CNC Software with ShopTurn 1.5 SP3 HF2

SINUMERIK 840D sl, CNC SOFTWARE 6-3)		6FC5842-1XG11-1YA0
with ShopTurn HMI, in 6 languages (EN,DE,FR,IT,SP,V-CH),		
on CF card; with license)	Export	6FC5842-1YG11-1YA0
SINUMERIK 840D sl, CNC SOFTWARE 31-5)		6FC5842-3XG11-1YA0
with ShopTurn HMI, in 6 languages (EN,DE,FR,IT,SP,V-CH),		
on CF card; with license)	Export	6FC5842-3YG11-1YA0

Information about installing and operating the software on SINUMERIK NCU 710.1, NCU 710.2, NCU 720.1, NCU 720.2, NCU 730.1, NCU 730.2.

The CNC software 1.5 SP3 HF2 is a software update status (internal Version 01.05.03.02.001) and comprises the following components:

- LinuxBase (NCU) V02.50.34.00
- NCK software V72.08.02
- PLC-BeSy V20.70.41 (PLC 317)
- PLC-FB15 V01.05.09
- CP V01.40.03
- MCP-Client V01.04.35
- SINAMICS software V02.50.55.00
- HMI-Embedded V07.50.24.00
- ShopTurn V07.50.24.00
- Technology Cycles V07.05.09.01
- Measuring Cycles V07.05.08.01
- Cycles ISO Compatibility V06.05.04
- Adapting Cycles V07.01.10

Functional improvements in SW 1.5 SP3 HF2 compared to SW 1.5 SP3 HF1
AP01045685
Sometimes wrong positions from SME25/SME125/SMC20.

Functional improvements in SW 1.5 SP3 HF1 compared to SW 1.5 SP3

AP00790072	Technology-independent measurement: Problem with CYCLE978 with CYCLE800
AP00966091	CYCLE95: Excessive minimum value for retraction path VRT
AP01005083	Pixel error when moving the mouse onto Ergoline-OP

1. Installing the CNC software:
For installing the CNC software you will need the NCU service system V02.50.21 or higher on USB stick.
New installation can only be performed either using the NCU Service System and TCU with keyboard or WinSCP. New installations without full keyboard or WinSCP can only be performed using an USB stick which has been prepared for automatic upgrade (cf. General information).
The software may only be installed on a SanDisk CompactFlash 5000 1GB. MLFB / order no. for the empty card: 6FC5313-5AG00-0AA0.

1.1 Installing the NCU service systems on USB stick:
The NCU service system is located as a USB stick image in the emergency_bootsys_ncu directory on the supplied DVD. The installdisk.exe copy routine is located in the same location. Connect a 512MB or 1GB USB stick, recommended type Siemens 6ES7648-0DCx0-0AA0, to your programmer ("PG") or PC (with Windows XP) and note down the letter of the drive to which the stick is connected. Run the installdisk routine to copy the NCU service system to the USB stick:
installdisk --verbose --blocksize 1m <image file> <drive letter:>

Example:

(-- means 2x minus sign)
installdisk --verbose --blocksize 1m D:\ emergency_bootsys_ncu\linuxbase-512M.img X:

or

installdisk --verbose --blocksize 1m D:\ emergency_bootsys_ncu\linuxbase-resize.img X:
(When using the...resize.img, boot the NCU once from USB stick in order to unpack the image on the stick).

It is best to copy the image in a DOS shell.
To this end, you will need administrator rights on your PG/PC.
After completion of installdisk, remove the USB stick from your PG / PC and insert it again. Under Windows, you will now see an empty USB stick. Now copy the desired CNC software (file with ".tgz extension) from the supplied DVD, ncu_sw directory to your USB stick.
You may insert the USB stick into one of the two USB ports on the NCU 7x0. When the NCU is turned on, it will boot from this USB stick. The operation is always performed via a TCU connected to the NCU, or via PG / PC using WinSCP with "Open Terminal".
For a description of the operation of the NCU service system, please refer to the /IM7/ Documentation on the supplied DVD.

1.2 Upgrading an NCU 7x0 with CNC software, version 1.5 and higher

With an upgrade, all user data on the CF card in the directories /user, /addon, /oem, and the license key are maintained. For the upgrade of systems equipped with several NCUs / PCU50, please refer to Section 7 "Networking".

- Boot the NCU from your USB stick (see 1.1).
- Select the Service Shell using F2 / F3.
- Log in with user name "manufact". This may require an external keyboard, since user names must be written in lower case letters.
- Enter the following command after the operator prompt $ in the service shell:
  sc restore -update /data/<cnc-sw>.tgz
  The software update is completed when prompt $ is displayed again.
- Remove the USB stick.
- Boot the NCU (Power ON / OFF).
- Now you can start up the PLC, drives and NC.

1.2.1 Upgrade using WinSCP

Systems without TCU at the NCU can be upgraded with WinSCP, for example on a PCU 50. For this purpose, log on at the NCU with WinSCP as "manufact" and copy the required CNC software (file with extension .tgz) from the ncu_sw directory on the supplied DVD to /card/user/system/home/manufact on the NCU. Then enter the following command under "Commands / Open Terminal":
sc restore -update -force <cnc-sw>.tgz
The software is installed when the "Syncing disks .... done" message is displayed. Afterwards, the NCU must be rebooted. The <cnc-sw>.tgz file on the NCU should be deleted.
Note: During the installation, messages may be displayed claiming that the communication partner has not reacted for some time. You can ignore these messages, but please do not cut the connection in this case.

1.3 New installation on CompactFlash card or upgrade from SW 1.3, SW 1.4 onto Software Version 1.5

IMPORTANT: This procedure completely deletes anything existing on the CompactFlash Card; this applies to an empty CompactFlash Card as well.

- Boot the NCU from your USB stick (see 1.1).
- Select the Service Shell using F2 / F3.
- Log in with user name "manufact"; when the CompactFlash Card is empty; use "admin", password "SUNRISE" instead. This may require an external keyboard, since user names must be written in lower case letters.
- Enter the following command after the operator prompt $ in the service shell:

sc restore -full /data/<cnc-sw>.tgz
The software update is completed when prompt $ is displayed again.

- Remove the USB stick.
- Boot the NCU (Power ON / OFF).
- When the system is started with TCU(s) for the first time, the TCU(s) must be logged on to the system. Enter the TCU name (default TCUx) and the address of an assigned MCP.
- Now you can start up the PLC, drives and NC.

1.4 Systems without TCU

If, for the software installation corresponding to 1.2 or 1.3 there is no OP with TCU available to operate the NCU-Service System, then operator control is also possible from the PCU50 or PC/PG using the WinSCP (Commands / Open Terminal) or VNC viewer.
Under certain circumstances, it is not possible to install the software without NCU Service System on the USB stick as there is not enough free memory space on the CF card.

1.5 Automatic upgrade using a boot-capable USB stick:

Using an appropriately prepared USB stick >= 1GB, it is possible to perform an automatic “restore –update” or “restore–full”. To do this, a script (autoexec.sh) and the CNC software must be copied onto the stick. The name of the script must not be changed. The CNC software copied to the stick must be renamed as either update.tgz or full.tgz. The SIM/NCK rotary switch on the NCU must then be set to position 7 and the NCU booted from the stick. The script now only runs when the rotary switch is in the 7 position. Otherwise, the system is booted from the stick and a branch is made into the service menu. From the CNC software designation, the script decides whether a “restore–update” or a “restore –full” is performed. In addition, before the restore operation, the script backs up the data on the CF card. The progress of the individual script steps can be tracked on the 7-segment display or also on the internal HMI. The precise significance of the displays and the individual steps are described in a separate document. The script is included on the DVD supplied.

2. New functions in NCK 72.00.00 (or later)

- The machine manufacturer can set in machine data $MN_ASUP_EDITABLE whether the Siemens System ASUP from _N_CST_DIR or a user ASUP from _N_CUS_DIR (_N_CMA_DIR) shall be executed with RET or REPOS events.
- ET200S-F I/Os are supported for Safety Integrated.
- GETFREELOC can be called up several times during empty location search in the tool management.
- The machine data $MA_JOG_AND_POS_MAX_JERK[axis] is now active with NEWCONF.
- The couplings TRAIL, LEAD and ELG are implemented for (BCS) and (MCS).
- Fixed-point approach in JOG: Max. four fixed points can be defined via the machine data $MA_FIX_POINT_POS[]. The number of valid fixed points is determined via the machine data 30610 $MA_NUM_FIX_POINT_POS.
  Approach procedure:
  - Select JOG mode
  - Activate via PLC interface (see 4.3)
  - Traverse axis using axis key (continuous resp. incremental) or handwheel until it stops automatically. The axis stops at the defined fixed point without emitting an alarm message.
- The number of possible parameterizable transformations has been extended from 10 to 20.
- Generic transformation has been extended to include a 7th axis.
- The nesting depth of subprograms and control structures has been increased from 8 to 16.
- The current axial setpoint and actual braking path can be read using system variables.
  Basic coordinate system:

$AA_DTBREB[AX]	Total braking path
$AA_DTBREB_CMD[AX]	Path – single interpolator
$AA_DTBREB_CORR[AX]	Overlaid movements
$AA_DTBREB_DEP[AX]	Coupled axis

  Machine coordinate system:
  $AA_DTBREM[AX]

$AA_DTBREM_CMD[AX]
$AA_DTBREM_CORR[AX]
$AA_DTBREM_DEP[AX]

- The number of safe cams (SN) has been increased to 30 with solution line.
- Empty location search within turret magazines can be performed using the same setting options as for chain magazines.
- The program attribute ACTBLOCNO is used to output the block number from the current block containing the alarm upon alarm output with active DISPLOF.
- Machine data $mc_ign_prog_state_asup defines whether the display of the ASUP name shall be suppressed. The ASUP is performed “silently“.
- The function Any–Asup–Active sets the new channel-specific VDI signal “anyAsupActive“ in DB.[channel].DBX378.0.
  The new channel-specific VDI signal “silentAsupActive“ is set in DB.[channel].DBX378 if an ASUP is performed silently.
- Interruptability of ASUPs:
  Certain Prog-Events shall not be prevented or stopped by VDI signal NC Stop DB[channel].DBX 7.3). This ensures that the Prog-Event is executed completely independently of the VDI signal.
  The VDI signals NC Stop at the block end DB[channel].DBX 7.2, NC Stop “Axes plus spindle” DB[channel].DBX 7.4 are discarded accordingly.
  The function is activated by the bit-coded machine data $MC_PROG_EVENT_IGN_STOP for any Prog-Events.
- The subprogram return RET(...) allows you to return to a block number or label of the calling program. This function has been extended such that you can also specify any string as return target.
- Time monitoring on synchronous spindles:
  When reaching setpoint-side synchronization, two time windows are initialized which are used to monitor the time span until actual value-side coarse resp. fine synchronization is reached.
  The size of the time windows is taken from MD 37240 $MA_COUP_SYNC_DELAY_TIME[0/1].
  Index 0 for fine synchronization and Index 1 for coarse synchronization. The value of zero deactivates the respective monitoring mode. If actual value-side coarse/fine synchronization is not reached before the relevant time window has expired, Alarm 22018 is output.

3. General information:

- After the new installation of CNC software 1.5 on a CompactFlash card, a general reset of the NCU and PLC must be performed: Turn the SIM / NCK switch to position 1, the PLC switch to position 3 and perform a POWER ON.
- NCU710 / NCU720: The PLC program must initiate an immediate (approx. 20 secs) reaction to alarm "2120 NCK fan alarm" by stopping the axes in a controlled manner.
- NCU730: The PLC program must initiate an immediate (approx. 20 secs) reaction to alarm "2120 NCK fan alarm" by stopping the axes in a controlled manner if **no** double-fan module is used.
- NCU710 / NCU720: The NCU will not power up if a fan module is not installed or if the fan is defective. This applies for NCU 730 with double fan module only in the case of failure of both fans.
- Alarm 15122 "%1 data have been restored, of which %2 machine data, %3 error"
  If %3 indicates that more than zero errors have occurred, it is not advisable to continue working with the data. To avoid further problems, you should read in an up-to-date backup copy of the data.
- It is not always possible to port an NCK series startup file from 840D powerline without processing to 840D sl.
- Prior to overwriting a licensed CF card, a backup of the license key is absolutely necessary The key is available in file ‘keys.txt’ and is stored under the path *card/keys/sinumerik*
  The backup of the key can be carried out, for example, with WinSCP, by the PG / PC.
- Each license is assigned to a specific CompactFlash card (card ID) and is only valid on the assigned card.
- Mini handheld unit
  An additional hardware is now required to operate a mini handheld with handwheel. As previously, the inputs are connected directly to the PLC I/Os or can alternatively be integrated into the input image of the MCP IE (X51…) using a relevant connection kit. For transmitting the handwheel signals, a machine

control panel (Ethernet or Profibus) or a handwheel Profibus connection module is now required as for the type MPI handheld unit.

- Data backups should be read in in the order NC, PL, drive. It will therefore make sense to create the NC, PLC and drive archives separately.
- Contour handwheel and speed override using a handwheel are only release with Profibus MCP.
- The RESET button on the NCU can be used to restart the NCU module. It is not ensured that this will also ensure a restart of the whole system in all configurations. This also applies for the PI service of PLC with FB4.
- The catalog /siemens/system/etc. contains a template for the current standard basesys.ini.
- With interconnected systems, we recommend to assign names to the individual stations in the basesys.ini (e.g. host name=NCU-2).
- If you wish to interconnect systems, the entry “SyncModeDHCPD_SysNet=ON_MASTER” in the basesys.ini must be declared a comment before networking the DHCP master. Power-up sequence when starting up the system for the first time after being interconnected: DHCP master (so far DHCP server), then all other stations.
- By default, Port 102 on X130 is blocked. If you wish to access the controller via X130 using STEP 7 or HMI, you have to activate Port 102.
  This can be achieved using 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 the VNC Viewer.
- The HT2 Ethernet handwheel is configured at the fifth handwheel interface.
  Example for three Ethernet handwheels. The third handwheel is located in HT2:

MD11350[0]=7	11351[0]=1	11352[0]=1
MD11350[1]=7	11351[1]=1	11352[1]=2
MD11350[2]=7	11351[2]=1	11352[2]=5

- **HT2**: The PLC application example for the HHU included with the Powerline toolbox cannot be transferred to the HT2 without several changes. The NCVAR variable descriptions must be adapted accordingly.
  FC13 now supports the visualization of four lines. As a result, the meaning of the “ROW“ parameter has changed as follows:

  0 = No display output
  1 = 1st line
  2 = 2nd line
  3 = Line 1 and line 2 alternately
  4 = 3rd line
  5 = Line 1 and line 3 alternately
  8 = 4th line
  F = Automatic change of all lines

  The display contents must be stored in the field “ChrArray“type string[64].
  The upper four keys are transferred to the PLC input image
  (the LEDs under these keys cannot yet be addressed).

  Sample parameter setting for the HT2 at FB1 (OB100):

```
BHG             :=5
BHGIn           :=DB100.BhgIn_170
BHGOut          :=DB100.BhgOut_150
BHGStatSend     :=DB100.DBD80
BHGStatRec      :=DB100.DBD84
BHGInLen        :=
BHGOutLen       :=
BHGTimeout      :=
BHGCycl         :=
BHGRecGDNo      :=2 // corresponds to the setting of the rotary switches * of the terminal box resp. MPP.
BHGRecGBZNo     :=2 // corresponds to the setting of the rotary switches * of the terminal box resp. MPP.
BHGRecObjNo     :=1
BHGSendGDNo     :=2
```

```
BHGSendGBZNo   :=1
BHGSendObjNo   :=1
BHGMPI
BHGStop
BHGNotSend
```

* The rotary switches S1 and S2 at the terminal box resp. MPP determine the DIP number used to register the node in the system (here, S1=0, S2=2). Address “0” is not permissible, as, with this setting, the DHCP Server cannot assign any addresses.

- **HT2:** The update rate of the display and the transfer of the handwheel signals (this improves the velocity control of the handwheel) can be improved as follows:
  Copy the mcp_client file in the directory card\siemens\sinumerik\mcp_client**mcp_client.ini** into the card**users**\sinumerik\mcp_client directory and in the HT2 section, change the entry **cycleMCPSend = 50**. The control must then be powered down/powered up.
- Alarm 46xx: The machine data for the Ethernet handwheel must be checked.
  Example for the first handwheel: MD11350[0]=7, 11351[0]=1, 11352[0]=1
- Combinations of NCUs with Sinamics Sxxx CUs have not been released. Exceptions are only permissible after prior consultation.
- Spindle traversing with limited torque (Focon) or spindle traversing to fixed stop:
  The performance display at the HMI is calculated as 100% of the limited torque and output.
- With NCK V67.05.03 or later and ADI4 board V01.04.04, the ADI4 board can be operated on 840D sl with internal PLC 317.
- As before, Dbsi 1 is not released for SINUMERIK with integrated drive control.
  NCU Bios V1.4 (or later) is required to operate 1GB CompactFlashCards in a NCU730.
  Upgrades are only required infrequently so that the BIOS is only available through e_support.
- Just as before, in conjunction with 840 D sl, the Starter is only released for diagnostic purposes.

4. Information about the NCK:

- If the number of part programs set via machine data nearly corresponds to the number of actually existing part programs, the value in the machine data must be increased (due to additional cycles).
- The interpolatory traversing of positioning axes is canceled with alarm 8031 „...Axis has not IPO functionality.“
- SPI and PW have (already always) been keywords and cannot be used as axis identifiers.
- The G643 function (block-internal smoothing) has been released for applications in the tool change area (e.g. optimizations for approaching the tool change position). The function has not been released for applications in the machining process.
- The G644 function (block-internal smoothing) has been released for applications in the tool change area (e.g. optimizations for approaching the tool change position). The function has not been released for applications in the machining process.
- Software variant for a max. of six axes: The default setting for the number of axes is 3.
- The "Extended measurement" function with distributed measuring input on the SINAMICS modules is not available (MEAC, MEASA, MEAWA).
- Gaps in the channel sequence are not permitted in systems with a maximum of two channels.
- Access to drive data via $nn_nn system variables is not enabled. The only available system variables are those transferred in telegram 116.
- Extended stop and retract (ESR) has not been enabled.
- ASUBs can now be started along the line of the 840Di functionality via the Profibus PLC I/Os. Machine data settings must be selected by the same method used on the 840Di system. There are no high-speed NCK I/Os mapped in DB10 of the PLC. However, the ASUB start options used in earlier versions (i.e. from DB10 or FC9 via PLC program) are still available.
- High-speed ET200 Simatic modules can be used as a replacement for the DMP modules used on Powerline. For this, you can operate one of the two Profibus lines in synchronism with the clock with PLC317 in combination with NCK V 67.07.05 or later (more details are provided in a separate description).
- With a 31-axis software on an NCU 710.1, you can use six out of 31 axes and four out of ten channels.
- Machine data 10008 $MN_MAXNUM_PLC_CTRL_AXES is no longer included in the data record. It has

been replaced by MD19160.

- MD 10062 $mn_posctrl_cycle_delay must be zero. Check the available data backup.
- Machine data 32250 $ma_rated_outval[ ] must have value zero.
- IMD (integrated monitoring & diagnostic) :
  Feedback in GUD variable _PM_MISSING_TOOL_REFRESH after writing the variables to "1" is updated only after image change.
- Safety: There is a new option handling referring to synchronized actions or synchronized action elements. Up to version 1.3 (NCK 62), synchronized action level 2 was set automatically at the same time with option SI. Thus, channel MD 28250, NUM_SYNC_ELEMENTS could be set to a value > 159. This has been changed with NCK version 67. Synchronized action level 2 is no longer included in Safety. Instead there is a new machine data, in which the synchronized action elements for SAFE.SPF are defined, i.e. MD 28251, NUM_SAFE_SYNC_ELEMENTS.
  This machine data can be written with max. value 500 for option SLP_I_O=1, and with a max. value of 5000 for option SPL_I_O=2.
  However, the number of synchronized action elements required should be determined in order to prevent the performance from being loaded unnecessarily.
  With system variable $AC_SAFE_SYNAC_MEM, the relevant number of free SI synchronized action elements can be read.
  If this variable is called prior to starting and after running SAFE.SPF, the difference is the number of elements occupied by SAFE.SPF. This difference adding a certain reserve should be entered in MD 28251.
  Index 1-5 must be indicated when writing MD 35300 via synchronized actions.
- Safety: Changed check sums of Safety integrated into the drive.
  The check sum for these functions must be confirmed again due to new drive-integrated safety functions.
  To this end, enter “95” (safety start-up) in parameter p10.
  Subsequently, copy the parameters of the current check sum to the parameters of the target check sum.
  p9798 to p9799
  p9898 to p9899
  Subsequently, p10 must be set to zero again. This will automatically save the values (p971=1 is set).
  Wait for the process of saving to be completed.
  The processes mentioned above can also be triggered in HMI Advanced under IBN/NC/Safety Integrated using the softkeys "Activate start-up mode" and "Deactivate start-up mode".
- With Software Version 1.5 HF2 / 2.4 SP1 or later, the HT2 can be used as handheld unit.
- The auxiliary function M6 is no longer output as standard after block search.
  Remedy:
  Change the auxiliary function group in MD 22040 $mc_auxfu_predef_group[5] into an unassigned group.
- If the value of MD18210 deviates from the default, MD18210 must be increased by min. 21 MB DRAM to upgrade the system from SW1.4 / 2.4 to SW1.5 HF4 / 2.5 HF 1.
- Tool Manager: There is a new handling option. If several real magazines are used, these must be enabled using the option. One real magazine is included in the basic scope.
  For compatibility reasons, the previous option bit $ON_TECHNO_FUNCTION_MASK Bit4=1 must still remain set. This is, however, not included in the license handling.
- With SMI encoder, the function “parking axis“ cannot be activated without voltage Off/On.
- The function “fixed-point approach G75“ is not permissible with active radius correction G41 / G42 (Alarm 14091 Index 9).
- With fixed-point approach with G75, the fixed point is now approached with non-linear interpolation (RTLIOF). As a consequence, the axes are traversed with the jerk set for positioning axes ($MA_JOG_AND_POS_MAX_JERK).
- The machine data 11295[0..9] is now memory-configuring. An alarm is not output in case of change. In existing archives, all values deviating from the standard must be removed before performing the upgrade.
- If you wish to use the service interface X127 with PG/PC on a NCU7x0.2 with SW 2.6, you have to ensure that the PG/PC network interface is operated in "DHCP Client" mode, that means "Automatically obtain IP address ".

After setting the PG/PC network interface fo a fixed IP (192.168.215.xx), a ping is sent to X127, but you cannot go online neither with STEP 7 nor using a SINUMERIK Operate. The firewall is activated by default.

- The maximum CU or NX drive system load may not exceed 80% (load displayed in parameter 9976[xx]).
- In the “Reset“ state, the NCU load exerted by position controller and interpolator may not exceed **50%**. The current load can be checked under Diagnosis / System resources.

- Enhanced functions and further developments in the NCK:

5. Information about the PLC:

- Toolbox, V01.05.01 or higher, and Step 7, version 5.4 SP4 or higher, are required.
- The current cycle time is no longer available in DB5, but in DB8 instead. The cycle time can also be called via the start information of OB1.
- The PLC module IM153-2 MLFB 6ES7 153-2AA02-0XB0 cannot be used.
  Remedy: 6ES7 153-1AA03-0XB0
- The PLC series startup file must be generated when the PLC is stopped.
  It cannot otherwise be guaranteed that the PLC will switch to cyclic operation when the data backup has been loaded.
- Sinamics parameters are required for star / delta switchover with FC17.
  Prerequisites:
  The relevant DDS/MDS must be set up in the drive.
  P827[0] □ P827[1] □ P827[n] must be set to different values.
  The contactor is switched over by the application=> P833, Bit0=1
  The pulses are disabled by the drive => P833, Bit1=0
  Initial PLC position: Star mode, therefore set bit 21.5 in the axis DB to “1” during power-up.
- The X127 service interface on the NCU is to be used exclusively for start-up and service.
- The machine control panel (MCP) is operated on Profibus or Ethernet. No mixed operation!
- Profibus 1 must be configured so that the CU, supply and drive softkeys are displayed at the HMI.
- The blocks FC1007 AG_LOCK, FC1008 AG_UNLOCK, FC1010 AG_CNTRL have not been released.
- The PLC FORCEN [FORCING] function in conjunction with 840D sl has only been released with some restrictions. It is possible with the PLC317 from the PLC operating system >= V20.70.37 (2.4 SP1 HF11) / PLC319 and higher, PLC operating system
  >= V26.90.10 (2.4 SP1 HF11) and higher.
- With 840D sl, it is not permissible to connect the VDI interface signal DB10.DBX57.3.

6. Information about the SINAMICS drive:

- No parallel use of measuring functions for i and n measurements via starter and HMI Advanced. Starter measurements require a Power OFF => ON and immediate measurement with the starter. Starter measurements must not be performed after HMI Advanced measurements.
- If a SINAMICS component is replaced and the software upgraded, the firmware release must be checked and, where necessary, the original firmware release restored.
  (Key word: macro 150399)
  Firmware releases can be checked in the following parameters:
  Control unit R18 ➔ Firmware release of CU
  Supply R128 ➔ Firmware release of infeed
  Drive MD R128 ➔Firmware release of power unit
  R148 ➔Firmware release of sensor module
- V1.4 / V2.4: For the time being, the starter is still required for determining the motor codes.

- Mains voltages: For smooth operation in the target country the following parameters must be set in the specified order:
  P010 = 1
  P210 = rated mains voltage in the target country
  P211 = rated mains frequency in the target country
  P340 = 1
  P3410 = 4
  P3900 = 3
  Save RAM to ROM
  Power OFF the system; power it ON again on the target network only.
  With the next ON command at ALM a network identification is run and the determined values are saved protected against power failure.
- The APC function increases the NCU load. Prior to using this function a performance estimate must be carried out.
- p1240, p1244, p1248, p1250:
  If Vdc control and DDS switchover have been activated in V2.4, check if correct values have been entered in the contents of indexes >0 of p1240 after upgrading the system.
- DMC20/HUB has not been released.
- With 840D sl and SW 1.5 SP1, 62.5 µs drive cycles have been released. The following conditions apply accordingly:
  - with this setting, two axes can be operated on one CU or NX.
  - only the same clock cycles can be set on a double-axis module.
- Temperature prewarning messages (DB[axis].DBX94.0, DBX94.1) sporadically occur.
  Remedy: Only evaluate the signal in the PLC if this is pending in the VDI interface for more than 1.5 s.
- For SINUMERIK software releases 1.3 and 1.4/2.4, different firmware versions of the components (NCU, NX, Motor Modules, DRIVE-CLiQ motors) involved in the **safety functions** can be combined (mixed) without having to adapt the firmware versions.
  The following applies from SINUMERIK software release 1.5/2.5 and higher:
  The firmware versions of the Motor Modules, Sensor Modules and DRIVE-CLiQ motors (include integrated Sensor Modules) involved in the safety functions must be adapted to the SINAMICS firmware version of the NCU.
  This occurs automatically when booting, if parameter p9826 (firmware, automatic) is set to 1 (standard setting). When using **Safety Integrated,** parameter p9826 (firmware, automatic) must be set to 1 and must not be re-parameterized. For the acceptance test of **Safety Integrated,** the safety firmware versions of all of the Motor Modules, Sensor Modules and DRIVE-CLiQ motors - involved in the safety functions - must be read-out, logged and checked against the following list:
  http://support.automation.siemens.com/WW/view/de/28554461

  Each line in the tables represents a possible combination of safety firmware versions.
- When booting, chassis units must be operated in the open-loop control mode (refer to MD35020).

- **Machine data change as a result of a changed Drive ES SlaveOM**
  The Drive ES SlaveOM is responsible for allocating addresses of drive units connected to Profibus of the PLC. The addresses are allocated in the PLC HW Config. The addresses allocated here must be entered into the NC machine data MD13050[nn] and, with active safety, in MD10393[nn] corresponding to the required axis configuration.
  When a Starter V4.1.n (or later) is installed (later, including the the installation of toolbox V2.6), the “new“ Drive ES SlaveOM is linked in. Contrary to the previous situation, now, an address space for 6 axes is reserved for each CU or NX. This is realized when newly creating or adapting a PLC hardware configuration. As a result, addresses may be shifted or gaps occur.
  This modified address allocation **must** then be communicated to the NC in machine data 13050[nn] and, with active safety, in MD10393[nn], as otherwise the allocations will no longer be correct.
  This is especially true if existing NC data back-ups (archives) are being used as otherwise these will no longer match the PLC hardware configuration.
  Further, in HW Config, the address of telegram 390 (PZD-2/2) of the first NX changes from 6516 to 6530. When using additional NX modules, the address is incremented with an offset of 10. Machine data MD13120[1+n] must be correspondingly adapted.

If existing PLC projects are imported into an S7 environment with “new” Drive ES SlaveOM, then the old (previous) address configurations are kept. However, the drive objects that have been newly inserted into this project receive the “new” address allocation.
A “Standard” (new) address allocation is achieved using “Reorganize addresses”. If, for NX objects, the “Standard telegram 1 (PZD-2/2)” is entered from an existing configuration, then this must be replaced by telegram 390 in order to achieve a standard, new address allocation.

Example:

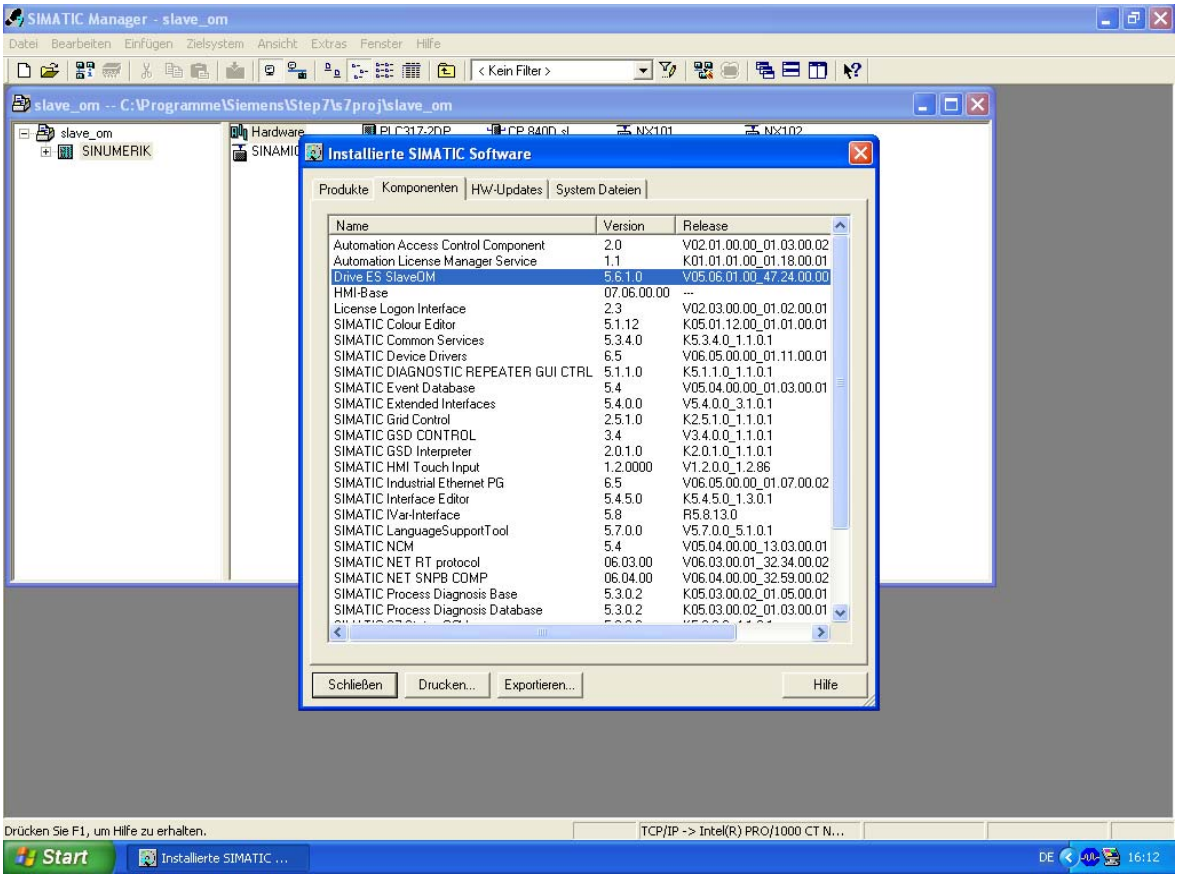
One NCU with a Sinamics Integrated and two NX10 each with 3 axes are configured.

Address allocation with the Drive ES SlaveOM version “old”:
On NX10_1, the 3rd axis has the address 4420
On NX10_2, the 1st axis has the address 4460

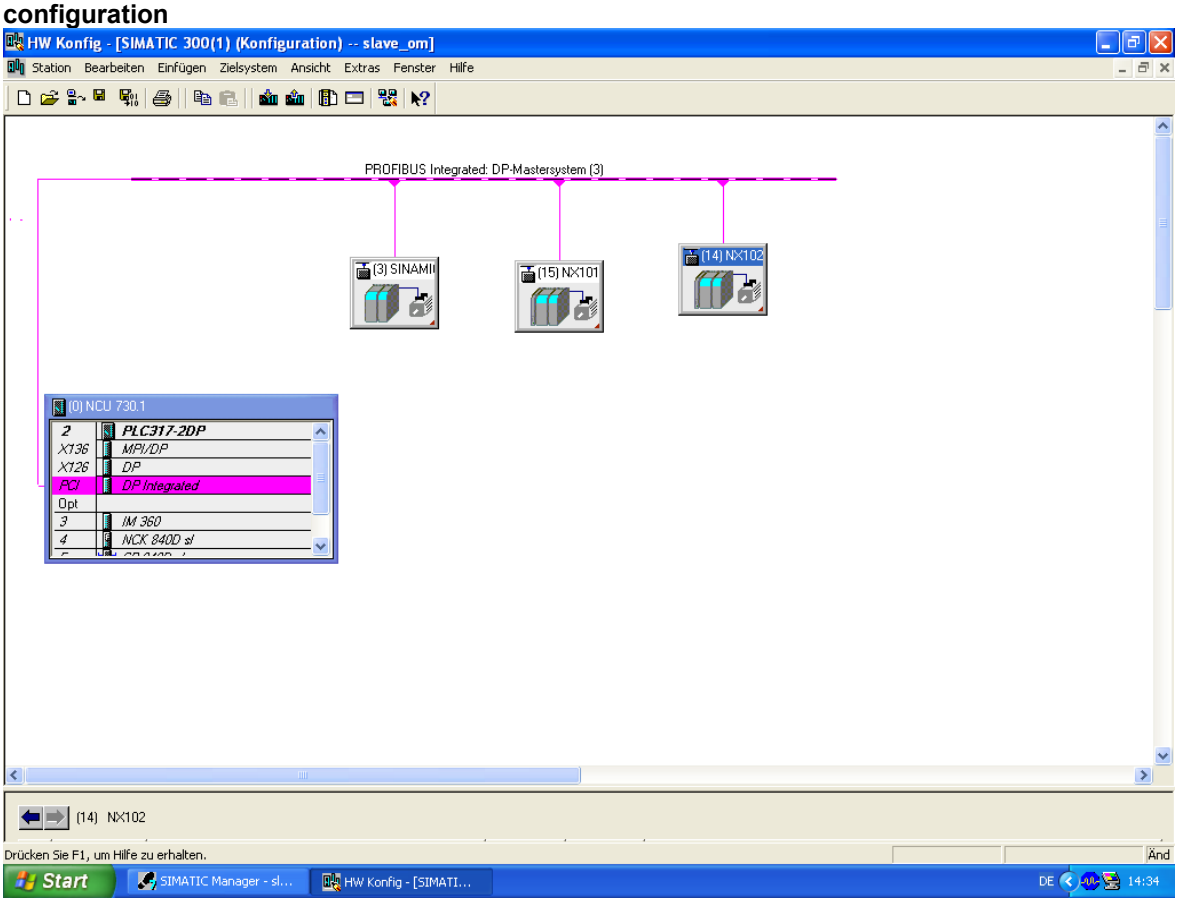
Address allocation with the Drive ES SlaveOM “new”
On NX10_1, the 3rd axis has the address 4420
On NX10_2, the 1st axis has the address 4580

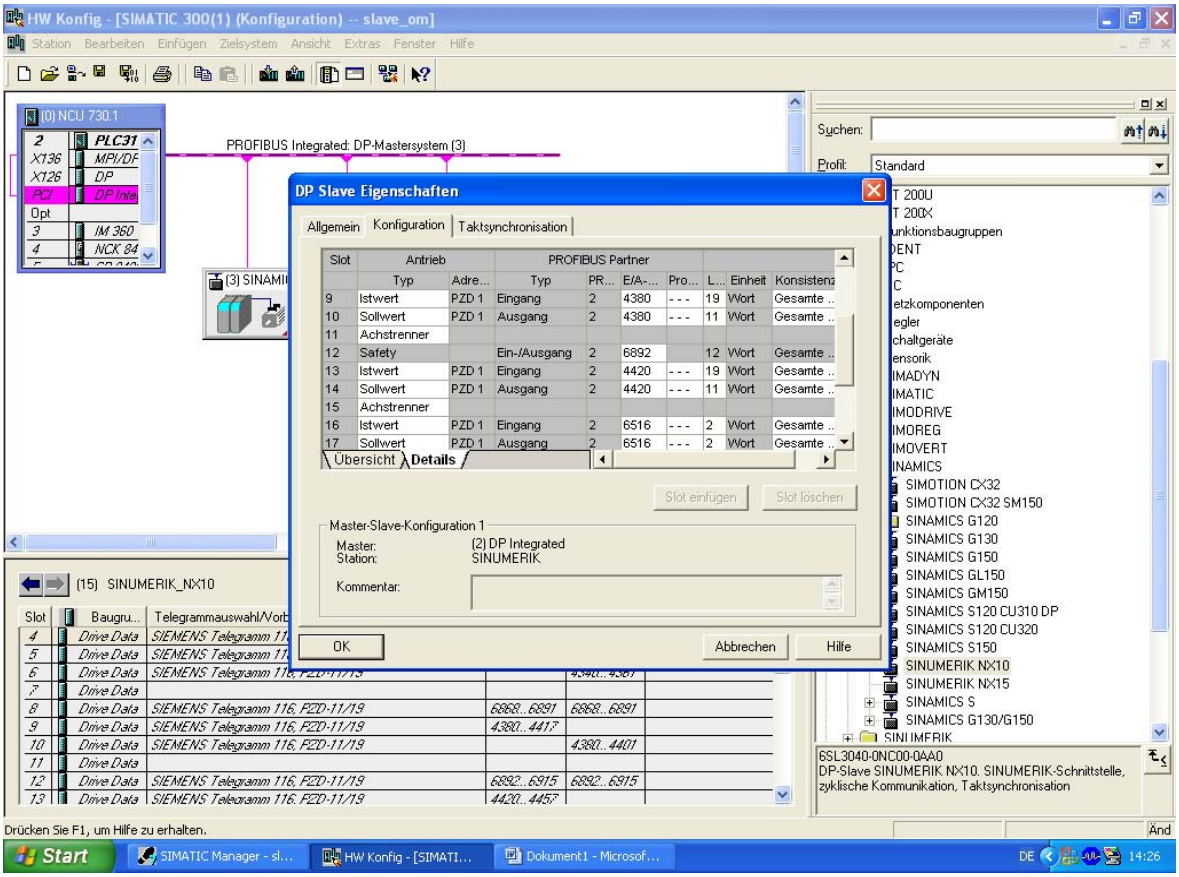
Useful hint:
To ensure that, when reconfiguring existing configurations, the migration to the new addressing can be completely performed, the system should start to initiate the address assignment by the “addresses” key at the lowest slave number.
If, when initiating the address assignment, the note “no change“ is output with each slave, all slaves have received the new addresses.

Version of Drive ES SlaveOM“old”

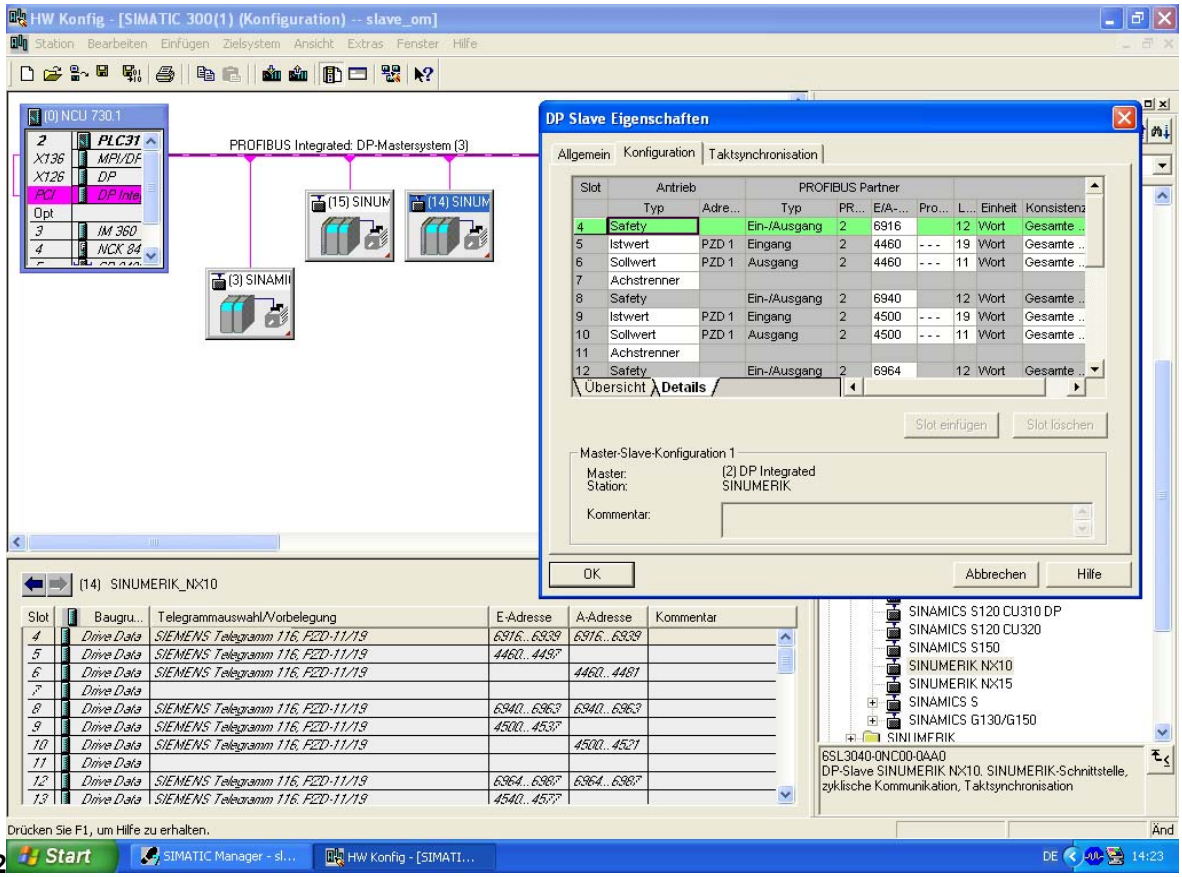


PLC HW configuration



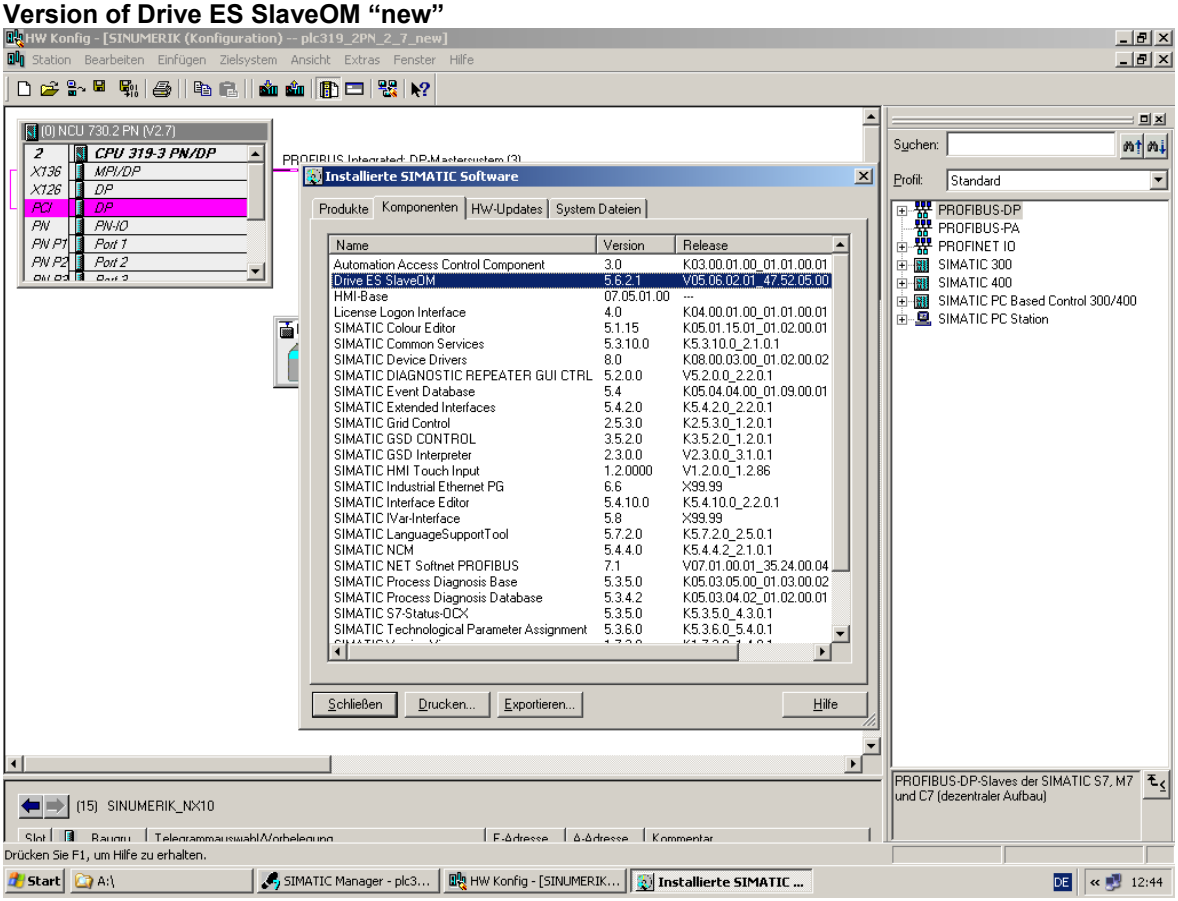


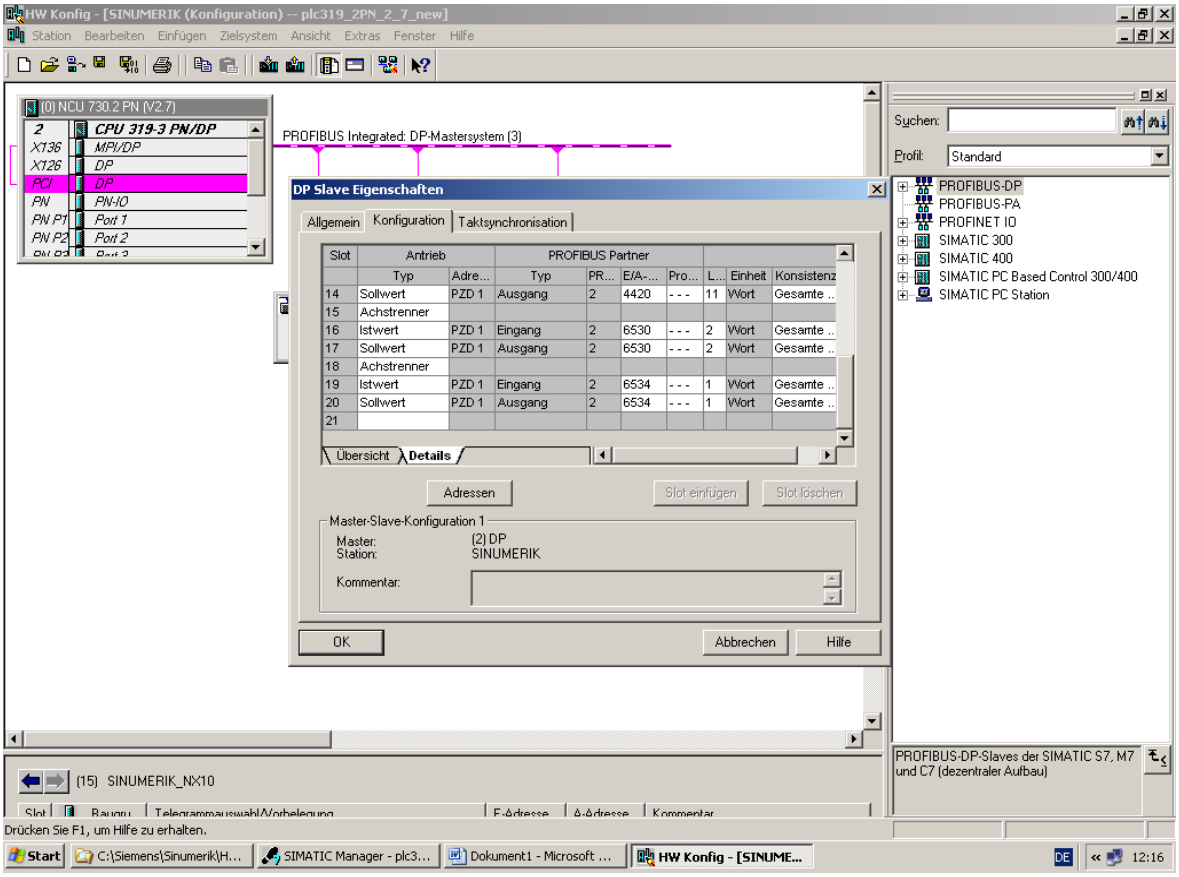
NX10_1



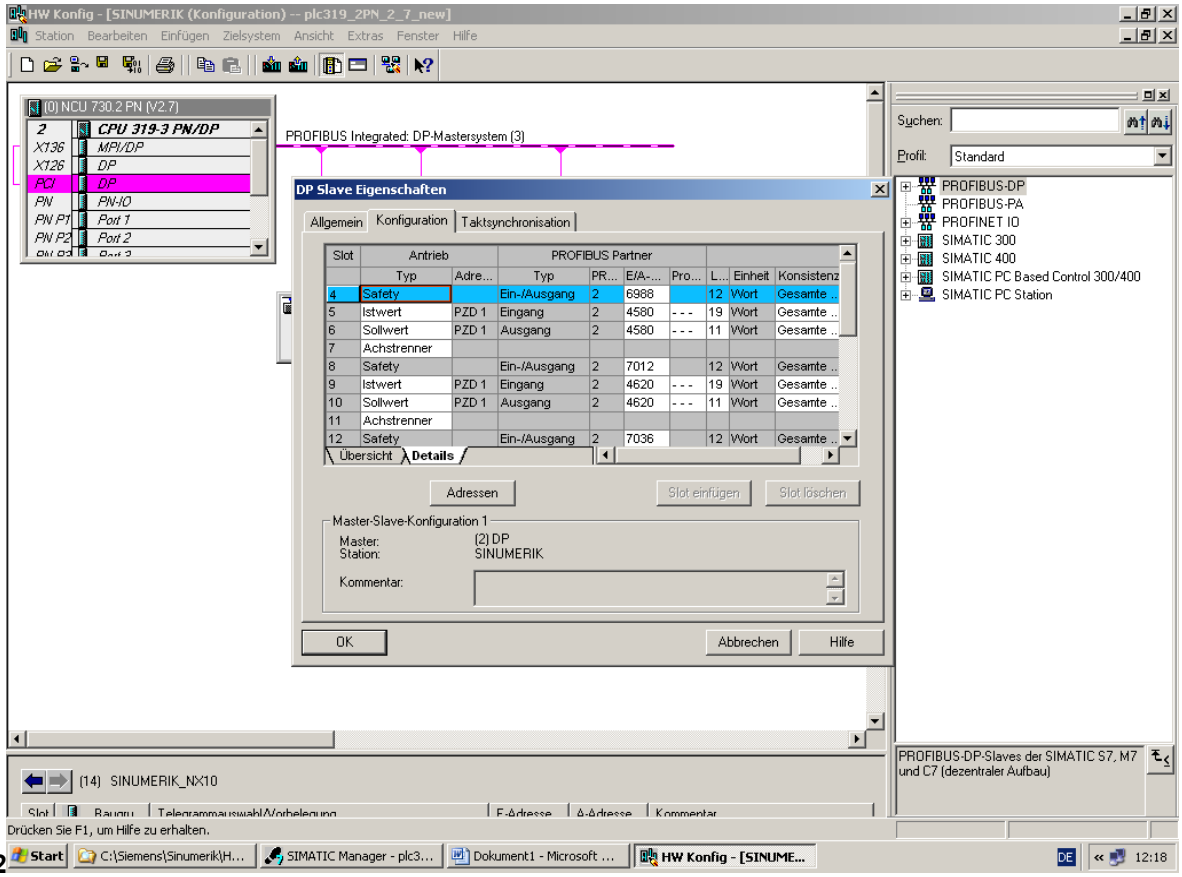
NX10_2

Version of Drive ES SlaveOM “new”



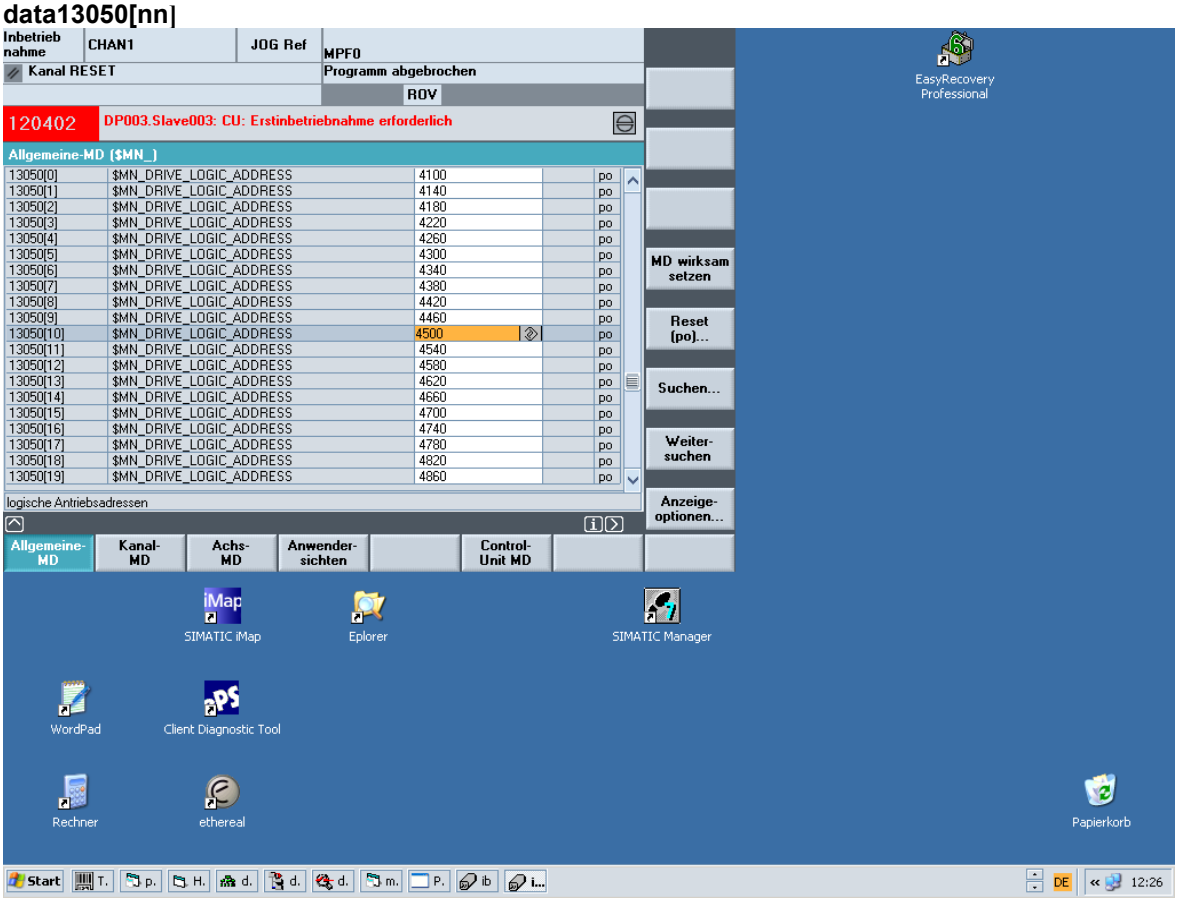


NX10_1

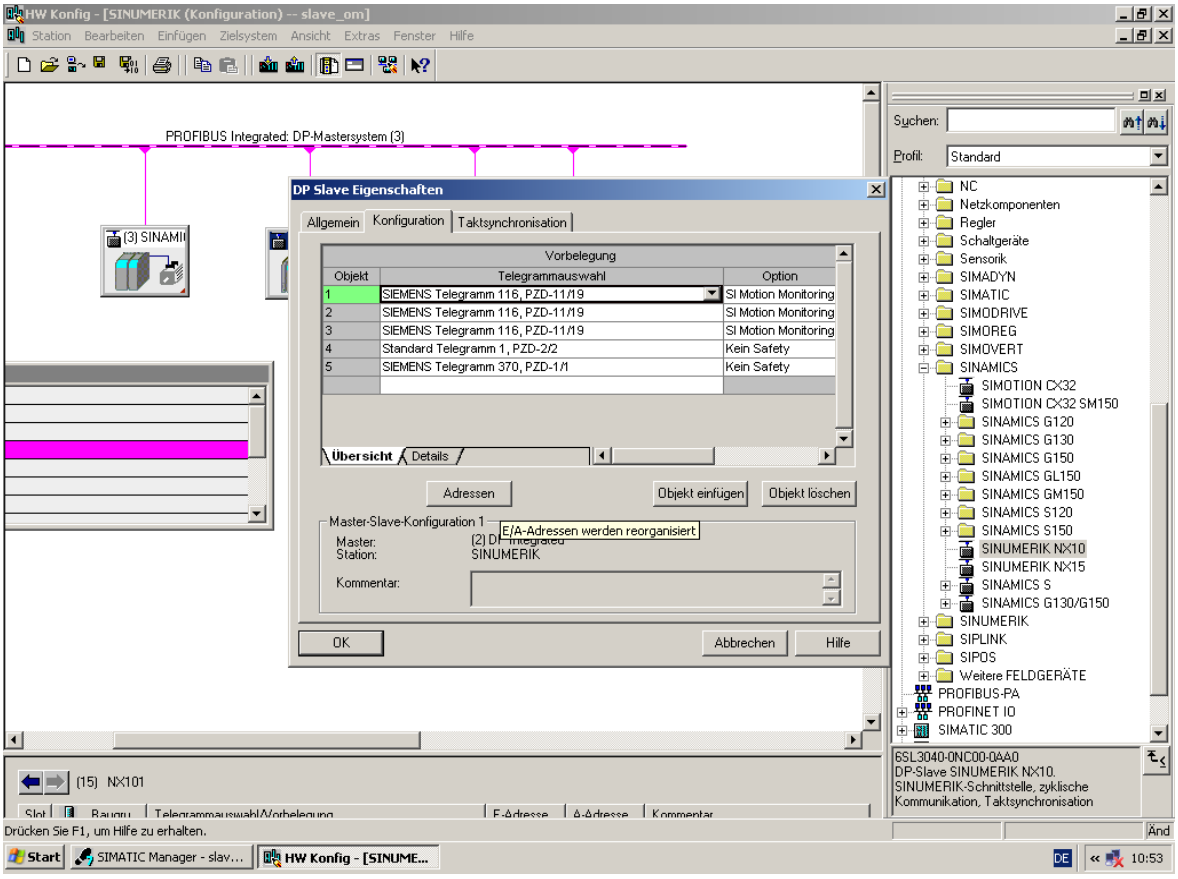


NX10_2

Machine data13050[nn]



Re-organizing addresses



7. Information about HMI Embedded:

- The HMI Embedded software V07.50.xx is started automatically as a component of the CNC software 1.5 in the power-up of the NCU7x0. HMI Embedded is also referred to as an "internal HMI".
- HMI Embedded can only be operated via a control panel with TCU connected via Ethernet to X120 on the NCU7x0.
- The configuring data, such as PLC message and alarm texts and HMI option package ("Wizard"), must be stored on the CompactFlash card in the NCU in the appropriate directories under /card/user/sinumerik/hmi/... or /card/oem/sinumerik/hmi/... (see documentation IM2, BE1). The file system on the CompactFlash card can be accessed in "online" mode only, i.e. when the NCU is running. The data can be transferred either by means of a USB stick or network link via the HMI operator interface (startup / HMI area) or by means of a network link and the WinSCP tool.
  IMPORTANT: File names of configuring files must normally be written in lower case letters.
- Display MD 9990 SW_OPTIONS has no significance. All HMI options must be activated through license management. Note: Network drives are no longer an option, but display MD 9990 bit 0 must be set, for ShopMill / ShopTurn MD19730 HMI_FUNCTION_MASK[0] Bit 2 = 1
-
- The option "Additional 256MB HMI user memory on the CompactFlash card" is required to be able to store part programs and archives on the CompactFlash card ("local drive"). NOTICE: An NCK general reset deletes the option and renders the "local drive" inaccessible until the option is set again. Existing data stored on the card are not affected.

- If you are operating an NCU without TCU (i.e. with HMI Advanced on PCU 50), it is advisable to deactivate HMI Embedded. You can do this by executing command "sc disable hmi" in the service shell.
- OP keyboards can be switched between upper case and lower case letters by simultaneously pressing Shift+Ctrl. To be able to use the characters represented when pressing Shift (e.g. Shift A => [ ), capitalization must be active.

7. 1 Limitations in HMI Embedded:

- Mouse operation is not supported, operation via touch panel (also not with HT8) is not permissible.
- Copying a file to CF card by overwriting an existing file with the same name will cause an "internal error" message. The file with the same name must be deleted before.
- Display MD 9210 USER_CLASS_WRITE_ZOA is not active for "active work offset" in the Parameter area.
- Display MD 9900 MD_TEXT_SWITCH does not have any effect.
- All configured network drives (access to server via Ethernet) must be permanently accessible during the runtime of HMI Embedded.
- For the commissioning of the SINAMICS drives, an external HMI Advanced, software version 7.5 or higher, or commissioning tool 7.5 is required (the CD SINUCOM includes the commissioning tool).
- Processing from external sources / EXTCALL is only permissible with single-channel machines.
- Loadable compile cycles cannot be backed up or read in using NC series startup archive. Loadable compile cycles are stored on the CF card of the NCU and retained in case of NC memory reset. They can be handled like HMI configuring data.
- In a part program with EXTCALL calls, there must be at least one other NC block between two sequential EXTCALL calls with a processing duration of at least one second.
- When very big files are saved on CF card, the message "Please wait – File being flashed" is displayed. This message must be acknowledged by pressing the Recall button. If you want to switch off the control after having saved the file, you will have to wait for one second for each MB file size.
- After reading in a NC series startup archive, an explicit restart of the HMI is required, e.g. through NCK reset or operation with CTRL + Q, Enter.
- It is not possible to integrate a user boot screen.

7.2 Improved functionality with respect to SW 1.5 SP2

AP00923145	Search / replace: user interface freezes when changing the access stage
AP00916793	When re-numbering, at the end, the status line is 107%
AP00900491	Exception when editing
AP00934629	ST: Sporadically, it is not possible to operate the HMI
AP00848217	HMI / ShopTurn sporadically cannot be operated after "Executing" in the editor.
AP00948846	Various crashes when reading tool offset values
AP00948842	Various crashes in the HMI Embedded Basis from the gl_win32.dll
AP00948831	Crashes in HMI Embedded Basis from the function txv_format_text ()
AP00862638	The user interface sporadically freezes.

8. Information about the NCU Base software:

- The Linux operating system of the NCU7x0 is also referred to as the NCU Base software, similar to the PCU Base software for the PCU 50. You will find the relevant documentation in IM7 on the DVD-ROM. The NCU base software ensures the start among other things of the NC, PLC, and HMI software during power-up, as well as the booting of the TCU if it is present. The NCU Base software is user-oriented, i.e. you must log in and enter a password in order to acquire specific access authorization. This current login and password protection is only relevant for accessing a Linux service shell or the CompactFlash card using WinSCP. For startup and servicing activities, the login ID is "manufact" and the password "SUNRISE" (case-sensitive!). To open the Linux service shell on the TCU, press keys "Area switchover" and "Recall" (F10 and F9) simultaneously, then "Scan for Servers", "2" or boot from the USB service system.
- For access via the network using WinSCP, enter the abovementioned login and password on the

"Session" screen. This login procedure also applies after the NCU7x0 has booted from a USB stick with the NCU service system provided that executable CNC software is installed on the CompactFlash card. If this is not the case (e.g. if the CompactFlash card is empty), enter the login ID "admin" and password "SUNRISE".

- You can display a list of the syntax of all available service commands in the service shell by running command "sc help".

8. 2 Networking:

- **Please observe the current documentation "Operator Components and Networking (IM5) " !**
- Important system and network settings for the NCU base software have been preset in the file **basesys.ini** in the directory /card/user/system/etc and may be modified there. The original basesys.ini file is located at /card/siemens/system/etc, it is entitled "template-basesys.ini".
  Each NCU within the plant / system network should be assigned a unique ("talking") computer name, together with the entry "Host name=..." in basesys.ini. Permissible characters are uppercases / lowercases, numbers and minus signs.
- The NCU DHCP server should always be switched on.
- When upgrading the system from SW 1.3 / 1.4 please ensure that no old mcp_client.ini file remains in /card/user/sinumerik/mcp_client.

8.2.1 Configuring 1 NCU with TCUs and MCPs

In this configuration, no special settings must be made in the basesys.ini.
The DHCP server and internal HMI-Embedded remain activated.

8.2.2 Configuring 1 NCU with 1 PCU 50, MCPs and TCUs, if required

In this configuration, make the following settings in the basesys.ini of the NCU:

Host name = ...
SyncModeDHCPD_SysNet = ON_MASTER

In general, the internal HMI-Embedded must be deactivated because two HMIs (HMI-Advanced and HMI-Embedded/JobShop) may only be operated on one NCU in particular cases.
On PCU 50, we recommend that you use PCU base software V8.1 SP3 or later.
In the System Network Center, tab TCU Support, select the default configuration "No boot support".
If the system is equipped with a TCU (HT8) that shall be switched to PCU 50 by default, 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*>.

8.2.3 Configuring several NCUs and, if necessary, 1 or several PCUs, TCUs, MCPs

In this configuration, make the following settings in the basesys.ini of the NCU:

Unique host name = ... for each NCU
Unique InternalIP= ....
Unique InternalNetMask= ....
Exactly one NCU with SyncModeDHCPD_SysNet = ON_MASTER

On PCU 50, you have to use PCU base software V8.1 SP3 or later.
In the System Network Center, tab TCU Support, leave the default configuration "Complete TCUsupport" unchanged. Under the tab DHCP Settings, we recommend to set "Sync mode low priority".
All operator stations within the plant / system network (TCUs, HT8, PCU 50) are managed on the NCU with "ON_MASTER", that means the config.ini files as well as the .leases file comprising all IP addresses assigned within the system which are relevant during the runtime are stored there.
The config.ini files are distributed to all other NCUs / PCUs by the master NCU using the service command "sc distribute tcudata".
Via the Service command "sc clear dhcp" and subsequent power ON / OFF of the complete system, the NCUs / PCUs are assigned preset IP addresses. New addresses are assigned to TCUs and MCPs and the .leases file is then distributed to all other NCUs / PCUs.

Important: The System Network Center, tab OPs allows to directly edit on each PCU 50 the config.ini files of all operator stations on the master NCU. The data are automatically distributed to all NCUs / PCUs each time they are changed.

9. Information about technology cycles:
CNC software 1.5 SP3 includes the technology cycles (standard cycles) 07.05.06 in an installable form.

Functionality:

This cycle release includes new functions, improved functionality and debugs. Only the differences with respect to SW 07.05.04.00 are described in this document.

New functions / expanded functionality / function improvements from SW 07.05.08.00 onwards:

2. Changing the cutting cycle (CYCLE95)

When only finishing, the contour is now always machined in the programmed direction - just the same as it is documented. For complete machining the contour is finished exactly the same as it is also roughed; this means that one and the same tool can be used for roughing and finishing.

Documentation

The current documentation for this SW release is as follows:

“Cycles Programming Manual” Edition 01.2008
Order No. [MLFB] 6FC5398-3BP20-1AA0

Compatibility

Prerequisites:

Software HMI:

HMI-Advanced	from SW 7.5

Software NCU:

Software 840D sl NCU 7x0.x[1]	from SW 1.5
with HMI-Embedded	from SW 7.5
with ShopMill-HMI	from SW 7.5
with ShopTurn-HMI	from SW 7.5

For applications involving angled tools with swivel cycle (CYCLE800):

Software 840D sl NCU 7x0.x:	SW 1.5 with NCK from 72.06.00

For using "Advanced Surface" (CYCLE832):

Software 840D sl NCU 7x0.x:	SW 1.5 with NCK from 75.01.00 onwards

Software NCU:

Software 840Di sl NCU	from SW 1.5
with HMI-Advanced	from SW 7.5
with ShopMill-HMI	from SW 7.5
with ShopTurn-HMI	from SW 7.5

Support of option "Advanced Surface" (CYCLE832):

[1] For applications with SW from 2.5, please note Attachment_3!

Software NCU:	
Software 840D sl NCU 7x0.x	from SW 2.6 with NCK from 75.02.00
Software HMI:	
HMI-Advanced	from SW 7.6

Retrofit regulations:

(1) with HMI-Advanced from SW 7.5
- read-in the archive under "Archive" / "Cycles archive"
- re-activate data block GUD7 and the SMAC macro block under "Definitions"

Errors that have been resolved (debugs): The following errors (bugs) have been resolved and in so doing, the following files modified:

Debugs up to SW 07.05.03.00

ARTS+ (RQ-id)	File	Change

Debugs up to SW 07.05.05.00

AP00564163	Cycle support, contour paths	
AP00577260	Cycle support	

Debugs up to SW 07.05.06.00

AP00367372	CYCLE800	
AP00391398	CYCLE93	
AP00582322	CYCLE800	
AP00676028		
AP00422788	CYCLE800	
AP00422789	Swivel in Jog	
AP00499911	CYCLE95	
AP00530120	Swivel in Jog	
AP00580081	CYCLE60	
AP00584240	Cycle support	
AP00598303	CYCLE90	
AP00661184	CYCLE95	
AP00661279	CYCLE95	
AP00684307	CYCLE84	
AP00707196	POCKET4	
AP00710876	CYCLE800	

Debugs up to SW 07.05.07.00

AP00700727	CYCLE832	

Debugs up to SW 07.05.07.01

AP00788944	CYCLE800	
AP00823117	CYCLE800	

Debugs up to SW 07.05.08.00

AP00405585	POCKET4
AP00521756	TOOLCARR
AP00665389	CYCLE76
AP00675013	CYCLE95
AP00734305	Cycle support
AP00786003	Cycle support
AP00788944	CYCLE800
AP00794767	CYCLE93
AP00804683	**PROG_EVENT**
AP00840407	Cycle support

Debugs up to SW 07.05.09.00

AP00330771	POCKET4
AP00401194	POCKET4
AP00401200	POCKET4
AP00425419	POCKET3
AP00494230	Cycle support
AP00495638	POCKET3
AP00496999	POCKET3
AP00543919	POCKET4
AP00603151	Cycle
AP00800253	support
AP00669469	Cycle support
AP00759424	CYCLE72
AP00784321	CYCLE95
AP00875397	POCKET3 POCKET4 SLOT1 SLOT2 LONGHOLE CYCLE899
AP00875936	POCKET4
AP00879888	CYCLE95
AP00884612	CYCLE93
AP00907136	Cycle support
AP00907143	Cycle support
AP00921659	Cycle support
AP00923711	CYCLE800
AP00923719	
AP00938780	CYCLE800
AP00940647	Cycle support
AP00965217	CYCE832
AP00971400	CYCLE93
AP00984973	CYCLE93

Secondary conditions:

(1) Restriction regarding the cycle support in conjunction with several channels and swivel data

sets for B axis kinematics
The "swivel plane" and "swivel tool" functions for swivel data sets to support the B axis must always be programmed in the channel in which the swivel data set is set-up.

(2) Restriction regarding cycle support in conjunction with re-compiling engraving screen
(due to **AP00336388)**:
When re-compiling CYCLE60, leading blanks and blanks at the end of the engraving text are automatically deleted. These must be re-entered.

(3) Restriction regarding CYCLE800 from SZ 07.05.06.00:
NC programs with a CYCLE800 call must be run-in again under the following conditions as the positioning behavior of the axes has changed as a result of errors that have been resolved (debugs):

1. When using angled tools with a basic tool orientation (MD18114 $MN_MM_ENABLE_TOOL_ORIENT=2 and programming using tool parameters $TC_DPV3[] to $TC_DPV5[]) it should be observed that the tool orientation of the machine changes. When approaching the initial state of the machine kinematics [CYCLE800(…,0,0,0,...)], the basic tool orientation is taken into account. This causes the rotary axes to move if the basic tool orientation deviates from the original tool orientation of the machine.
   When changing tools, it should be noted that the basic tool orientation is always updated after the tool change. This is realized using the NC commands TOROT (for G17) or TOROTY (for G18) and TOROTX (for G19).
   Observe the SW release of the NCU regarding compatibility !
2. The NC programs must be run-in again if values are entered in the zero offset of the rotary axes of the swivel data set.
   This does not apply to the rotary axis that rotates around the tool axis (axis C rotates around Z for a machine in the G17 plane).
3. NC programs with the CYCLE800 call and "Direct rotary axes" must also be run-in again.

(4) Restriction regarding cycle support in conjunction with re-compiling with high speed settings
For the following configuration, recompiling without comments from CYCLE832() is not possible:
With NCU SW 1.x and HMI-Adv 7.6.

Notes for using the swivel function (CYCLE800):

(1) Note for using CYCLE800 in conjunction with rotary axis positioning for head/table kinematics:

For swivel additive, parameters $TC_CARR13 and $TC_CARR14 are not deleted.
If the angular range of rotary axes >=720 degrees (-360 to +360), for TCOFR, a range is defined which permits continuous swiveling. This is advantageous if a rotary axis is located in the pole position and is to be swiveled using the 2nd rotary axis.

NC programs with "swivel additive" must be run-in again if the angular range of the rotary axes of the swivel data set >=720 degrees.
Angular range, rotary axis 1 $TC_CARR30[n] $TC_CARR32[n]
Angular range, rotary axis 2 $TC_CARR31[n] $TC_CARR33[n]
with n..number of the swivel data set

(2) Note regarding the behavior when swiveling in the pole position
The behavior when swiveling in the initial state (pole position) has changed (if _ZSD[14] =1 -> refer further below). With this debug, the input values when rotating around the tool axis, can also be reproduced (tracked) at the machine. This can only occur if the rotary axis involved also has the traversing range in the TCARR data.
Example:
Machine with mixed kinematics
Rotary axis 1 -> B around Y

Rotary axis 2 -> C around Z
Programming:
CYCLE800(...., rotation around Z 300 degrees,...)
If, in rotary axis C, a traversing range of 0 ..360 degrees has been entered, axis C must move to 300 degrees. This is also true if, in rotary axis C, -360 to +360 degrees has been entered.
If, for instance, the traversing range of the C axis is 0 to +180, the value of 300 cannot be approached and therefore the expectations of the programmer cannot be fulfilled. In this case, 120 degrees would be approached.

Setting via _ZSD[14]
= 0 CYCLE800: Compatibility
= 1 Evaluation of the input values CYCLE800 in the pole position of the machine kinematics

Commissioning standard cycles for SINUMERIK 840D sl, SW 1.x

Preconditions

- The software release of the standard cycles, which are to be commissioned, must be on the CF card. Otherwise, no additional software tools are required for commissioning.

- Before starting commissioning, it must be carefully ensured that the existing state is archived; if required, read-out the series commissioning archive.

Step 1 - read-in the archive

Read-in the required archive files:

```
cyccust_gr.arc      ;*Archive for user cycles, German
cyccust_uk.arc      ;*Archive for user cycles, English
cycles.arc              ;*Archive with all standard cycles (milling and turning cycles)
cycmill.arc             ;*Archive with all milling cycles
cycturn.arc             ;*Archive with all turning cycles
defines.arc             ;*Archive with definition files
```

In the "Services" area, "Program data" softkey, change to the "Cycle archive" using the cursor keys and open using the Input key. In the "Cycle archive" area, change to "Standard cycles" using the cursor keys and open using the Input key.
Read-in the archive files required using the first vertical softkey "Read-in archive".
It must be noted that the archive files must be individually selected and read-in. After all of the required archive files have been read-in, an NCK reset must be made via the "Start up" operating area, "NCK reset" softkey.

Step 2 - Activate the definition files

In the "Program" operating area, change into the 3rd plane using the etc. key ">". There, press the "Definition files" softkey, then individually select the definitions and activate them using the "Activate" softkey.

Step 3 – Activate the swivel cycle softkey in the "Start up" area

In the operating area "Program", "Standard cycles", open the file "common.com" using the Enter key and in the line

```
;SC616 = STARTUP.COM
```

delete the semicolon. Then, close the file using the "Close editor" softkey.
After this, the "NCK reset" softkey should be pressed from the "Start up" operating area.

Step 4 – Activate the thread chasing softkey in the machine area "Jog"

In the operating area "Program", "Standard cycles", open the file "common.com" using the Enter key and in the line

```
;SC108 = MA_JOG.COM
```

delete the semicolon. Then, close the file using the "Close editor" softkey.
After this, the "NCK reset" softkey should be pressed from the "Start up" operating area.

Using the technological cycles SW 7.5 in HMI-Advanced from 7.5 onwards with SINUMERIK 840D sl, from SW 2.5

The configuration HMI-Advanced from SW 7.5 and higher with NCU 7x0.x SW 2.5 and higher contains two different cycle software releases:

- The total package NCU 7x0.x comprises: NCU from SW 2.5 onwards (with HMI sl SW from 2.5 and higher) with cycle packages from SW 2.5 and higher (technological cycles, measuring cycles) on CF card, these match the cycle support in the editor of this internal HMI;

- HMI-Advanced from SW 7.5 and higher also brings the cycle archive SW 7.5 (technological cycles, measuring cycles) into the archive path.

With this total configuration, the cycles loaded in the NCU can only be operated with the cycle support that belongs to this cycle SW release - but not mixed.

To use the cycle support of the HMI-Advanced from 7.5 onwards, the cycles should be loaded from its archives - but not the cycles from the CF card of the NCU.

10. Notes on the measuring cycles:
Measuring cycles V07.05.07 are included in the CNC-SW 1.5 SP3 in a form that allows them to be installed.

Functionality:

This measuring cycle release SW 7.5 includes new functions, improved functionality and debugs.
New functions from SW 07.05.01.00 onwards:

1. **New measuring cycle CYCLE996 to measure kinematics (only in conjunction with 840D sl and HMI-Advanced):**

   Using the new measuring cycle CYCLE996, multi-axis kinematics

   can be measured at machine tools. This cycle is based on measuring a calibration ball under three different axis positions for each rotary axis of the machine kinematics. The calibration ball is measured internally using the CYCLE997 measuring cycle.

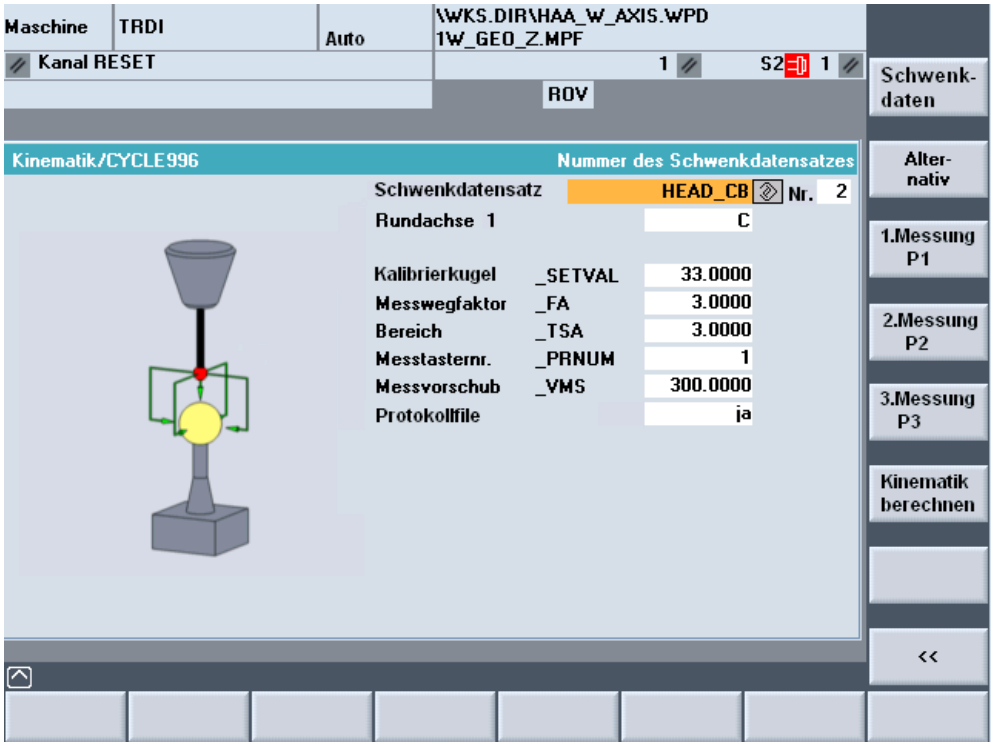
   Kinematics with rotary axes - with and without measuring system (manual rotary axes) - can be measured. The measurement sequence can be randomly selected - either rotary axis 1 or rotary axis 2. If the machine kinematics only has one rotary axis, then this is measured as rotary axis 1. During the course of the measurement, no 5-axis transformation (TCARR or TRAORI) is active, i.e. the machine kinematics is measured without active transformation.

   The precondition is a calibrated 3D probe in the spindle and a calibration ball that is measured with different - and the largest possible angular positions of the rotary axis. The machine must, at the start of the measuring operation, be mechanically measured as 3-axis machining including spindle (spindle radial eccentricity).Before calling measuring cycle CYCLE996 in the NC program, linear and rotary axes must be pre-positioned at start positions P1 to P6.

The basis data of the kinematics are always the data of the tool carrier TCARR that can be orientated. As support, when performing the 1st measurement, the swivel data set can be displayed - just like the start-up menu CYCLE800 - and if required, changed. If a dynamic 5-axis transformation (TRAORI) is to be supported, then transformation type 72 should preferably be used (vectors from TCARR data).
After the 3rd measurement of both rotary axes and the CYCLE996 setting "Calculate kinematics", the measurement results are written to result parameters. Further, a protocol (report) file with the measurement results can be output.

The complete measurement and calculation of the vectors of a machine kinematics with two rotary axes comprises seven CYCLE996 calls. Between the cycle calls, the user must re-position the rotary axis and linear axes to be measured. During the measurements, it is not permissible to position the rotary axis to be measured. The probe must be able to reach the "equator of the ball" of the calibration ball.
The 1st measurement must be made in the initial state of the kinematics. After the last measurement, the calculation of the machine kinematic vectors is realized by calling "Calculate kinematics".

The cycle can be programmed using the measuring cycle support; the input screen form is located under "Measure milling" / "Measure workpiece" / ">>" / "Kinematics".



For commissioning:
The user interface for the cycle is only displayed if, in the commissioning (start-up) screen form for measuring cycles, Automatic has been set in the input field "Kinematics/CYCLE996" _MZ_MASK[8] = 1.

Option:
The "Measure kinematics" function requires option P18.

New functions from SW 07.05.04.00 onwards:

2. New adjustable parameters for CYCLE996:
The following parameters should be programmed in the main program for "Measure kinematics" with CYCLE996.
_CHBIT[25]
0 = Only enter calculated offset vectors if the operator has acknowledged M0 with NC start.
1 = Immediately enter calculated offset vectors (no M0)
- for calculate kinematics and measuring version "enter measurement result".
- only if the tolerance of the offset vectors is not exceeded when making the calculation.

_CHBIT[26]
0 = Measure the calibration ball in parallel with the axis
1 = Measure the calibration ball at an angle
With this version it is possible, e.g. for 90 degree positions of the calibration ball to measure the kinematics without the shaft of the calibration ball mechanically obstructing the measurement operation.
The start angle parameter _STA1 must be externally specified in the main program.
Value range of the start angle _STA1, 0 to 360 degrees.
The pre-switching angle _INCA is equal to 90 degrees.
Parameter _SPEED[1] is used as feed on the circular path.
Also refer to the programming manual, measuring cycles: Measuring ball CYCLE997

_TNVL: Limit angle, calculating triangular distortion for kinematics [degrees].
When parameter _TNVL is entered, the distortion of the triangle of the measurements is monitored. Practical values for parameter _TNVL are 20 to 45 degrees.
If the distortion of the triangle lies outside parameter _TNVL, then error 61430 "Kinematic vectors not computed -> Error code: %4" is displayed.

Programming example:
```
_CHBIT[25]=0 _CHBIT[26]=1 _STA1=45 _TNVL=20
…
CYCLE996
```

Measure kinematics with active 5-axis transformation (TRAORI):
The vectors of the 5-axis transformation must have been roughly set as precondition for measuring the kinematics with active TRAORI.
With measuring cycle release 07.05.04.00, it is possible to approach the measuring positions to measure the kinematics with TRAORI active For the actual measurement of the calibration ball, CYCLE996 disables the transformation and then subsequently enables it again.

New functions from SW 07.05.05.00 onwards:

3. New parameter, start angle for workpiece measurement with CYCLE977:
From this version, parameter _STA1 to input angular values is available for CYCLE977! This angle that is entered represents, for the measuring versions hole and lug, a start angle offset - and is functionally identical to the parameter "Probing angle" when making measurements in JOG.
Referred to the measurement of the grooves and ribs rotated around the feed axis, using _STA1, the measurement can be adapted so that the workpiece probe probes the measuring location at right angles.

4. Extensions to use mono probes for workpiece measurements, corner, angle and plane – CYCLE961 and CYCLE998:
Now, for workpiece measurement involving corners, angles and planes and for align edge, the mono probe can be used without any restrictions - both in JOG as well as in the automatic mode.
As a consequence, the user is responsible for the correct use of a mono probe corresponding to the manufacturer's data. Especially when measuring angles, transverse stress and the deflection direction of the probe must be carefully taken into consideration.

New functions from SW 07.05.06.00 onwards:

5. New programmable adjustable parameters for CYCLE996
_CHBIT[28] **Measuring version CYCLE997 for orbits of the ball, for measurements at an angle**
(_CHBIT[26]=1), track (follow-up) spindle
0 = For ball orbits, do not track spindle
1 = For ball orbits, track spindle
(this can be used for table and mixed kinematics)
_CHBIT[29] **Basis, normalization of the orientation vectors**
0 = Normalization based on the calculated orientation vectors (V1xyz, V2xyz)
1= Normalization based on the input values of the tool carrier (TCARR) that can be orientated - of the orientation vectors (V1xyz, V2xyz)

Documentation

The current documentation for this SW release is as follows:

"Measuring Cycles Programming Manual" Edition 01.2008
MLFB 6FC5398-4BP20-1AA0

Compatibility: Prerequisite:

Software HMI:	
HMI-Advanced	from SW 7.5
Software NCU:	
Software 840D sl NCU 7x0.x[2]	from SW 1.5
for measure kinematics CYCLE996	NCK from 72.05.01
with HMI-Embedded	from SW 7.5
with ShopMill-HMI	from SW 7.5
with ShopTurn-HMI	from SW 7.5
Software NCU:	
Software 840Di sl NCU	from SW 1.5
for measure kinematics CYCLE996	NCK from 72.05.01
with HMI-Advanced	from SW 7.5
with ShopMill-HMI	from SW 7.5
with ShopTurn-HMI	from SW 7.5

Retrofit regulations:

[2] For applications with SW from 2.5, please note Attachment_3!

(1) with HMI-Advanced 7.5:
- read-in the archive under "Archive" / "Cycles archive" / "Measuring cycles"
- re-activate the data blocks GUD5 and GUD6 under "Definitions" (if required, also GUD7, if measuring in JOG is used)
- activate the softkeys for measuring cycle support and measuring in JOG in the com files aeditor.com, startup.com and ma_jog.com
- the user interface must be activated to use the CYCLE996 cycle;
(2) with NCU 840D sl: refer to Attachment_1

Errors that have been resolved (debugs): The following errors (bugs) have been resolved and in so doing, the following files modified:

Debugs up to SW 07.05.01.00:

ARTS+ (RQ-id)	File
AP00447608	**CYCLE982**
AP00503444	**Measuring cycle support**
AP00504071	**Measuring in JOG**
AP00507301	**CYCLE982 measurement result screen**
AP00514963	**CYCLE979**
AP00531680	**CYCLE109 Measuring in JOG**

Debugs up to SW 07.05.03.00:

AP00447541	**CYCLE997**
AP00577260	**Measuring cycle support**
AP00591948	**Measuring cycle support**
AP00592986	**CYCLE996**
AP00603309	**CYCLE978**
AP00606093	**Measuring cycle support**
AP00626470	**CYCLE978**
AP00626486	**CYCLE998**
AP00629687	**CYCLE996**
AP00835584	**CYCLE115 CYCLE961**
AP00565321	**CYCLE996**
AP00612233	**CYCLE996**
AP00662737	**Measuring in JOG**

AP00662862	**Measuring in JOG**
AP00700608	**CYCLE982**
AP00742308	**CYCLE996**
AP00750493	**CYCLE996**
AP00757891	**E_MT_LEN** **E_MT_RAD** **E_MS_CAN**
AP00760648	**Measuring cycle support**
AP00764933	**CYCLE997**
AP00765106	**Measuring cycle support**
AP00765237	**Measuring cycle support**
AP00768418	**Measuring cycle support**
AP00774313	**Measuring cycle support**
AP00793932	**Measuring cycle support**
AP00799674	**CYCLE996**
AP00833672	**Measuring cycle support**
AP00834801	**Measuring cycle support**
AP00843271	**CYCLE977**
AP00843280	**Measuring cycles, general**
AP00853882	**CYCLE976**
AP00853889	**E_MS_CAN**
AP00858539	**Measuring cycle support**
AP00868322	**CYCLE996**
AP00330739	**Measuring in JOG**
AP00364957	**CYCLE971**
AP00384059	**CYCLE982**
AP00387424	**Measuring cycle support**
AP00439384	**CYCLE998**
AP00591402	**Measuring in JOG**
AP00750996	**CYCLE977**
AP00764261	**CYCLE997**
AP00769661	**Measuring cycles**
AP00832833	**Measuring in JOG**
AP00843289	**Measuring cycles**
AP00853987	**CYCLE997**
AP00874192	**CYCLE996**
AP00888241	**CYCLE998**

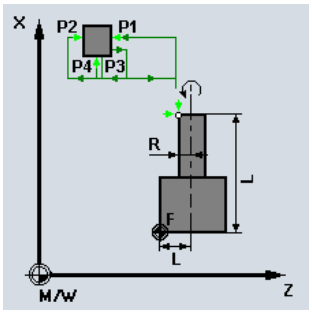
AP00892392	**Measuring cycle support**
AP00895421	
AP00904965	**Measuring in JOG**
AP00914985	**CYCLE997**
AP00916107	**Measuring cycles under ShopMill**
AP00947323	**Measuring cycle support**
AP00952221	**CYCLE998**
AP00955096	**Measuring in JOG under ShopMill**
AP00956038	**CYCLE998**
AP00956554	**Measuring cycle support**
AP00973263	**Measuring in JOG**
AP00985764	**CYCLE998**
	Measuring in JOG

Secondary conditions:

(1) Restrictions regarding measuring cycle support:
a) The step editor does not recognize measuring cycle calls as steps and are correspondingly displayed.
b) Measuring cycle calls, which are in the NC program without comment lines (with #), cannot be re-compiled in the screen forms of the measuring cycle support.

(2) Restriction regarding the use of "Measuring in JOG" with 840D sl:
With 840D sl, the "Measuring in JOG" function can only be used with a PCU 50.

(3) Restriction when measuring tools with CYCLE982:
With the measuring version _MVAR=10xx2 and the tool position shown in the diagram, it is not possible to measure the milling tool.



(4) Restriction regarding measuring cycle support – CYCLE982:
The distinction between the calibration tool or turning tool when calibrating (8th position in parameter _MVAR) cannot be programmed using the measuring cycle support, value 0 for the calibration tool is always generated.

(5) Restriction when measuring multi-axis kinematics - CYCLE996 (due to AP00382720):
Measuring the kinematics is only possible, if in the initial state of the kinematics, there is no offset in the rotary axes, e.g. the rotary axis position = 0.
This involves the parameter:
$TC_CARR24[n],$TC_CARR25[n], with n=number, swivel data set

(6) Restriction when measuring outer contours (shaft, rib, outer rectangle/square)
Due to RQ AP00447656, parameter _CHBIT[13] (spindle coupling at the coordinate rotation in the plane) must be set to zero.

Notes on using this measuring cycle SW release:

Note when measuring tools with CYCLE982 (from SW 7.3):
Measuring turning, milling and drilling tools on lathes with tool carrier that can be orientated (B or C axis) has been implemented for the 90° multiples of the tool positions.

Notes on measuring circle segments with CYCLE979:
(due to AP00348997)
When measuring circle segments < 90 degrees, it should be noted that due to the mathematics, measuring points that deviate from the circle have significant influence on the accuracy of the results (center point, diameter)! This is the reason that when measuring small circle segments, the measurement must be performed with a high degree of precision.

Good results can be achieved by applying the following measures:
The circle segment to be measured should ...
- be free of remains of previous machining operations - burrs, etc.
- the production technology used should guarantee a circular shape that is as precise as possible, and which should have the lowest possible surface roughness!
- be measured with high quality probes, which means that the probe balls must have a spherical shape that is as homogeneous as possible.
- be measured using the 4-point measurement version (_PRNUM=1xxx).
- be measured using an up-to-date calibrated probe.

Information on measuring tools with CYCLE982 in conjunction with CUTMOD:
The command CUTMOD = -2 is internally programmed in the CYCLE982 measuring cycle. This is kept after the cycle has been called; this means that the value that was programmed before the cycle call is no longer valid.

Note on CYCLE998 or on the function "Align edge" in the JOG mode from MC 07.05.06.00 onwards
Alarm 61189, "Invalid rotary axis positions" has, in conjunction with measuring cycles, the following significance from version
MC 07.05.06.00 onwards:
Alarm 61189 can be output for the "1-angle measurement with CYCLE998" in the automatic mode and for the function "Align edge" when measuring in JOG, together with the rotary axis correction target!
When this alarm is output, no correction is made to the zero offset of the selected rotary axis - the measurement program must be cancelled with "RESET"!

Cause:
The measured physical angle does not lie in the working plane or is not vertical to this. As a consequence, the result of the 1-angular measurement is sub-divided into several result components, which cannot be corrected in a rotary axis with only one degree of freedom (e.g. rotary axis C rotates around Z)!

Remedy:
Correction target, instead of "Correction in the rotary axis", select -> "Correction in the coordination rotation".

Note when measuring in the JOG mode:
Modified behavior when calibrating the workpiece probe (radius):
New adjustable parameter to transfer the determined radius value into the tool correction parameter radius of the active workpiece probe when measuring in JOG:
_CBIT[15]=0: No transfer (standard value)
_CBIT[15]=1 The determined radius value is transferred
Note: _CBIT[15] is effective unchanged - also in the automatic mode (CYCLE976)!

Commissioning measuring cycles for SINUMERIK 840D sl, SW 1.x

Preconditions
- The software release of the measuring cycles, which are to be commissioned, must be on the CF card. Otherwise, no additional software tools are required for commissioning.
- Before starting commissioning, it must be carefully ensured that the existing state is archived; if required, read-out the series commissioning archive.

Step 1 - Machine data settings

The following machine data apply when using Siemens standard and measuring cycle packages.

(1) Machine data (memory-configuring) for Global User Data (GUDs):
MD 18130: MM_NUM_GUD_NAMES_CHAN=130

(2) Machine data (memory configuring) for the number of cycles and number of transfer parameters
MD 18180: MM_NUM_MAX_FUNC_PARAM=800

(3) Additional machine data for measuring cycles
MD 11420: LEN_PROTOCOL_FILE=5
(maximum protocol file size is set in K byte)
MD 28082: MM_SYSTEM_FRAME_MASK Bit0 and Bit5 =1
(system frames for scratching and cycles)

Step 2 - read-in the archive

Read-in the required archive files:

cyccust_gr.arc	;*Archive for user cycles, German
cyccust_uk.arc	;*Archive for user cycles, English
mcycles.arc	;*Archive with all measuring cycles (milling and drilling)
mcycmill.arc	;*Archive with all measuring cycles, milling
mcycturn.arc	;*Archive with all measuring cycles, turning
defines.arc	;*Archive with definition files (GUD5.DEF, GUD6.DEF, GUD7_MC.DEF)
jogmcyc.arc	;*Archive with all measuring cycles for measuring in jog

In the "Services" area, "Program data" softkey, change to the "Cycle archive" using the cursor keys and open using the Input key. In the area "cycle archive" change to "Measuring cycles" using the cursor keys and open using the input key.
Read-in the archive files required using the first vertical softkey "Read-in archive".
It must be noted that the archive files must be individually selected and read-in. After all of the required archive files have been read-in, an NCK reset must be made via the "Start up" operating area, "NCK reset" softkey.

Step 3 - Activate the definition files

In the "Program" operating area, change into the 3rd plane using the etc. key ">". There, press the softkey "definition files", then individually select the definitions GUD5.DEF, GUD6.DEF and GUD7.DEF and press the "Activate" softkey.

Step 4 - Activate the entry softkeys "Measuring, turning" and "Measuring, milling" for the measuring cycle support

In the operating area "Program", "Standard cycles", open the file "common.com" using the Enter key and in the line

;SC326 = AEDITOR.COM	;* SK "Measuring, turning"
;SC327 = AEDITOR.COM	;* SK "Measuring, milling"

delete the semicolon. Then, close the file using the "Close editor" softkey.
After this, the "NCK reset" softkey should be pressed from the "Start up" operating area.

Step 5 – Activate the softkey "Measuring cycles" in the "Start up" area

In the operating area "Program", "Standard cycles", open the file "common.com" using the Enter key and in the line

;SC617 = STARTUP.COM

delete the semicolon. Then, close the file using the "Close editor" softkey.
After this, the "NCK reset" softkey should be pressed from the "Start up" operating area.

Using the measuring cycles SW 7.5 in HMI-Advanced from 7.5 and higher with SINUMERIK 840D sl from SW 2.5

The configuration HMI-Adv SW 7.5 with NCU 7x0.x SW 2.5 contains two different cycle software releases:

- The total package NCU 7x0.x comprises: NCU2.5 (with HMIsl SW 2.5) with cycle packages SW 2.5 (technological cycles, measuring cycles) on a CF card, these match the cycle support in the editor of this internal HMI;
- HMI-Adv 7.5 also brings the cycle archive SW 7.5 (technological cycles, measuring cycles) into the archive path.

With this total configuration, then the following always applies: Cycles that are downloaded into the NCU can only be operated with the cycle support that belongs to this SW release - but not mixed.

To use the cycle support of the HMI-Advanced from 7.5 onwards, the cycles should be loaded from its archives - but not the cycles from the CF card of the NCU.

11. Notes on the cycles (ISO cycles) for the "Online ISO Dialect Interpreter":
CNC software 1.5 SP3 includes the ISO cycles V06.05.04 in an installable form.

Installing the ISO cycles:
Installation of the ISO cycles requires previous commissioning of the standard cycles. Before you begin the startup process, you should ensure that the existing status is saved to an archive; read out a series startup archive if necessary.

Documentation:	ISO dialects for SINUMERIK	6FC5297-6AE10-0APx
	Programming Manual ISO Milling	6FC5298-6AC20-0BPx
	Programming Manual ISO Turning	6FC5298-6CA00-0BGx

Step 1 - Machine data settings:
You can activate the Online ISO Dialect Interpreter using the machine data 18800 $MN_EXTERN_LANGUAGE. The language types ISO dialect M (Milling) or T (Turning) can be selected via machine data 10880 $MN_EXTERN_CNC_SYSTEM.

Step 2 - Reading in the archives

isomill.arc	;*Archive with all ISO cycles for milling
isoturn.arc	;*Archive with all ISO cycles for turning
defines.arc	;*Archive with definition file (GUD7_ISO.DEF)

For the procedure for reading in the archive files, please refer to: "Information about the technology cycles"

Step 3 - Activating the definition files
Activate the GUD7.DEF definition file. For the procedure for activating the definitions, please refer to: "Information about the technology cycles"

List of G codes with the corresponding envelope cycles and technology cycles:

Milling:

G Code	Envelope Cycle	Siemens Technology Cycles
G05	CYCLE305	
G08	CYCLE308	
G10.6	CYCLE3106	
G22	CYCLE322	
G23	CYCLE323	
G27	CYCLE328	
G28	CYCLE328	
G30	CYCLE330	
	CYCLE396	
G71.1 / 72.2	CYCLE3721	
G81	CYCLE381M	CYCLE82
G82	CYCLE381M	CYCLE82
G85	CYCLE381M	CYCLE85
G86	CYCLE381M	CYCLE88
G89	CYCLE381M	CYCLE85
G83	CYCLE383M	CYCLE83
G73	CYCLE383M	CYCLE83
G84	CYCLE384M	CYCLE3841
G74	CYCLE384M	CYCLE3841
G76	CYCLE387M	CYCLE86
G87	CYCLE387M	CYCLE861

Turning:

G Codes A/B/C	Envelope Cycle	Siemens Technology Cycles
G05	CYCLE305	
G10.6	CYCLE3106	
G22	CYCLE322	
G23	CYCLE323	
G27	CYCLE328	
G28	CYCLE328	
G30	CYCLE330	
	CYCLE396	
G50.2 / 51.2	CYCLE3512	
G70 / 70 / 72	CYCLE370T	
G71 / 71 / 73	CYCLE371T	CYCLE395
G90 / 77 / 20	CYCLE371T	
G72 / 72 / 74	CYCLE372T	CYCLE395
G94 / 79 / 24	CYCLE372T	
G73 / 73 / 75	CYCLE373T	
G74 / 74 / 76	CYCLE374T	CYCLE375T
G75 / 75 / 77	CYCLE374T	CYCLE375T
G76 / 76 / 78	CYCLE376T	CYCLE398
G92 / 78 / 21	CYCLE376T	CYCLE398
G83	CYCLE383T	CYCLE375T
G87	CYCLE383T	CYCLE375T
G84	CYCLE384T	CYCLE84
G88	CYCLE384T	CYCLE84
G85	CYCLE385T	CYCLE375T
G89	CYCLE385T	CYCLE375T

12. Information about ShopTurn:

General information:

Upgrade in the metric unit system only!
The ”Work step programming” option is required to create ShopTurn programs.
The tool type "Limit stop" has been assigned number 730. Number 750 was used for the "Limit stop" tool in ShopTurn before. The change represents an adaptation to HMI Advanced.
When setting the channel MD 20108 PROG_EVENT_MASK not equal to 0, you must ensure via a setting in the MD 20192 PROG_EVENT_IGN_PROG_STATE that the OPI variables progStatus and chanStatus are not influenced by the PROG_EVENT.
Channel MD 20192 PROG_EVENT_IGN_PROG_STATE:
The behavior of event-triggered program calls (Prog-Events) can be influenced at the OPI interface.
In this case, the variables progStatus and chanStatus are not influenced, despite active Prog-Event processing.
Bit 0 = 1 : Ineffective reserved bit
Bit 1 = 1 : Prog-Event after part program end does not affect progStatus and chanStatus
Bit 2 = 1 : Prog-Event after machine control panel reset does not affect progStatus and chanStatus.
Bit 3 = 1 : Prog-Event after runup does not affect progStatus and chanStatus.
Bit 4 = 1 : Reserved.
Bit 5 = 1 : Safety-Prog event during runup does not affect progStatus and chanStatus.

Machine data for ShopTurn:
An overview is provided under the Tools zero operating area => Extended softkey bar => Machine data softkey.

N10134 $MN_MM_NUM_MMC_UNITS=6 ;minimum
N10260 $MN_CONVERT_SCALING_SYSTEM=1 ;exact
N10602 $MN_FRAME_GEOAX_CHANGE_MODE=1 ;exact
N10610 $MN_MIRROR_REF_AX=0 ;exact
N10700 $MN_PREPROCESSING_LEVEL='H25' ;exact: bit0=1, bit2=1, bit5=1
N10702 $MN_IGNORE_SINGLEBLOCK_MASK='H10' ;exact: bit4=1
N10714 $MN_M_NO_FCT_EOP=32 ;variable
N10722 $MN_AXCHANGE_MASK='H4' ;exact: bit2=1
N10735 $MN_JOG_MODE_MASK='H1' ;exact: bit0=1
N11270 $MN_DEFAULT_VALUES_MEM_MASK='H1' ;exact: bit0=1
N11346 $MN_HANDWH_TRUE_DISTANCE=2 ;variable
N11410 $MN_SUPPRESS_ALARM_MASK='H1007' ;exact: bit0=1, bit1=1, bit2=1, bit12=1
N11415 $MN_SUPPRESS_ALARM_MASK_2='H20' ;exact: bit5=1
N11450 $MN_SEARCH_RUN_MODE='H7' ;exact: bit0=1, bit1=1, bit2=1
N11470 $MN_REPOS_MODE_MASK='H8' ;exact: bit3=1
N11604 $MN_ASUP_START_PRIO_LEVEL=1 ;minimum
N17530 $MN_TOOL_DATA_CHANGE_COUNTER='HF' ;exact: bit0=1, bit1=1, bit2=1, bit3=1
N18080 $MN_MM_TOOL_MANAGEMENT_MASK='HB' ;exact: bit0=1, bit1=1, bit3=1
N18118 $MN_MM_NUM_GUD_MODULES=7 ;minimum
N18120 $MN_MM_NUM_GUD_NAMES_NCK=20 ;minimum
N18130 $MN_MM_NUM_GUD_NAMES_CHAN=300 ;minimum
N18150 $MN_MM_GUD_VALUES_MEM=120 ;minimum
N18160 $MN_MM_NUM_USER_MACROS=30 ;minimum
N18170 $MN_MM_NUM_MAX_FUNC_NAMES=200 ;minimum
N18180 $MN_MM_NUM_MAX_FUNC_PARAM=3000 ;minimum
N18280 $MN_MM_NUM_FILES_PER_DIR=400 ;minimum
N18320 $MN_MM_NUM_FILES_IN_FILESYSTEM=400 ;minimum
N20090 $MC_SPIND_DEF_MASTER_SPIND=? ;exact ?= Main spindle number from MD35000
N20110 $MC_RESET_MODE_MASK='H04041' ;exact: bit0=1, bit6=1, bit14=1, bit16=0
N20112 $MC_START_MODE_MASK='H400' ;exact: bit10=1
N20128 $MC_COLLECT_TOOL_CHANGE=0 ;exact
N20130 $MC_CUTTING_EDGE_RESET_VALUE=1 ;exact
N20150 $MC_GCODE_RESET_VALUES[5]=2 ;exact

```
N20150 $MC_GCODE_RESET_VALUES[15]=3                ;variable
N20150 $MC_GCODE_RESET_VALUES[48]=3                ;variable
N20152 $MC_GCODE_RESET_MODE[5]=0                   ;exact
N20152 $MC_GCODE_RESET_MODE[7]=1                   ;exact
N20152 $MC_GCODE_RESET_MODE[12]=0                  ;exact
N20152 $MC_GCODE_RESET_MODE[14]=1                  ;exact
N20152 $MC_GCODE_RESET_MODE[15]=0                  ;exact
N20202 $MC_WAB_MAXNUM_DUMMY_BLOCKS=10              ;minimum
N20240 $MC_CUTCOM_MAXNUM_CHECK_BLOCKS=4            ;exact
N20250 $MC_CUTCOM_MAXNUM_DUMMY_BLOCKS=5            ;exact
N20310 $MC_TOOL_MANAGEMENT_MASK='H80400B'          ;exact: bit0, 1, 3, 14, 16, 23=1
N20320 $MC_TOOL_TIME_MONITOR_MASK='H1'             ;exact: bit0=1
N20734 $MC_EXTERN_FUNCTION_MASK='H8'               ;exact: bit3=1
N24006 $MC_CHSFRAME_RESET_MASK='H1'                ;exact: bit0=1, bit5=0
N24007 $MC_CHSFRAME_RESET_CLEAR_MASK='H20'         ;exact: bit0=0, bit5=1
N24008 $MC_CHSFRAME_POWERON_MASK='H0'              ;exact: bit0=0, bit5=0
N24040 $MC_FRAME_ADAPT_MODE='H7'                   ;exact: bit0=1, bit1=1, bit2=1
N24805 $MC_TRACYL_ROT_AX_FRAME_1=2                 ;exact
N24855 $MC_TRACYL_ROT_AX_FRAME_2=2                 ;exact *1)
N24905 $MC_TRANSMIT_ROT_AX_FRAME_1=2               ;exact
N24911 $MC_TRANSMIT_POLE_SIDE_FIX_1=1              ;exact
N24955 $MC_TRANSMIT_ROT_AX_FRAME_2=2               ;exact *1)
N24961 $MC_TRANSMIT_POLE_SIDE_FIX_2=1              ;exact *1)
N27100 $MC_ABSBLOCK_FUNCTION_MASK='H1'             ;exact: bit0=1
N27860 $MC_PROCESSTIMER_MODE='H33'                 ;exact: bit0=1, bit1=1, bit4=1, bit5=1
N27880 $MC_PART_COUNTER='H101'                     ;variable
N28000 $MC_MM_REORG_LOG_FILE_MEM=75                ;minimum
N28010 $MC_MM_NUM_REORG_LUD_MODULES=20             ;minimum
N28020 $MC_MM_NUM_LUD_NAMES_TOTAL=1000             ;minimum
N28040 $MC_MM_LUD_VALUES_MEM=200                   ;minimum
N28082 $MC_MM_SYSTEM_FRAME_MASK='H61'              ;exact: bit0=1, bit5=1, bit6=1
N28083 $MC_MM_SYSTEM_DATAFRAME_MASK='H1F'          ;exact: bit0, 1, 2, 3, 4=1, bit5=0, bit6=0
N28400 $MC_MM_ABSBLOCK=1                           ;exact
N28402 $MC_MM_ABSBLOCK_BUFFER_CONF[0]=2            ;exact
N28402 $MC_MM_ABSBLOCK_BUFFER_CONF[1]=4            ;exact
N28450 $MC_MM_TOOL_DATA_CHG_BUFF_SIZE=80           ;minimum
N28560 $MC_MM_SEARCH_RUN_RESTORE_MODE='H1'         ;exact: bit0=1
N32084 $MA_HANDWH_STOP_COND[?]=0xFF                ;Variable for X, Z and - as necessary - for Y
N35035 $MA_SPIND_FUNCTION_MASK[?]='H110'           ;exact: bit10=0, ?= Axis index Main spindle
N35035 $MA_SPIND_FUNCTION_MASK[?]='H110'           ;exact: bit10=0, ?= Axis index Counterspindle
*1)
N35035 $MA_SPIND_FUNCTION_MASK[?]='H110'           ;exact: bit10=0, ?= Axis index tool spindle *2)
N35040 $MA_SPIND_ACTIVE_AFTER_RESET[?]=2           ;exact, ?= Axis index for main spindle
N35040 $MA_SPIND_ACTIVE_AFTER_RESET[?]=2           ;exact, ?= Axis index for counterspindle *1)
N35040 $MA_SPIND_ACTIVE_AFTER_RESET[?]=2           ;exact, ?= Axis index for tool spindle *2)
N37050 $MA_FIXED_STOP_ALARM_MASK[?]=2              ;exact, ?= Axis index for 4th axis *1)
N41100 $SN_JOG_REV_IS_ACTIVE=0x0E                  ;variable *3)
N42440 $SC_FRAME_OFFSET_INCR_PROG=0                ;exact
N42442 $SC_TOOL_OFFSET_INCR_PROG=0                 ;exact
N42528 $SC_CUTCOM_DECEL_LIMIT=0.1                  ;variable
N42940 $SC_TOOL_LENGTH_CONST=18                    ;exact
N42950 $SC_TOOL_LENGTH_TYPE=2                      ;exact
N42600 $SC_JOG_FEED_PER_REV_SOURCE=0               ;exact: 0;-3 Default: 0 *3)
N43300 $SA_ASSIGN_FEED_PER_REV_SOURCE=0            ;exact: 0;-3 Default: 0 for X, Z and - as
necessary - for Y *3)
```

*1) with counterspindle only

*2) with tool spindle only
*3) with "Manual machine" otpion only

Recent additions to range of display machine data:

```
N9428 $MM_MA_SPIND_MAX_POWER=100                ;variable
N9429 $MM_MA_SPIND_POWER_RANGE=200              ;variable
N9718 $MM_CMM_OPTION_MASK_2='H200'              ;Mask out variable MD overview
N9729 $MM_CMM_G_CODE_TOOL_CHANGE_PROG=-         ;variable
N9777 $MM_CMM_ENABLE_TIME_DISPLAY='H7F'         ;variable
N9836 $MM_ST_TAILSTOCK_DIAM=0                   ;variable
N9837 $MM_ST_TAILSTOCK_LENGTH=0                 ;variable
N9838 $MM_ST_BORDER_TOOL_LEN_X_REV_2=0          ;variable, >0 only with 2 toolholders
N9897 $MM_ST_OPTION_MASK_MAN_FUNC=0             ;variable
N9899 $MM_ST_TRACE=0                            ;variable
```

When the software is upgraded, the display machine data are not initialized with defaults. For this reason, they must be checked according to the Description of Functions.

Machine data for a second toolholder on the X axis:

- A tool length where the tools start to be located on a 2nd turret is defined through a display machine data (0 = no 2nd turret available): 9838 $MM_ST_BORDER_TOOL_LEN_X_REV_2
- For a proper function of the TNRC in the mirrored coordinate system CUT2DF must always be selected: (is set automatically in F_HEAD)
  20150 $MC_GCODE_RESET_VALUES[21]=2 ; exact
  20152 $MC_GCODE_RESET_MODE[21]=0 ; exact
- The following MD must be set so that LIFTFAST (thread cutting retraction) also reacts to the mirror:
  21202 $MC_LIFTFAST_WITH_MIRROR = 1 ; exact
- WPFRAME is required for mirrors and offsets:
  24006 $MC_CHSFRAME_RESET_MASK Bit 4=1 (=’H11’)
  24007 $MC_CHSFRAME_RESET_CLEAR_MASK Bit 4=0 (=’H00’)
  28082 $MC_MM_SYSTEM_FRAME_MASK Bit 4=1 (=’H31’)
  28083 $MC_MM_SYSTEM_DATAFRAME_MASK Bit 4=1 (’H1F’)
- The tool wear must also be mirrored:
  42910 $SC_MIRROR_TOOL_WEAR = 1 ; exact

PLC interface:
In the OB100, during the FB1 call, the parameter MMCToIF = True (transfer of the MMC signals to the interface...operating types, program influence) must be set. If the parameter is not set to TRUE, no operating mode change commands (on program selection) or program controls (program test, dryrun...) can be implemented via the interface.

The manual tool function (e.g. for multifix holder) can be used under the following conditions:
N22550 TOOL_CHANGE_MODE = 0
N22562 TOOL_CHANGE_ERROR_MODE ='H2'
The interface signals in the PLC behave as follows:

- Changing from T0 to a tool from the turret and vice versa activates DB73
- Changing from T0 to a manual tool and vice versa activates DB72 with loading/unloading the manual tool
- Changing from a tool from the turret to a manual tool first activates DB73 with T0 and then DB72 with loading of manual tool
- Changing from a manual tool to a tool from the turret activates DB73 with unloading manual tool (DBX 4.6)

PLC sample modules are available in the tool box under \8x0d\ShopMill_Turn\ShopTurn\tm_rev_xy.awl.

==========

==========

Requirements:
If you wish to install ShopTurn on the PCU 50.3, the HMI Advanced software (see SINUMERIK 840D sl compatibility list for version) must already be installed on the PCU 50.3.
The internal HMI must be deactivated.

In addition, check the version of the cycle support. The cycle support files for the measuring cycles (MCSUPP.ARC) and standard cycles (SCSUPP.ARC) are stored under Archives\Cycles archives.

==========

Installation / software upgrade with USB on PCU 50.3

==========

Copy the contents to any directory of the USB hardware (stick; card reader; etc.).
Power up the PCU50 at the desktop.
Power on the PCU 50, press key "3" when the message with the PCU Base software version is displayed (e.g. V08.xx.xx.xx) and enter the password (e.g. SUNRISE).
Connect the USB hardware to the USB front connection of the OP, for example. After WIN-XP has recognized the hardware, copy the directory contents to D:\INSTALL.

Installing the software:
Power OFF/ON the PCU50. The software is installed automatically when the PCU software is powered up or when SETUP.EXE is called from D:\INSTALL directory.

Call SETUP.EXE during installation to delete the contents of D:\INSTALL prior to PCU 50 power OFF in order to avoid duplication of the installation after PCU 50 power OFF/ON.

==========

Installation / software upgrade on PCU 50.3 via network

==========

This installation method can be used only if the PCU 50 has an existing network link which allows data to be transferred directly to the PCU 50 hard disk.
Copy the contents to any directory of a networked PC / PG.
Power up the PCU50 at the desktop.
Power on the PCU 50, press key "3" when the message with the PCU Base software version is displayed (e.g. V08.xx.xx.xx) and enter the password (e.g. SUNRISE).

Then copy all files from the installation package via network to D:\INSTALL of the PCU 50.

Installing the software:
Power OFF/ON the PCU50. The software is installed automatically when the PCU software is powered up or when SETUP.EXE is called from D:\INSTALL directory.

Call SETUP.EXE during installation to delete the contents of D:\INSTALL prior to PCU 50 power OFF in order to avoid duplication of the installation after PCU 50 power OFF/ON.

How to proceed when upgrading the NCK:

Save GUD data
HMI Embedded: In the CNC ISO mode Services / Programs Data / NC active data / Global user data / All / Generate Archive.
HMI ADV: Services / Data Out / NC active data / user data

- Create NCK and PLC archives; to do this, switch PLC to stop and back up display machine data
- Install current NCU software (see first installation of NCU software described in these Upgrade Instructions).
- Read in NCK and PLC archives

- Adapt machine data, take note of alarm 4400 if it is displayed.
- Unload / delete definitions

- Cycles and definitions from ShopTurn with HMI on NCU:
  Read in the definition archives from the operating area Services\Cycle archive\Standard cycles\defines.arc, sequence cycles\st_def_e.arc, and measuring cycles\defines.arc and activate (SMAC.DEF, GUD5.DEF, GUD6.DEF and GUD7_MC.DEF).
  Read in the cycle archives from the operating area Services\Cycle Archive\Standard cycles.arc, sequence cycles\st_cyc_e.arc, measuring cycles\mcycles.arc.
- Cycles and definitions in ShopTurn with PCU 50.3:
  Read in the definition archive from the "Services\Archives\ST_DEF.ARC" operating area. Load the definitions (GUD7_JS.DEF, GUD7_MC.DEF, GUD7_SC.DEF, GUD7_ST.DEF, SMAC_SC.DEF and SMAC_ST.DEF) and activate the definitions SMAC.DEF, GUD5.DEF, GUD6.DEF and GUD7.DEF.
  Read in the cycle archive from the operating area "Services\Archives\ST_CYC.ARC" and load the cycles
  Loading the manufacturer and standard cycles directory

(NOTICE: Adapt the cycles ST_CUST.SPF(cyccust_gr.arc), CYCLE198.SPF and CYCLE199.SPF if necessary after consultation with the machine manufacturer).
- Read in the GUD data
- Perform a general NCK reset

The "Prog_Event" system cycle is used by the standard cycles and by ShopTurn. If "Prog_Event" is also required to implement user functions, it must be ended by means of the M function from machine data N10714 $MN_M_NO_FCT_EOP. If you create a cycle with the name CYCPE1US.SPF or CYCPE1MA.SPF, it will be called before the Siemens calls in Prog_Event. If you create a cycle with the name CYCPE_US.SPF or CYCPE_MA.SPF, it will be called after the Siemens calls. These cycles can be stored under User cycles or OEM cycles, as "Prog_Event" is overwritten in the OEM cycle directory during upgrading.

Debugs, improvements in ShopTurn 1.5 SP3 with respect to 1.5 SP2
==========
AP00403563: contour violation when milling spigots (2 spigots)
AP00451202: OMPOCKET(11) - contour violation when milling spigots
AP00500620: OMPOCKET(12) - contour violation (islands) for base finishing
AP00479958: OMPOCKET(13) - contour violation for contour pocket, solid machining
AP00768755: OMPOCKET(14) - the contour is violated for spigot_milling when retracting
AP00775768: OMPOCKET(16) - contour violation for "spigot milling"
AP00355861: OMPOCKET(21) - memory no longer available
AP00355863: OMPOCKET(22) - TCU cannot be operated for miller radius <=3.8MM.
AP00355795: OMPOCKET(26) - message, residual material not completely calculated
AP00355820: OMPOCKET(27) - message, residual material not completely calculated
AP00355805: OMPOCKET(38) - no solid machining
AP00406154: OMPOCKET(40) - Alarm 112052 "No residual material available" although residual material is available
AP00523830: OMPOCKET(49) - contour with self cut
AP00824199: OMPOCKET(51) - message, no path to generate
AP00824775: OMPOCKET(56) - message, no path to generate
AP00871689: OMPOCKET(63) - islands are milled away
AP00874817: OMPOCKET(65) - message, no path to generate
AP00738364: OMPOCKET(67) - instead of an error there is only an alarm if residual corners are kept
AP00890912: OMPOCKET(69) - contour violation when finishing islands and for helical plunging.
AP00913012: OMPOCKET(80) - SM message: "Memory no longer available" for program check
AP00916011: OMPOCKET(82) - the FM "No path to generate" when finishing the edge on the counterspindle.
AP00916019: DIAMCYCOF X-position display changes from diameter to radius when approaching

AP00869415: TURNCUT(01) - ST: System error 103 or 105 for contour turning in plunge turning
AP00849072: TURNCUT(05) - CYCLE952 moves, for plunge turning from the 4th infeed with rapid traverse and violates the contour.
AP00839114: TURNCUT(06) - CYCLE952/Alarm 61714 "System error contour turning- 103"
AP00775193: TURNCUT(18) - ST: HMI crash as a result of the contour generator
AP00770652: TURNCUT(20) - contour stock removal: Contour violation when approaching the contour with limited machining area
AP00865326: TURNCUT(21) - Error message - error for contour calculation if the radius is at starting point 1
AP00871226: TURNCUT(22) - ST contour violation with program example (turncut)
AP00879181: TURNCUT(25) - Bluescreen if finishing tolerance UX is equal to 0.4
AP00899728: TURNCUT(27) - ST single block SBL1 not active after stock removal, roughing
AP00904143: TURNCUT(31) - contour violation when approaching
AP00916014: TURNCUT(32) - unnecessary retraction motion in X for stock removal along the contour with relief cut
AP00916205: TURNCUT(33) - ST retraction error for stock removal, face, finishing, relief cut no
AP00916007: TURNCUT(34) - Alarm 10754 with program example
AP00923986: TURNCUT(44) - floating point calculation error - HMI-PR, CL1950, when machining contour, plunge turning, Alarm 1019 is output
AP00908990: Incorrect speed after block search. Collected speed is only output if the new tool is the same as the previous tool (F_S_ASUP)
AP00931751: OMPOCKET(85) - part program with 16 islands: Calculation not made
AP00893053: Excessively high speed for GSP coupling (F_SUB_SP)
AP00893535: Contour programming: Element-related feed and cutting edge selection
AP00944621: TURNCUT(51) - error message "Machining not possible" for program selection
AP00890940: Retraction missing in UP -> _E_JS_PRG. (F_END, F_HEAD adapted)
AP00904899: Screen freezes when renumbering and search replace on (from basis 07.50.22.00)
AP00916019: X-position display changes from diameter to radius when approaching the retraction plane
AP00893535: Contour programming: Element-related feed and cutting edge selection
AP00947363: For inch, turning tool measurement does not function
AP00893792: For contour path milling, face Y, for CP, the incorrect cursor text is used
AP00917073: ST face machining, with mirroring X, the X axis moves to the center of rotation.
AP00766006: No material removed for 2nd simultaneous recording if drilling centered was active.
AP00944094: ST G code program: for block search after TRANSMIT, TRANSMIT becomes inactive.
AP00916018: C-axis position is not displayed in the simulation (tailstock with NC axis without counterspindle).
AP00918957: Simulation: For grooves on the peripheral surface, only the plunge points are displayed (only with B axis).
AP00890870: Simulation display incomplete for a circular pocket milling for side milling (last infeed).
AP00936963: ST simulation G code program: Velocity increase in the volume model not variable.
AP00903334: Peripheral machining results in mirroring at the counterspindle (this is not an error).
AP00916963: For thread cutting, in the simulation, the tool moves through the unmachined part.
AP00622527: ST_SL simulation thread noncuts >=3 error (as for RQ-AP00916963).
AP00907109: Simulation for widescreen: At the 1st zoom, chuck jaws are not displayed.
AP00871064: Offset on counterspindle slides does not function, for start with rear side.
AP00884123: At the end of CYCLE72, OFFN=0 -> f_mi_con.que missing.
AP00934359: ST_SL programming error: Retraction through the workpiece after inside machining
AP00931627: Error for tool length calculation at the face (_TM1 always 0). drilling cycles: F_DR, F_DR_PEC, F_DR_REA, F_DR_SIN, F_DR_TAP
AP00919596: For undercut, form E (F_UCUT_E) with B-axis rotation, the incorrect cutting edge position is determined
AP00907075: Error when called twice from TSM (F_TFS)
AP00890947: Tool change, only duplo number (cycle: F_TFS)
AP00958426: TSM screen form under JOG: Counterspindle does not function in m/m
AP00948846: Various crashes when reading tool offset values (from basis 07.50.22.00)
AP00965658: When machining at face Y, incorrect sequence of the CP positioning.

AP00939206: Contents of cmm_single.mpf are not deleted

Debugs, improvements in ShopMill 1.5 SP3 HF1 with respect to 1.5 SP3
==========
AP00890809: Swiveling: Retraction with tool change depending on the program structure
AP00984846: Modified contours do initially not become active when being called up again in the program
AP00999861: Generator/interpreter problem with the approach/retraction screen
AP00958990: 840Dsl SW1.3 SW1.4 SW1.5 ShopMill error upon tapping with scaling
AP00986626: Message "5942 face-milling cutter diameter cannot be measured"
AP00355823: Excessively deep groove on inclined surface (if tool exactly fits to the bottom and D>0) (cycle930)
AP00984008: Asian languages: Cursor control with G code steps (as from basis 07.50.24.00)
AP01005769: Error with tool having a long digit sequence
AP01006474: JOG measuring not functional with active PROG_EVENTS
Also see **12. Information about ShopMill:** General:

Debugs, improvements in ShopTurn 1.5 SP3 HF2 with respect to 1.5 SP3 HF1
==========

- New NCK (V72.08.02) and SINAMICS-Software (V02.50.55.00).

Restrictions which apply to ShopTurn:

- A successful machine test (segment-specific test according to acceptance protocol) is required for the release of the following functions:
  The acceptance protocol is stored on the software DVD under:
  sw_1.5_HF5\documentation\acceptance_protocol
- You can use only one TOA area.
- T=location number refers exclusively to the first turret / tool holder.
- The active location is not displayed with two turrets.
- The "Manual machine" option does not support a counterspindle.
- For the manual machine version, "0" can be entered in the feed field mm/rev or mm/min. The axes will in this case traverse using the data of axis machine data 32020 or 32050. This behavior occurs in ShopTurn when 0 is entered for the setup feedrate under "Settings".
- AP00631062: OP switchover problem with installed add-on software, e.g. Transline.
  Remedy: The resolution can only be switched over on PCU50 if ShopMill or ShopTurn has been entered as runup task in the Regie.ini.
- AP00658965: Exception for the operator interface when using the interface signal DB19.dbx18.0 (tool data adjustment).
  Remedy: The interface signal need no longer be used with standard ShopMill/ShopTurn data $MN_TOOL_DATA_CHANGE_COUNTER set to 0xF (MD 17530) and $MC_TOOL_DATA_CHG_BUFF_SIZE set to at least 80 (MD28450).

Compat. list:

SW Release	HMI-Emb.	ShopMill	ShopTurn	NCU 7x0.x	840Di sl	Usable with PCU 50.3 and HMI-Adv.	OP 08T	OP 10x	OP 12x	OP 15x	OP019-532C
1.5 HF5	07.50.15	07.50.11	07.50.12	72.05.01	-	all 07.05 releases from 07.05.00.01	X	X	X	X	X
1.5 SP1 HF1	07.50.19	07.50.15	07.50.17	72.06.02	-	all 07.05 releases	X	X	X	X	X

						from 07.05.00.01					
1.5 SP1 HF2	07.50.20	07.50.16	07.50.18	72.06.03	-	all 07.05 releases from 07.05.00.01	X	X	X	X	X
1.5 SP2	07.50.21	07.50.17	07.50.19	72.07.01	-	all 07.05 releases from 07.05.00.01	X	X	X	X	X
1.5 SP3	07.50.23	07.50.20	07.50.22	72.08.01	-	all 07.05 releases from 07.05.00.01 all 07.06 releases from 07.06.01.00	X	X	X	X	X
1.5 SP3 HF1	07.50.24	07.50.22	07.50.24	72.08.01	-	all 07.05 releases from 07.05.00.01 all 07.06 releases from 07.06.01.00	X	X	X	X	X
1.5 SP3 HF2	07.50.24	07.50.22	07.50.24	72.08.02	-	all 07.05 releases from 07.05.00.01 all 07.06 releases from 07.06.01.00	X	X	X	X	X