3 Documents / Resources
5.3.1 References

SINUMERIK ONE CNC Software

“

Specifications:

- Product: SINUMERIK ONE CNC software V6.15 SP4
- Release Date: 2024-03-18

Product Information:

The SINUMERIK ONE CNC software V6.15 SP4 is a comprehensive software package that includes various components for CNC machining and control. It does not process or store personal data, only technical function data.

Software Components:

The software components included in version 6.15 SP4 are:

- PLC-1500F: 02.08.09.09
- PLC-Support: 02.00.00
- NCK-Adapter: 01.05.006

New Functions and Options:

The software introduces new functions and options, including SW 6.13 HF5 and SW 6.14.

Product Usage Instructions:

Precondition:

- **Software:** Ensure compatibility with the required software tools such as Toolbox, SINUMERIK Integrate, and Create MyHMI/WinCC.
- **Hardware:** Check hardware compatibility for smooth operation.
- **Storage Media:** Prepare the required storage media for installation.

- **Software Tools:** Make sure to have Toolbox, SINUMERIK Integrate, and Create MyHMI/WinCC tools available.
- **Other Requirements:** Any additional requirements specified by the manufacturer.

Installation:

Follow these general steps to install the CNC software:

1. Prepare the installation media with the latest version of the software.
2. Run the installation wizard and follow the on-screen instructions.
3. Configure any settings as required during the installation process.

FAQ:

Q: How do I ensure data protection compliance when using the product?

A: The product does not process or store personal data. If personal data is linked or stored alongside technical function data, ensure compliance with data protection regulations.

Q: What are the new functions introduced in SW 6.13 HF5?

A: SW 6.13 HF5 has not been released yet, so specific details on new functions are not available at this time.

“`

View Fullscreen

SINUMERIK ONE

CNC software V6.15 SP4

2024-03-18

Contents of the Installation and Upgrade Instructions

1 SOFTWARE COMPONENTS ……
…… 3 2 NEW FUNCTIONS and OPTIONS ……
…….. 5

2.1
Functions……
…….. 5 2.2
Options……
…….. 8 3
PRECONDITION……
…….. 10 3.1
Software……
……. 10 3.2 Hardware
……
…….. 10 3.3 Storage media
……
……. 10 3.4 Software Tools
……
…… 10 3.5 Other requirements
……
……. 10 4 INSTALLATION
……
…… 11 4.1 General information on installing the CNC
software …….. 11 5
NOTES AND
RESTRICTIONS……
…….. 12 5.1 General
information……
……. 12 5.2 Security changes
……
…… 12 5.3 References to other documents
……
…… 12 5.4 Hints CNC software
……
…….. 12 5.5 Notes on SINUMERIK ONE Operate
……
…. 21 5.6 Notes on SINUMERIK STEP 7 Toolbox
……

24 5.7 Information on using Starter with SINUMERIK ONE
…… 25 5.8
Information on using Startdrive with SINUMERIK ONE
…… 25 5.9 Note on the
Siemens cycle packages
…….
. 25

SINUMERIK ONE
CNC software V6.15 SP4

2024-03-18

This document is used as instructions for the software update to SINUMERIK ONE CNC-Software 6.15 SP4. Please read the documents carefully because they contain important information for installation and use of the software.
Siemens observes standard data protection principles, in particular the principle of privacy by design. For this product, this means:
The product does not process or store any personal data, only technical function data (e.g., time stamps). If the user links this data with other data (e.g., shift schedules) or if he/she stores personal data on the same medium (e.g., hard disk), thus personalizing it, he/she has to ensure compliance with the applicable data protection stipulations.

SINUMERIK ONE

CNC software V6.15 SP4

1 SOFTWARE COMPONENTS

The CNC software 6.15 SP4 (internal version 06.15.04.00.022) comprises the following components:

Component	Version
PLC-1500F	02.08.09.09
PLC-Support	02.00.00
NCK-Adapter	01.05.006
PLC-Loader	02.00.02
SMTED	04.95.04.00.004
Sinumerik Machining Technology Extension	04.95.04.00.004
Cycles	

04.95.04.00.006

NCK (inkl NRK)

138.11.00

SNCK

04.08.00.00.001

HMIARC-Tool

04.09.00.01.001

CP1543-1

02.00.09.01

Hypervisor

01.28.00.02

Diagnosis Data Collector

02.09.02.00

COS

02.00.23.01

CM1542-5

02.00.10.02

Sinumerik ComServer

01.10.00.01

DiagSrv Application

01.22.00

SINUMERIK_DRM

01.00.00.00

NCK Filesystem Driver

04.07.01.09.001

SINAMICS

05.20.73.51

2024-03-18

LinuxBase SINUMERIK ONE Operate SinIntClient OPC UA Server

SINUMERIK ONE
CNC software V6.15 SP4
06.01.33.00 04.95.04.00.000118 04.00.27.00.002 03.01.05.00.000023

2024-03-18

2 NEW FUNCTIONS AND OPTIONS
2.1 Functions
SW 6.15 SP4 · global SafeUserData signature is added in the diagnostics area.
SW 6.15 SP3 · Archiving NC-data in Punch tape format. · Start-Stop PLC via HMI-OA Interface.
SW 6.15 SP2 HF3 · keine
SW 6.15 SP2 HF2 · keine
SW 6.15 SP2 · keine
SW 6.15 SP1 HF5 · None
SW 6.15 SP1 HF3 · None
SW 6.15 SP1 HF1 · None
SW 6.15 SP1 · Extension of nutrast compensation · Extension ILC/IDC · DYNxxx implizit mit CALCFIR · ORIAXESplus · Potenzfunction
SW 6.15 HF1 · None
New functions in SW 6.15 · Run MyRobot / Direkt Control V2.0. · Protect MyMachine Multichannel (collision avoidance new). · Y-Parting. · Support of the energy-saving profiles (CtrlEnergy) by the PLC basic program.

SINUMERIK ONE
CNC software V6.15 SP4

2024-03-18

· Support of the function “star-delta switching” by the PLC basic program (LBP_CtrlStarDelta[FC17]). · Additional telegram 1100 “PZD extension 0/32” on the CU. Can be used together with SINAMICS function module p0171. Bit 00 “PZD extension” on the CU-DO and SINAMICS Technology Extension TRCBUF (Trace Buffer), MLFB

6SL3077-0AA00-6AB0 (CoL). · NCU Servicesystem, menu for clear SD-Card. · HMI support for manual IBN of SINAMICS Technology Extensions. · Motor Encoder simulated.
SW 6.14 HF3 · None
SW 6.14 · Top Speed Plus · Free adaptation of controller parameters, use telegram 146 together with SIC/SCC · Extension of PROFIdrive manufacturer telegrams with signal “close brake” in the control word STWx.y or SCC · DQ telegram extension of the encoder telegram for additional signals from vibration sensors · Safety Integrated: Acceptance test support in SINUMERIK Operate · Safety Integrated: dbSI Commissioning support (commissioning masks in SINUMERIK ONE Operate) · Single Channel Safety Encoder for SINUMERIK ONE (dbSI) · DSF archives: type and version verification · DSF Archives: Archive Type Customized Archive · Time synchronization between PLC and NC
SW 6.13 HF6 · None
SW 6.13 HF5
Has not been released
SW 6.13 HF4 · Machine registration will be mandatory, disabling the alarm by machine data is not available anymore.
SW 6.13 HF1 · None
SW 6.13 · Reduction of minimal possible PLC Scan time · Operate has QT 5.12 · Easier operation for ATW in conjunction with Touch panels. · Improved Wizard for optimizing spindle with AST (Auto Servo Tuning). · New NC code “SUPD”: the tool offsets can be suppressed each code line · Introduction of angle head adapter · Tool measuring, possibility for extended error reaction (SD54750 $SNS_MEA_ALARM_MASK Bit 16)

· Easy OSS registration via QR-Code on Operate possible · Improvement of the extended interface between SINUMERIK and CAS (Collision Avoidance System) of ModuleWorks · New dynamic group “DYNPREC” · Introduction of new Setpoint filters (new modes in MD AX_JERK_MODE) · Tool requirements determination for JobShop program available
SW 6.12 HF2 · OPC-UA V3.0SP1 · Software runs on PPU1740
SW 6.12 HF1 · No new functions
SW 6.12 · CU320-2PN can be used as NC axes · New OPC-UA Version V3.0 included · Additionally, see readme of SW 4.92

2.2 Options
SW 6.15 SP4 · none
SW 6.15 SP3 · P86 PCU-Basesoftware /IPC for SW-version >= V15, 6FC5800-0BP86-1YB0
SW 6.15 SP2 HF3 · none
SW 6.15 SP2 HF2 · none
SW 6.15 SP1 HF5 · none
SW 6.15 SP1 HF3 · none
SW 6.15 SP1 HF1 · none
SW 6.15 SP1 · none
SW 6.15 HF1 · none
SW 6.15 · P85 SINUMERIK ONE AccessMyMachine /OPC UA Advanced 6FC5800-0BP85-0YB0 · P83 SINUMERIK ONE AccessMyMachine/OPC UA OEM 6FC5800-0BP83-0YB0 · S58 Turning Y-Parting 6FC5800-0BS58-0YB0 · E00 CA ECO multichannel for ONE 6FC5800-0BE00-0YB0 · G00 CA Standard multichannel for ONE 6FC5800-0BG00-0YB0 · F00 CA Advanced multichannel – interfacesupport ONE

6FC5800-0BF00-0YB0
SW 6.14 HF3 · none
SW 6.14
· S62 Top Speed Plus 6FC5800-0AS62-0YB0
· P57 Interpolation turning 6FC5800-0BP57-0YB0
· S41 One Dynamics Operate 6FC5800-0BS41-0YB0 · S42 One Dynamics 3 Axis Milling 6FC5800-0BS42-0YB0 · S43 One Dynamics 5 Axis Milling 6FC5800-0BS43-0YB0 · P26 Path acceleration limitation 6FC5800-0BP26-0YB0 SW 6.13 HF6 · none SW 6.13 HF4 · none SW 6.13 HF3 · none
SW 6.13 HF2 · none
SW 6.13 HF1 · none

SINUMERIK ONE
CNC software V6.15 SP4
SW 6.13 · Top Speed 6FC5800-0BS13-0YB0 · 3D Feature Scanner 6FC5800-0BP70-0YB0 · Werkzeugbedarf ermitteln 6FC5800-0BM77-0YB0 · Winkelkopf-Adapter 6FC5800-0BM56-0YB0 · Konfigurierter Halt 6FC5800-0BS24-0YB0
SW 6.12 HF1
· No new options
SW 6.12
· Jerk adjustment 6FC5800-0AS22-0YB0
· Intelligent Load Control 6FC5800-0AS11-0YB0

SINUMERIK ONE
CNC software V6.15 SP4

2024-03-18

3 PRECONDITION

3.1 Software
· Sinumerik Service System >= V06.40.00.00 (int. V064.00.00.0029) · OnboardSW (InitNCU) Image >= V06.00.38.00. · When SINUMERIK ONE Operate is used on IPC/PC, Version 6.15 SP4 must be used.

3.2 Hardware

· The SINUMERIK ONE must only be operated on SINUMERIK NCUs 1740, 1750, 1760, PPU1740

3.3 Storage media

· SD-Card 8GB – für die Standardversion 6FC5852-1XG00-_YA_.

3.4 Software Tools
· TIA-Kombinationen TIAP

Toolbox

SINUMERIK Integrate Create MyHMI /WinCC

V17

V17 Update 12

V17

V18 Update 2

V18 SP1 Update 1

V18 update 2

V19

V19

V19

· SINUMERIK Integrate Create MyHMI /3GL V6.15 SP4 · SINUMERIK Commissioning V6.15 SP4 · SINUMERIK Service System >= V06.40.00.00
· Access MyMachine /P2P V4.9 SP2 HF1 (04.09.02.01) · Lock MyCycles (Option 6FC5800-0BP54-0YB0), Protector v2.5.4
· Wkonvert Wizard V2.6 (V02.06.00.00.004) · Create MyConfig from 6.4

3.5 Other requirements
· The use of SINUMERIK ONE Operate internally on a NCU requires the Option: o S00 SINUMERIK Operate /NCU (6FC5800-0BS00-0YB0) or o S87 SINUMERIK Operate /universal client (6FC5800-0BS87-0YB0)
· The use of SINUMERIK ONE Operate externally on an SIMATIC IPC (PCU) requires the Option: o P88 SINUMERIK ONE Operate /PCU (6FC5800-0BP88-0YB0) nd P86 PCU-Basesoftware (6FC5800-0BP86-0YB0)
· The use of SINUMERIK ONE Operate extern on PC/PG requires the Option: o P87 SINUMERIK ONE Operate /PC (6FC5800-0BP87-0YB0) or o N00 SINUMERIK ONE Operate /PCU ALM (6FC5860-2YF00-1YB0)

2024-03-18

4 INSTALLATION

4.1 General information on installing the CNC software

· The Sinumerik Service System on a USB stick is required to install the CNC software. New installations can be performed via the Sinumerik Service System system and TCU.

· The description of the procedure can be found in the installation manual (new installation and upgrade) published in the SIEMENS Industry Online Support

· The InitNCU Image Version 06.00.38.00 must be installed on NCU17x0 / PPU1740

5 NOTES AND RESTRICTIONS

5.1 General information

The language extensions of V6.15 cannot be used. Language extensions from V6.15 SP1 can be used, new texts from V6.15 SP4 may be added in English.

5.2 Security changes

SW 6.15 SP4 · OpenSSL o Update auf Version V3.1.4.

SW6.15 SP3 · OpenSSL o Update to V3.0.7.

5.3 References to other documents

2024-03-18

Inhalt/ Name

Functional improvements Boundary conditions Cycles SINUMERIK Service System (früher ,,eboot”) TIA-Toolbox Baseprogram

Datei

Resolved function limitation In this document, see chapter 5 appendix_cycles
SINUMERIK_Service_System
Path…DocumentsReadme LiesMich_PLCBP./ Readme_PLCBP LiesMich_PLCBPPlus / Readme_PLCBPPlus

Kommentar
New from V18 is spart of the Toolbox file structure

5.4 Hints CNC software
· [V6.15 SP4] 1:n, use server mode only. · [TIAP TB] Important information about the basic program can be found in the “Readme”, available from TB V18
(see table “References to other documents”). · The following boundary conditions of Advanced-/TOP- Surface and TopSpeed/PLUS must be observed:
Machinedata setting: https://support.industry.siemens.com/cs/ww/de/view/109738423
Top Surface must only be configured in the first and another channel and only be active in one channel. Operation with active Protect MyMachine / 3D Primitives (CA ECO) or Protect MyMachine Open is possible. Operation with active Protect MyMachine /3D STL oly possible with restrictions.

Operation only with 3D WRK differential tools as CUT3DFD or CUT3DCD. Top Surface uses the cycle832 tolerance value for the contour as opposed to Advanced Surface, where the axis tolerance value is used. This results in a difference of SQRT(3) of the effective contour tolerance and, if necessary, surface and speed or processing time differences when comparing Advanced Surface and Top Surface. Top Surface performs

an improved jerk view compared to Advanced Surface. If the data throughput of data-intensive part programs leads to a decrease in the IPO buffer level, the FIFOCTRL function intervenes and reduces the path speed to avoid a stop. · Safe removal of external storages (6.15 SP1): Safe remove of external storages (Hardware) like USB-Stick etc., for this you must select the menu Prog. Manager or System data. Safe unplugging is shown by a triangle symbol

· Reactivate deselected transformation after record search With spindle programming during sentence search, in particular when using MD11450 SEARCH_RUN_MODE – Bit 2, it is necessary to select the transformation at different times (at the beginning of the PROG_EVENTs or customerspecific in the PROG_EVENT), otherwise there will be conflicts between the transformation and spindle axis. The active transformation is not suppressed with MD11450 SEARCH_RUN_MODE – Bit 2 = 0 with active record search, i.e. the transformation remains active via the record search The active transformation is suppressed with MD11450 SEARCH_RUN_MODE – Bit 2 = 1 with active set search, i.e. the transformation must be activated again via SEATRAON. All spindle programming must be completed before the transformation can be activated using SEATRAON.
There are the following options for this (only relevant for MD11450 SEARCH_RUN_MODE, bit 2 = 1): 1. Automatically by the default setting MD52212 $MCS_FUNCTION_MASK_TECH,Bit 19 = 0 If the default setting is active, the suppressed transformation is automatically reactivated with SEATRON at the beginning of the PROG_EVENT program. This can lead to conflicts (see also Alarm 17640 – “Spindle operation for transformed axis not possible”) between the transformation and spindle axis, otherwise the case 2 is to be used. 2. User-specific with the setting $MCS_FUNCTION_MASK_TECH, bit 19 = 1 Before the transformation call, it must be ensured that the transformation can be called, e.g. spindle is in axis operation. o by programming SEATRAON With the predefined PROCEDURE SEATRAON, the user can reactivate the suppressed transformation after a sentence search. The following transformations containing a spindle axis are taken into account: Classic transformations: TRANSMIT, TRACYL and TRAORI, as well as OEM transformations. Transformations based on the kinematic chain: TRANSMIT_K, TRACYL_K, TRAORI_STAT, TRAORI_DYN, TRAINT.
Calling SEATRON is only allowed in the PROG EVENT program, which is automatically activated when the last set of actions is changed ($P_PROG_EVENT==5).

o User-specific by explicitly calling the suppressed transformation with e.g. TRAFOON in the sentence search ASUP.
The new record search variables are available for selecting the transformation. There are the following new set search variables for conventionally defined transformations: $P_SEARCH_TRAFO $P_SEARCH_TRAFO_NUM $P_SEARCH_TRAFO_PAR

Variables for transformations based on kinematic chains:
$P_SEARCH_TRAFO_NAME $P_SEARCH_TRAFO_PARSET
These variables contain the transformation data to be activated in the search, analogous to the system variables $P_TRAFO, $P_TRAFO_NUM, $P_TRAFO_PAR and $PC_TRAFO_NAME and $P_TRAFO_PARSET in the normal program flow. The variables only provide the desired data in the ASUP in the state ($P_PROG_EVENT==5) until the call of SEATRAON or the on (e.g. TRAFOON) or switching off (TRAFOOF) of the transformation. If no transformation is reactivated in the record search ASUP (the suppressed one), the following alarm 14450 – “Transformation deleted in scan” occurs. · If a trace file has been saved with a GIV Version <6.15 and contains spindle variables that were used in system variable format (e.B. $AC_SGEAR) and not in BTSS format, these variables may be invalid. Workaround: Remove and reinsert trace configuration of the affected spindle variabel. · SafeUserData – CNC Software Stand 6.15 The basis for using the “SafeUserData” feature is the STEP 7 library “SINUMERIK ONE SafeUserData V01.10.00.00”. This STEP 7 library is expected to be available soon. · Copy to USB is aborted with error message when copying large files (>150MB) from front USB (USB1.1) to a USB stick, error messages may occur (“USB device is no longer available!”). As a workaround you can increase the TimeOut in

usersinumerikhmicfgsystemconfiguration.ini (input in milliseconds), e.g.: [miscellaneous] usbDriveTimeout=900000 · USB sticks on interfaces with USB standard 1.1, affected are the front interface of OP010, OP012 and OP015.It is recommended to use only USB sticks up to a maximum of USB 3.0 standard on the USB 1.1 interfaces, as interrupted copy processes can occur when using newer USB memories. We recommend that you check the copied files for completeness · USB sticks, general recommendation for use at SINUMERIK-Operate: After data has been copied to the stick or changes have been made to files on the stick, a short waiting time (about 20 seconds) should be observed before the stick is removed. · Sinumerik Integrate 4, when using CNC-SW 6.15 in conjunction with SINUMERIK Integrate 4.1 Sp10 (Manage MyTools, Manage MyPrograms, Analyze MyPerformance or Analyze MyCondition), the HF of the 4.1 SP10 HF4 must be installed. The description of the installation procedures can be found in the SINUMERIK Integrate 4.1 SP10 HF4 documentation. Sinumerik Integrate versions <4.1 SP10 HF4 cannot be used with Operate 4.95.

· SINUMERIK Integrate 5, when using version CNC-SW 6.15 in conjunction with (Manage MyResources /Tools, Manage MyResources /Programs, Optimize MyProgramming /CAM integration, Analyze MyPerformance), version SINUMERIK Integrate 5.1.2.0 must be installed. The description of the installation procedures can be found in the SINUMERIK Integrate 5.1.2.0 documentation. Sinumerik Integrate versions <5.1.2.0 cannot be used with Operate 4.95.
· Constraint of downgrading from 6.15 -> 6.14 For security reasons, the encryption of the network credentials had to be changed. This can lead to problems with authentication. To prevent this, please proceed as follows: Create an HMI and System Settings archive

before upgrading to 6.15 Upgrade to V6.15 read in the created archive …. Upgrade to a version <V6.15 (if desired) Read the archive that was created before the upgrade.
· New Collision Avoidence Algorithus (from 6.15) If the options are enabled: o Protect MyMachine /3D STL – 6FC5800-0AS02-0Yx0 / Protect MyMachine /3D STL add. 1 Channel – 6FC58000BG00-0Yx0 o Protect MyMachine /3D Primitives – 6FC5800-0BS03-0Yx0 / Protect MyMachine /3D Primitives, add. 1 Channel – 6FC5800-0BE00-0Yx0 a new CA algorithm is automatically activated. The IPO load will be higher due to the new CA algorithm. Collision avoidance is a performance intensive function. Depending on the hardware used (NCU/PPU1740, NCU1750, NCU1760), the set interpolation clock, the size of the collision model and other simultaneously used, performance intensive NC functions such as TopSurface, TopSpeed, synchronous actions, tracing), it may happen that the collision checks cannot be completed in time of 1 IPO cycle. In this case, the Collision protection is limited and alarm 26303 “Collision check incomplete” is displayed. This alarm can be suppressed by the machine data (MD11415 $MN_SUPRESS_ALARM_MASK_2 bit 29) Technology G-code groups 59 must be set correctly (DYNNORM, DYNPOS, DYNROUGH, DYNSEMFIN, DYNFINISH, DYNPREC). Collision avoidance uses the emergency brake ramp in exceptional cases (alarm 26285).The following machine data must be checked and set sensibly: MD36610 $MA_AX_EMERGENCY_STOP_TIME MD36620 $MA_SERVO_DISABLE_DELAY_TIME Depending on the machine and control configuration (low axis dynamics, small IPO cycle), the following machine data may have to be increased: MD18951 $MN_MM_COLLISION_PREVIEW_LEN MD18952 $MN_MM_COLLISION_BRAKEPATH_LEN
· Change behavior of block search with program test (from 6.15) The block search with program test has been changed so that the new variant “without approach” is automatically used. Via the MD 51024 $MNS_BLOCK_SEARCH_MODE_JS or 51028 $MNS_BLOCK_SEARCH_MODE, the old variant can also be offered by setting bit 6, if required. When searching for a ShopTurn cycle, the new variant “without approach” must be used!
· NCU SD card (from 6.14), the folder /addon has the access level “root” , customer applications are to be placed in the folders /oem, or /user.

SINUMERIK ONE
CNC software V6.15 SP4

2024-03-18

· An SD card blank image is also supplied with the NCU service system on Pridanet and can be used for repairing the SDcard in case of problems. The function is available from V06.00.83.01. For detailed information see readme of NCU service system.
· It can happen that the texts (including softkey texts) are not displayed after installation of the GIV. In this case the GIV has to be installed again.
· Deleting the PLC memory card and/or loading the factory settings (deleting retentive data) via TIAP requires power off/on or a reset on the button of the NCU/PPU after termination.
· Starting with SW 6.14, the Operate Navigation menu is displayed by default. If an HMI-OA application has been configured with a self-defined resolution, your image may appear beyond the margin because the Operation Navigation menu does not provide enough space. The display of the navigation menu can be undone by changing the display machine date “MM_Sidescreen” from “3” to another value.
· From SW 6.14, auxiliary function output and alarm 26052. Adjustments in the internal behavior of the auxiliary functions can in rare cases result in NCK message 26052. Furthermore, the adjustment can be noticeable during the entire program runtime if many auxiliary functions are used. Workaround: Reduction of the web speed to approx. 2/3 of the programmed feed.
· From SW 6.14, PLC Archive from 6.13 are compatible · The MD $MA_SAFE_FUNCTION_ENABLE is on Sinumerik ONE not available.
Imported data from 840D sl could generate alarms 15085 or 12550 · The system bootloader is part of the FW and cannot be upgraded separately · The “initNCU” is renamed and is displayed as “OnboardSW” at the HMI · An empty PLC remains in stop when it is started · Quick commissioning of the drives without PLC hardware configuration is rejected with errors · Meaning of SD Error’s in 7-segment-display

1: system card device missing in HwInfo 2: system card device didn’t register (-> HW problem) 3: card missing, or ext4 partition on it missing 4: fsck failed with uncorrectable errors 5: mounting filesystem failed · PPU1740 with hardware version >= “B” (FS: B) can only run with SW >= 6.13 HF2 in case of mismatch, error “SIGERROR 14” on 7-segment display occurs during the start-up. · NCUs 1750/1760 with hardware version >= “E” (FS: E) can only run with SW >= 6.13 HF2 in case of mismatch, error “SIGERROR 14” on 7-segment display occurs during the start-up. · With SW 6.13, the minimum possible PLC cycle time has been reduced from 2xIPO + 3ms to approx. 1xIPO + 0.5ms. It depends on the size of the PLC user program, which cycle time will be set. It is recommended to document the cycle time before and after the upgrade and to check any time-critical applications in case of lower cycle time. · From SW 6.13, the new cycle protector >= v2.3.0 should be used. · From SW 6.13, introduction of a new licensing procedure, the sent *.zip-file may not be unpacked, it has to be read in via HMI mask. · From SW 6.13, the SD card of 4GB must be exchanged to 8GB (new licensing necessary) · Procedure for PLC upgrade in TIAP from V15.1 to V16 is described in a separate document located on Pilot sharepoint. · Profinet IRT, NCK drives or equidistant I/O peripherie on X150 only · DSF Archive

PLC DSF, a Typ-and Version check is executed during reading in NC, HMI etc., DSF does not depend on NCU type. · From SW 6.13 compressors COMPON and COMPCURV are no longer supported. Programming of COMPON or COMPCURV also over a cycle (e.g., CYCLE832) is rejected with alarm 14089 “Function no longer supported” and the parts program is interrupted. · Archive creation only possible from access level USER (from SW 6.13) As of SW 6.13, it is no longer possible for security

reasons to create an archive with a key switch level. There must be at least the password for the user. · Machine registration (from SW 6.13 HF4) Alarm 150402 won't be displayed, even the registration not yet done. Machine registration (from SW 6.13) It is now easy to register the machine. Register a QR code is displayed under DIAGNOSE-Version-Machine, which is transmitted to a server by a QR code scanner and returns a registration ID. After entering this ID, the machine is registered. As long as it is not registered, alarm 150412 is displayed. Alternatively, it is possible to register manually using the web address specified there. · Change the maximum value of the machine date 28090, $MC_MM_NUM_CC_BLOCK_ELEMENTS · Due to a fix, the maximum value of the machine date 28090, $MC_MM_NUM_CC_BLOCK_ELEMENTS, has been changed from 130 to 64. Because internally only a maximum value of 64 was worked, this change does not affect the number of sentence elements. However, if a value greater than 64 has been entered in an archive for this machine date, it is rejected when loading the archive with the alarm 17090 "Value greater than upper limit" and the default value "0" remains standing. Here, the user must make an adjustment in the machine date. · New password guidelines apply. When an NC password is changed, the password for the respective Linux account is also changed. I.e. when the manufacturer's password is changed, the password for the Linux account "manufact" is also changed. IMPORTANT: After one of the existing NC passwords (e.g. SUNRISE) has been changed, it is no longer possible to change back to the old password, because it does not correspond to the new guidelines (minimum length 8 characters, uppercase letter, lowercase letter, number). If an access level in NCK is to be set by PLC via FB4 (PI-Service_N_LOGIN), a password that is in effect for NCK (and simultaneously for Linux) can only be exactly 8 characters long. This scenario must be taken into consideration when assigning new passwords. Otherwise passwords can be 8 – 32 characters long. · If an upgrade takes place within an active test license period (Trial License), then the current remaining time is shortened to 1/10. The next period then has the normal length again. · Persistent storage of the SRAM data: As of SW 6.11, there has been a switch from synchronous mode to the asynchronous mode for backing up SRAM data. To this end, the default value of MD $MN_MM_MEMORY_CONFIG_MASK has been changed from "0" to "1". The background is that now, in a normal scenario, the processing is no longer stopped when the data is flushed. · At the same time, the alarm 15120 "If a power failure occurs now, the last changed data is lost (index/buffer size=%1)" is no longer suppressed. The alarm is intended to indicate to the manufacturer/user that his

configuration at this point is unfavorable for the control behavior. This alarm can still be hidden via MD $MN_SUPPRESS_ALARM_MASK_2 Bit 3 =1.

· Auto Servo Tuning – AST If an upgrade of CNC SW < V4.5 SP3 takes place, the stored session files of AST (Auto Servo Tuning) are not compatible and the AST settings (strategy, measured parameters, etc.) are lost.
· The default directory for XML files is changed from card/user/sinumerik/hmi/log/optimization (SW 4.5) to card/user/sinumerik/nck/data/optimization (as of SW 4.7SP2). It is possible to perform an XML export. The files are saved under card/user/sinumerik/hmi/log/optimization. If “Clear History” is used, then a backup is saved under card/user/sinumerik/nck/data/optimization/backup DDS0 is always preselected under “Optimization of active filters”, not the DDS currently selected via PLC/NC. The DDS must be explicitly selected via “Filter group”.
· AST filter settings: When there is a switchover to a different axis, for example, the changes are lost. This also applies to the filter groups and DDS settings
· Commissioning wizard: The drive wizard does not function for axes on CU320-PN with a connected TB30 module · DSF archive:
Create MyConfig (e.g.: also CMC Diff) or AMM /P2P as of V4.8 HF1 can be used. Alternatively, the archives can also be opened with a ZIP program /e.g., 7zip or WinZip).
· Restrictions for EES: Currently, any kind of editing of the files with EES mode active on the USB is not possible for a USB stick on the TCU in combination with a PCU, but it is possible to process them. If a USB stick on a TCU is simultaneously accessed by several HMIs/NCUs, no coordination takes place between the components, which

means that there is a risk that a program that is being processed can be modified/destroyed by the other component. · When TRAILON is activated by means of synchronous action with active transformation (TRAORI), an erroneous coupling may occur if a RESET is triggered while coupling is active. In the event of a RESET, the transformation is deactivated for an IPO cycle. As a consequence, a setpoint value is specified in the MCS instead of in the BCS. Thus, the coupling is no longer done to the geometry axis, but to the machine axis, which causes a position jump. TRAILON interpolates this jump and generates a move to an incorrect coupling position (MCS), which leads to large and fast axis movements. Solution: The user must ensure that, when TRAILON is activated, the transformation is active and the tripping/setting of RESET is suppressed in the PLC. · Function G643 (block-internal smoothing) has been released for applications in the tool change area (e.g. optimized approach to the tool change position). The function has not been released for applications in the machining process. · Function G644 (smoothing at max. possible dynamics) has been released for applications in the tool change area (e.g. optimized approach to the tool change position). The function has not been released for applications in the machining process. · If different computational accuracies are set for the rotary and linear axes via $MN_INT_INCR_PER_MM and $MN_INT_INCR_PER_DEG, this must also be taken into consideration 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. · SSH port 22 on X130 interface: The SSH port 22 on the X130 interface in the firewall is disabled for safety reasons. The consequence of this is that the controller is no longer reachable from the outside via this interface. Among other things, this affects the Operate on PCU/IPC/PC (uses this for some functions), WinSCP and AMM. This port can be permanently enabled in Operate (only possible on NCU) under Commissioning -> Network ->

Company network -> Change -> Checkbox “SSH (TCP/22)”, see screenshot:

2024-03-18

Alternatively, this can also be done via a manual entry in the file /card/user/system/etc/basesys.ini (just as ports 102 and 5900 may currently already be enabled there). · The content of /siemens/sinumerik/mcp_client/mcp_client.ini has been changed to support the new “WarningTimeout” parameter. Therefore, all users who have their own mcp_client.ini in /card/user/sinumerik/mcp_client must import the new file from /siemens/sinumerik/mcp_client and make their changes there. Otherwise, an MCP on X120 would no longer function. · In connection with jerk filter type MD32402=2, the programmed thread start position SF was not adhered to for G33 with SPCOF (position coupling of the actual spindle value with the axis setpoint value). · The thread start position is now adhered to regardless of the programmed speed. · The passwords for the NCK and the corresponding Linux accounts of the NCU basic system are now changed with the “Change password” Operate dialog. · Tool management The number of adjacent locations was increased from 7 to 11 “Monifact” function: Monifact is separated from the TOA context and is treated completely in a channel-specific way. The respective active Monifact value can be read into the channel-specific status data (block C/S) “aMonifact”. Now there is no conversion of the service life values of the BTSS (block T/TS). This means that the tool service lives in Operate are no longer evaluated using MONIFACT – i.e. they are displayed in real-time. If the real remaining service lives of tools are to be specified while taking $A_MONIFACT into consideration, then the application must link the remaining service life ($TC_MOP2) with $A_MONIFACT The service life monitoring is evaluated with no changes using the Monifact value (i.e. part programs are or remain fully compatible). · The function “flat D numbers” (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. · Drive parameters r2780 to r2789: As of SINAMICS V 5.1, in addition to the function module parameter p/r0108, three new function module parameters p/r0171 – p/r0173 have been introduced, via which the various new (optional) SINAMICS functions can be switched on/activated. The parameters belonging to these new (optional) SINAMICS functions are only provided with values if the associated functionality is switched on/activated via one of the three new function module parameters p/r0171 – p/r0173. This relation is not yet supported by SINUMERIK

Operate and

leads to no values but only “#” being displayed for the SINAMICS parameters if the functionality has not been switched on/activated via the associated function module parameters p/r0171 – p/r0173.

5.5 Notes on SINUMERIK ONE Operate

Note: In addition, when using Operate on IPC/PC, the corresponding upgrade instructions must be observed.

· The separate boundary conditions of the PLC Basic program plus regarding Operate must be observed.

· If there are interruptions in the camera image when displaying a camera image in the

Display Manager, try to reduce

the resolution of the camera image and the refresh rate. This can happen if program on the NCU. If it is not possible to

further reduce the parameters for the camera image, you can also display the camera image in the Side Screen widget

· Access to Operate or Linux by means of VNC Viewer via the company network X130 is provided with

password

protection.

A password with a maximum of 8 characters can be set with service commands on the Linux level:

sc vncpwd set companynetwork <my_pass>

sc vncpwd reset companynetwork

The password protection can also be deactivated in the file usersystemetcsinumerikvnc.ini with the entry

Authrequired=0 (default =1)

· The VNC operation in the plant network X120 can also be protected with a password that deviates from the default “password”. This is only possible with Panels with TCU30.3, however, not for mixed operation with TCU20.2 or HT8. The password can be entered on the TCU30.3 via the TCU menu and is saved on the TCU. To change the password, the old password must be entered. This is important if the TCU30.3 is to be used on a different system with a different password. It is currently not possible to reset the password. If you have forgotten the password, the TCU must be sent in.
· The interface X127 behaves like the X120 interface and does not require a password in the basic setting.

· Simulation, display “half-section”: In the simulation, the half-section view is used for more precise viewing of inside turning operations. This view was not designed for viewing milling operations. Displaying milling operations in this way can result in long simulation times.
· If the alarms 4075 or 4076 occur when the simulation is started, this is due to NC machine data which has the Siemens protection level for writing. These can have values that deviate from the default values in the customer archives. These NC-MDs must be deleted from the archive, e.g. with CMC / UPDiff.

· The mold making view cannot be used for blocks with POLY and G91. · The mold making view cannot be used with BSPLINE blocks. · For GUD arrays, only the elements 0 to max. 65534 can be displayed or changed, even if the array in the NC is larger.

· When processing part programs from network drives or when editing files on network drives, a stable, interruptionfree network connection to the network drives must be ensured on the user’s end.

· OPC-UA:

There are two ways to establish the connection:

.

Connection without security

.

Connection with the security policy “Basic128Rsa15” respectively “Basic256” and the security mode

“SignAndEncrypt” Siemens always recommends setting up a connection with security, as only in this way can the confidentiality of the data transmitted be ensured.
· HMIFunctionMode: It is possible to configure an HMI-Base installation on PCU 50 / IPC / PC in such a way that no conflicts occur with a SINUMERIK Operate that is operated on the same NC/PLC. The configuration data “HMIFunctionMode” is used for this. This can be found in the systemconfiguration.ini file in the [miscellaneous] section. The following three values can be adopted:
PanelMode or <empty> InterfaceMode TerminalMode
The values “PanelMode” or “<empty>” lead to the HMI base installation behaving like a regular operator panel. In this case, conflicts can occur with a SINUMERIK Operate that is being operated simultaneously on the same NC/PLC. If this data receives the value “InterfaceMode”, the function of the HMI base installation is essentially reduced to the function of a passive interface to the NC/PLC. With this setting, conflicts with a SINUMERIK Operate that is being operated simultaneously on the same NC/PLC can be avoided. The HMI base installation no longer logs in as a regular operator panel for the PLC, no longer monitors the PLC interface, and also no longer supplies the PLC interface. This HMI base installation · does not write to the PLC interface · does not monitor PLC hard keys · does not monitor MMC commands · does not monitor language selection via PLC · does not monitor PLC commands · does not monitor Ctrl-Energy · does not monitor dark-ON via PLC · does not set HMI-Ready in the PLC
The value “TerminalMode” is used if the SINUMERIK Operate that is operated in the NCU is used as a tool loading station. In this case, it also behaves toward the PLC in an essentially passive way, so that no conflicts with the SINUMERIK Operate that is used on the main control panel occur.
Example (systemconfiguration.ini file in the addon, oem or user directory):
[miscellaneous] HMIFunctionMode=InterfaceMode

SINUMERIK ONE
CNC software V6.15 SP4

2024-03-18

· Archiving / series commissioning: The new display machine data 9115 $MM_SAVE_CREDENTIALS can be used to set whether files with login data are saved in an archive. $MM_SAVE_CREDENTIALS 0 = Dialog is opened (default) 1 = No dialog, login data is not saved 2 = No dialog, login data is always saved
For MD9115=0 and existing login data, a message box is displayed: “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?”
This prompt can be answered with Yes or No. If the answer is “No” or MD9115=1, the following files are not saved: Logdrive.ini, Logdrive_ini.sav, Logdrive.oldstyle Extdev.ini UserDataBase.xml
· It is possible to integrate a VNC Viewer as an operating area in SINUMERIK Operate for access to a remote PC. The VNC area configuration dialog can be found in the area Commissioning / HMI / Operating area menu (VSK 5) With softkey “Level >>” you can switch through all of the operating area softkey bars. The first available operating area softkey is always automatically selected in each displayed bar. With the “Change” softkey, the parameters of a blank softkey or of a VNC operating area softkey can be changed. The following parameters can be configured: VNC connection
– Type: Only “VNC” (or <blank> for deleting) possible. – Computer name: IP address or DNS name of the host on which the associated VNC server is running. – Port: TCP port that the server uses. – Password (optional): Password for protected access to the VNC server. – Operate Header: Determines whether the header of Operate is to be visible, while the VNC area is selected and in
the foreground. – Position 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 will be displayed. – Softkey text: Either language-independent text for labeling the softkey or text ID for a language-dependent
foreign language text. A text ID can only be used in combination with the specification of the text file and of the text context. – Text file: Name of the text file which includes the context and the foreign-language text. – Text context: Context of the foreign-language text. – Softkey icon: Name of an image file which will be used as an icon for the softkey. Note: The function keys F1 to F12 are not passed on to the remote PC by default.

5.6 Notes on SINUMERIK STEP 7 Toolbox
This toolbox can be used for the engineering of the SINUMERIK hardware. The software is an option package for TIA Portal STEP 7 Professional.
Following Hardware is supported: · SINUMERIK ONE STEP 7 Toolbox V17 update 12 NCU 1750, NCU1760: V6.13, V6.14, V6.15, V6.20
· SINUMERIK STEP 7 Toolbox V18 SP1 update 2 NCU 1740, NCU 1750, NCU1760, PPU1740: V6.15, V6.20, V6.21, V6.22
· SINUMERIK STEP 7 Toolbox V19 NCU 1740, NCU 1750, NCU1760, PPU1740: V6.15, V6.20, V6.21, V6.22, V6.23

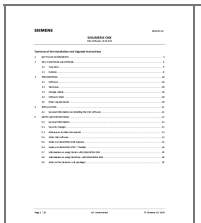
Supported functions:
· Creating a hardware configuration · Configuring the drive telegrams (telegram configuration) · Support of SINUMERIK Safety Integrated (F-PLC) · Import of SINUMERIK DB2 alarm texts (Area 500.000 … 999.999) · Export of all alarms (e.g., system diagnostics) and text lists for SINUMERIK ONE Operate · Creating SINUMERIK

PLC DSF archives with alarm texts, optional with Safety Summary · SINUMERIK PLC Basic Program V6.15, V6.20, V6.21, V6.22, V6.23 · SINUMERIK PLC Basic Program plus V10.x, V11.x, V12.x, V13.x · SINUMERIK ONE PLC Basic Program plus V2.1 including SINUMERIK User Alarms · Upload device from SINUMERIK DSF Setup archive to the project as a new station · Show hardware and and software component version of NCU in online mode from V6.22 in diagnostics and online editor on NCU level. · Use of NC-variables · TIA Portal Add-in “Sinumerik-AxisDriveAssignment.addin” for axis to drive assignment. · Openness clients to support the automated workflow
o Axis drive assignment o Archive generation for the assignment of PROFIsafe addresses to F modules (F-Address Assignment) o Archive generation of the complete PLC Configuration and the PLC User Program o Configuration data o Simple mapping to support the creation of the safety program for machine series o F address assignment (archive creation for the assignment of PROFIsafe addresses to F modules)

5.7 Information on using Starter with SINUMERIK ONE
The starter version V5.4 SP2 HF1 (version V5.4.2.1) with SSP SINUMERIK V5.2 SP3 (internal version 05.20.73.00_07.10.08.00) must be used. Note: Standard drive commissioning and commissioning of SINUMERIK Safety Integrated plus must be carried out via SINUMERIK Operate or SINUMERIK Commissioning and are not supported by the starter. Offline commissioning via starter is also not supported.
5.8 Information on using Startdrive with SINUMERIK ONE
Startdrive is not released for SINUMERIK ONE
5.9 Note on the Siemens cycle packages

The following cycle packages are contained on in the CNC SW 6.15 SP4: · Technological cycles (standard cycles) · Measuring cycles · ISO cycles · ShopMill cycles · ShopTurn cycles · Grinding cycles · AST cycles · Adapting cycles

Documents / Resources

	SIEMENS SINUMERIK ONE CNC Software [pdf] User Guide CNC software V6.15 SP4, SINUMERIK ONE CNC Software, SINUMERIK ONE, CNC Software, Software

References

- User Manual

SIEMENS

CNC Software, CNC software V6.15 SP4, SIEMENS, SINUMERIK ONE, SINUMERIK ONE CNC Software, Software

Leave a comment

Your email address will not be published. Required fields are marked *

Comment *

Name

Email

Website

Save my name, email, and website in this browser for the next time I comment.

Post Comment

Search:

e.g. whirlpool wrf535swhz

Search