SINUMERIK 840D sl, CNC SOFTWARE 6-3 with SINUMERIK Operate, 6-languages (EN,DE,FR,IT,SP,V-CH) on CF card; with license.	 Export	6FC5850-1XG20-6YA0 6FC5850-1YG20-6YA0
SINUMERIK 840D sl, CNC SOFTWARE 31-5 with SINUMERIK Operate, 6-languages (EN,DE,FR,IT,SP,V-CH) on CF card; with license.	 Export	6FC5850-3XG20-6YA0 6FC5850-3YG20-6YA0
SINUMERIK 840D sl, CNC-SOFTWARE 6-3/31-5 with SINUMERIK Operate, 6-languages, (EN,DE,FR,IT,SP, V-CH) on DVD; without license.	 Export	6FC5850-3XC20-6YA8 6FC5850-3YC20-6YA8

Information on the installation and operation for SINUMERIK NCU 710.2, NCU 720.2, NCU 720.2 PN, NCU 730.2, NCU 730.2 PN.

The CNC software 2.6 HF2 comprises the following components:
(internal version V02.06.00.02.003)

- NCK Software V75.04.02
- SNCK V02.04.00.00.001
- PLC OpSys V20.70.40 (PLC 317)
- PLC OpSys V27.90.06 (PLC 319)
- PLC-FB15 V02.06.07
- CP V01.32.09
- MCP Client V01.04.31
- SINAMICS Software V02.60.45.00
- HMI-sl V02.06.00.00.038
- LinuxBase V02.60.32.00
- SNC V01.02.00.01.002
- Technology Cycles V02.06.21.03
- Measuring Cycles V02.06.21.03
- Shopmill Cycles V02.06.21.03
- ShopTurn Cycles V02.6.21.03
- Cycles ISO Compatibility V02.06.21.03
- MachineDataForCycles V02.06.21.03
- Adapting Cycles V02.06.01

1. Installing the CNC software:
The NCU service system >= V02.60.29 on a USB stick is required to install the CNC software.
The software can be installed for the first time using the NCU service system and TCU with full keyboard or using WinSCP. Installation without keyboard or WinSCP is only possible using a USB stick that has been prepared for automatic upgrade (refer to 1.5).
The software may only be installed on a SanDisk CompactFlash 5000 1GB. The Order No. [MLFB] of the empty card is 6FC5313-5AG00-0AA0.

1.1 Installing the NCU service system on a USB stick:
The NCU service system is available as USB stick image in the directory emergency_bootsys_ncu of the DVD supplied. The installdisk.exe program for copying is located next to it. Connect a USB stick >= 512MB, preferably Siemens 6ES7648-0DCx0-0AA0, to your PG or PC (with Windows XP) and determine the associated driver letters. Call up installdisk to copy the NCU service system to the USB stick:
installdisk --verbose --blocksize 1m <Image file> <Drive letter:>

© Siemens AG 2009
All rights reserved

Example:
(-- means 2x minus characters)
installdisk --verbose --blocksize 1m D:\ emergency_bootsys_ncu\linuxbase-512M.img X:
This means that the capacity of each USB stick is limited to 512MB.
Alternatively:
installdisk --verbose --blocksize 1m D:\ emergency_bootsys_ncu\linuxbase-512M.img X:
If the resize.img is used, then after it has been downloaded, an NCU must be booted from this USB stick so that the image on the stick is unzipped and the FAT partition is generated. This means that the full capacity of the USB stick is therefore available.

The command specified above is preferably executed in a DOS shell.
To do this, you require administrator rights on your PG/PC
After completing installdisk, remove the USB stick from the PG / PC and re-insert it into the PG / PC. Under Windows, you will now see an empty USB stick. Now, copy the required CNC software (file with the .tgz extension) to this USB stick from the DVD supplied, directory ncu_sw.
The USB stick can now be inserted in one of the two USB connectors of the NCU 7x0. When it is powered-up, the NCU boots from this USB stick. Operator control is either from a TCU that is connected to the NCU or via PG / PC using WinSCP under "Open Terminal".
The use of the NCU service system is described in the documentation /IM7/ on the DVD supplied.

1.2 Upgrading an NCU 7x0.2 with CNC-SW 2.6 preliminary

A SanDisk 1GB CF card is required! It is not permissible to upgrade from other software releases. A new installation is then required! It may then be necessary to observe Section 9.5 upgrading cycles.
For an upgrade, all user data on the CF card in the directories /user, /addon, /oem and the license key is kept.
When upgrading systems with several NCUs / PCU50, please carefully note Section 7 regarding networking!

- Boot the NCU from the USB stick (refer to 1.1).
- Using F3, select the service shell.
- Log-on as user "manufact". In this case, it may be necessary to use an external keyboard as the user name must be written using lower-case letters.
- After the $ operator prompt of the service shell, enter the following command:
  sc restore -update /data/<cnc-sw>.tgz
  The software update has been completed when the $ prompt reappears.
- Remove the USB stick.
- Boot the NCU by powering it down/up again.
- The PLC, drives and NC can then be commissioned.

1.3 New installation

IMPORTANT: With this procedure, the contents of the CF card are completely deleted, this also applies for an empty CF card.

- Boot the NCU from the USB stick (refer to 1.1).
- Using F3, select the service shell.
- Log on as user "manufact" - for an empty CF card with "admin", password "SUNRISE". In this case, it may be necessary to use an external keyboard as the user name must be written in lower-case letters.
- After the $ operator prompt of the service shell, enter the following command:
  sc restore -full /data/<cnc-sw>.tgz
  The software installation has been completed if the $ prompt reappears.
- Remove the USB stick.
- Boot the NCU by powering it down/up again
- When booting for the first time with TCU(s), the TCU(s) must be log onto the system. The TCU name (default TCUx) and the address of an assigned MCP must be specified.
- The PLC, drives and NC can then be commissioned.

1.4 Systems without TCU

© Siemens AG 2009
All rights reserved

If there is no OP with TCU to operate the NCU service system when installing the software corresponding to 1.2 or 1.3, then operation is also possible from PCU50 or PC/PG using WinSCP (Commands / Open Terminal) or VNC viewer.
Under certain circumstances, it is not possible to install the software without an NCU service system on the USB stick because 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 execute an automatic "restore -updated" or "restore -full". In this case, a script (autoexec.sh) and the CNC software must be copied to the stick. It is not permissible to change the name of the script. The CNC software copied to the stick must be renamed 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 only runs when the switch is in the 7 position. Otherwise, the NCU boots from the stick and a branch is then made into the service menu. Depending on the designation of the CNC software, the script decides whether a "restart -update" or a "restore -full" is to be carried out. In addition, before the restore, the script runs a data backup of the CF card. The progress made in the individual script steps can be tracked on the 7-segment display or also at the internal SINUMERIK Operate. The precise significance of the displays and the individual steps are described in a separate document. The script is available on the DVD supplied.

2. Excerpt of the new options / functions from NCK 75.00.00 onwards

- **New options for NCK and SINUMERIK OPERATE**
- 6FC5800-0AP00-0YB0 OPERATE WITHOUT SINUMERIK OP
- 6FC5800-0AP13-0YB0 RESIDUAL MATERIAL DETECTION AND MACHINING
- 6FC5800-0AP17-0YB0 SHOPTURN/SHOPMILL
- 6FC5800-0AP54-0YB0 CYCLE PROTECTION
- 6FC5800-0AP60-0YB0 SINUMERIK OPERATE RUNTIME OA PROGRAMMING
- 6FC5800-0AP64-0YB0 SINUMERIK OPERATE RUNTIMELIZENZ OA EASY SCREEN
- 6FC5800-0AP65-0YB0 SINUMERIK OPERATE RUNTIMELIZENZ OA SOLUTION PARTNER
- 6FC5800-0AS07-0YB0 ADVANCED SURFACE
- 6FC5800-0AS16-0YB0 SPLINE-INTERPOLATION
- 6FC5800-0AS31-0YB0 HANDLINGSPACKAGE
- 6FC5800-0AS32-0YB0 TECHNOLOGY PACKAGE MILLING SINUMERIK MDYNAMICS 3 AXES
- 6FC5800-0AS33-0YB0 TECHNOLOGY PACKAGE MILLING SINUMERIK MDYNAMICS 5 AXES
- **New functions for NCK**
- The user interface between NC and PLC (for FC21) has been increased from 1kByte to 4kByte.
- The active transformation can be determined using the system variables $P_TRAFO_PARSET.
- The possible number of all of the programs in the NCU has been increased.
  ($mn_mm_num_files_in_filsystem = 768)
- The "compressor" function is no longer marketed as option. However, the corresponding machine data must be set to activate the function.
- New program commands CTOL and OTOL for contour and orientation tolerance.
  The tolerance programmed in this way applies to all of the compressors - today and in the future - the blending types G642, G643, G645, OST and the orientation smoothing ORISON.
  For specific applications, in addition, the tolerance for each axis can be separately programmed using ATOL[AXIS].
- FB 2 read NC variable and FB 5 read GUD variable
  have been expanded by a format which can be used to save 64 bit (real 2) floating point values with the full resolution.
- FC13 display control for HT2 has been expanded. More positions can now be displayed. In this case, the prerequisite is that the variables can be transferred as 64 bit floating point values.
- In combination with a Sinumerik 840D sl NCU, two chassis units with DAC motor modules have been released. The Sinumerik safety functionality is only released when used with DAC motor modules. The SAFE BRAKE CONTROL function from dbSI-0 has not been released.

3. Notes:

© Siemens AG 2009
All rights reserved

- After installing CNC-SW 2.6 on the CF card, a general reset must be made for the NCU and PLC: SIM/NCK switch in position 1, PLC switch in position 3 and then power-down/power-up.
- NCU730: When alarm "2120 NCK fan alarm" is output, then an immediate response (approx. 20 seconds) must be realized in the PLC program and the axes shut down in a controlled fashion, if a double-fan module is **not** being used.
- NCU710 / NCU720: The NCU does not boot if either the fan module is not available or if it is defective. This only applies to the NCU730 equipped with a double fan module if both of the fans have failed.
- Alarm 15122 Power ON after power failure "%1 data was restored, of which %2 machine data, %3 errors"
  If %3 - the number of errors occurred - is greater than zero, then it is not recommended to continue processing with the data. In order to avoid subsequent problems, an actual data back-up should be read-in (downloaded).
- It is not always possible to port an NCK series commissioning archive from 840D power line to 840D sl without processing.
- Before overwriting a licensed CF card, it is absolutely necessary to back up the license key. The key is included in the 'keys.txt' file and is located in the */card/keys/sinumerik* path.
  The key can be backed-up e.g. using WinSCP from the PG/PC.
- The licenses are permanently linked to the particular CF card (card ID) and can only be used on this card.
- Data backups should be read-in (downloaded) in the following sequence - NC, PLC, drive. It makes sense to separately generate the NC, PLC and drive archives.
- Contour handwheel and velocity override using the handwheel are only released with Profibus MCP.
- The reset knob at the NCU allows the NCU module to restart. It cannot be guaranteed that the complete system can be restarted for all configurations. This also applies to the PI service from the PLC with FB4.
- A template of the actual standard basesys.ini is available in the catalog /card/siemens/system/etc.
- Port 102 on X130 is interlocked as standard. If the control is to be accessed via X130 using Step 7 or a SINUMERIK Operate, then port 102 must be enabled.
  This can be done by making the following entry in the file /user/system/etc/basesys.ini:
  [LinuxBase]
  FirewallOpenPorts=TCP/102
  If required, port 5900 can also be enabled for an external VNC viewer.
- The Ethernet handwheel of the HT2 is connected (marshalled) to 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 (handheld unit), included in the toolbox for Powerline, cannot be transferred to the HT2 without first making some changes. The variable descriptions of the NCVAR must be adapted. FC13 now supports the visualization of four lines. As a consequence, the significance of the "ROW" parameter has changed as follows:

  0 = no display output
  1 = 1st line
  2 = 2nd line
  3 = line 1 and line 2, alternating
  4 = 3rd line
  5 = line 1 and line 3, alternating
  8 = 4th line
  F = automatic change of all lines

  The contents of the display must be saved in the "ChrArray" field, type string[64].
  The upper four keys are now transferred into the input image of the PLC
  (the LEDs under these keys still cannot be addressed).

  Example of the parameter setting for the HT2 at FB1 (OB100):

  ```
  BHG             :=5
  ```

© Siemens AG 2009
All rights reserved

```
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 switch * the connection box or the MPP.
BHGRecGBZNo     :=2 // corresponds to the setting of the rotary switch * the connection box or the MPP.
BHGRecObjNo     :=1
BHGSendGDNo     :=2
BHGSendGBZNo    :=1
BHGSendObjNo    :=1
BHGMPI
BHGStop
BHGNotSend
```

* Rotary switches S1 and S2 at the connection box or MPP are used to define the DIP number with which the participant registers itself in the system (here, S1=0, S2=2).). Address "0" is not permissible, as with this setting, the DHCP server cannot allocate any addresses.

- HT2: Display of 12 characters incl. the decimal point and sign with FC13. In order to be able to use the extended display at the HT2, the values (these are generally the position values) must be transferred to the FC13 in the real 2 format. In order that FC13 can appropriately process the values, the Parameter DataType must be set to B#16#30. The values must be transferred in exactly the same way as they are read from FB2 / FB5 in the real 2 format. It is not possible to interpret the values between output FB2 / FB5 and input in FC13. The values read-in from FB2 / FB5 in the real 2 format must be appropriately converted in FC13.
  The Digits parameter specifies the number of decimal places (value range 1..9). The display update is managed in FC13. The signals "Acknowledge numerical display" in the EBm+5 and "New data for selected line" in ABm+1 no longer have to be interconnected in the PLC.
  Example:
  - displaying four axis position values that are read with FB2.
  - The results of the read operation from FB2 are saved in the real 2 format in DB399.DBX0.0,... DB399.DBX8.0, etc.
  - the display is to three places after the decimal point.

```
FUNCTION FC 399 : VOID
TITLE =
{ S7_language := '9(1) English (USA)  01.04.2009  15:23:07' }
VERSION : 0.0
CODE_VERSION1

VAR_TEMP

END_VAR
BEGIN
NETWORK
TITLE =  NETWORK


//attention: FC13 writes to AB1 of HT2

//1st axis actual position to row 1

 SET   ;
 =     DB399.DBX  150.7; //always one in QB m.7

CALL FC    13 (
Row                 := B#16#F,// display four rows
ChrArray            := DB106.display, // string array[64]
Convert             := TRUE,
Addr                := DB399.DBX    0.0, // 1st result from FB2
DataType             := B#16#30,
StringAddr           := 16,  // row 1 address in string
```


All rights reserved

```
Digits                  := B#16#3,
Error                   := M    610.0);

//2nd axis actual position to row 2

CALL FC    13 (
Row                     := B#16#0,
ChrArray                := DB106.display,
Convert                 := TRUE,
Addr                    := DB399.DBX    8.0, // 2nd result from FB2
DataType                := B#16#30,
StringAddr               := 32,  // row 2 address in string
Digits                  := B#16#3,
Error                   := M    610.1);

//3rd axis actual position to row 3

CALL FC    13 (
Row                     := B#16#0,
ChrArray                := DB106.display,
Convert                 := TRUE,
Addr                    := DB399.DBX   16.0, // 3rd result from FB2
DataType                := B#16#30,
StringAddr              := 48,  // row 3 address in string
Digits                  := B#16#3,
Error                   := M    610.2);

//34th axis actual position to row 4

CALL FC    13 (
Row                     := B#16#0,
ChrArray                := DB106.display,
Convert                 := TRUE,
Addr                    := DB399.DBX   24.0, // 4th result from FB2
DataType                := B#16#30,
StringAddr              := 64,  // row 4 address in string
Digits                  := B#16#3,
Error                   := M    610.3);

BE    ;

END_FUNCTION
```

- 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
- Combination of NCUs with Sinamics Sxxx CUs have not been released. Exceptions are only possible after prior consultation:
- Traversing the spindle with limited torque (Focon) or traversing the spindle to a fixed stop:
  The power displayed at the SINUMERIK Operate is calculated from the limited torque as 100% and displayed.
- From NCK >= V67.05.03 onwards and ADI4 board V01.04.04, the ADI4 board can be operated at the 840D sl with internal PLC 317.
- Dbsi 1 , just like before, has not been released in conjunction with SINUMERIK and integrated drive control.
- An NCU Bios >= V1.4 is required for operating a 1GB CompactFlashCard in an NCU730.
  The BIOS is only available through e_support as an upgrade is only required very infrequently.
- Starter, in conjunction with 840 D sl, has, as before, only been released for diagnostics.
- If the number of part programs set by the machine data approximately correspond to the number of actually used part programs, the value in the machine data must be increased (as a result of the additional cycles).
- Interpolatory traversing of positioning axes is prevented with Alarm 8031 "...axis has no IPO functionality."
- SPI and PW are keywords (they have always been keywords) and cannot be used as axis identifiers.
- The function G643 (internal block blending) has been released for applications in the tool change area (e.g. optimizing the approach to the tool change position). It has not been released for applications in the

© Siemens AG 2009
All rights reserved

machining process itself.

- The function G644 (blending with possible dynamic performance) has been released for applications in the tool change area (e.g. optimizing the approach to the tool change position). It has not been released for applications in the machining process itself.
- Software version, max. 6 axes: The standard pre-assignment for the number of axes is defined as 3.
- The "extended measuring" function with the distributed measuring input at the SINAMICS modules does not function (MEAC, MEASA, MEAWA).
- No channel gaps are possible in a system with a maximum of two channels.
- Access to drive data via system variables $nn_nn has not been released. Only the system variables transferred in telegram 116 are possible.
- ESR has not been released.
- Now, analog to the 840Di functionality, ASUBs can be started via Profibus PLC I/O. The machine data should be selected the same as for 840Di. There are no fast NCK I/O (peripherals) that are mapped in the DB10 of the PLC. However, the known ASUB start options (from DB10 or FC9 via PLC program) are still possible.
- Fast ET200 Simatic modules can be used to replace the DMP modules used for Powerline. In this case, for the PLC317 - in conjunction with NCK >= V 67.07.05 - there is a possibility of operating one of the two Profibus lines in clock cycle synchronism (details on this are provided in a separate description).
- When using 31-axis software on an NCU710.1, 6 of the 31 axes can be used and 4 channels from 10 channels.
- Machine data 10008 $MN_MAXNUM_PLC_CTRL_AXES is no longer in the data set. It has been replaced by MD19160.
- MD 10062 $MN_POSCTRL_CYCLE_DELAY must be zero. Check the existing data backup.
- Machine data 32250 $MA_RATED_OUTVAL[] must have a value of zero.
- IMD (Integrated Monitoring & Diagnostic) :
  The feedback signal in the GUD variable _PM_MISSING_TOOL_REFRESH after writing “1” to variables is only updated when the screen changes.
- Safety: There is a new handling option that involves synchronized actions or the synchronized action elements. Up to status 1.3 (NCK 62), using the SI option - synchronized action level 2 was simultaneously internally set. This meant that the channel MD 28250, NUM_SYNC_ELEMENTS was able to be set to a value > 159.
  This situation changes with NCK release 67. The synchronized action level 2 is no longer included in Safety. This is the reason that there is a new machine data in which the synchronized action elements for SAFE.SPF are defined, MD 28251, NUM_SAFE_SYNC_ELEMENTS.
  With option SLP_I_O=1, a maximum of 500 can be written to this machine data - and for SPL_I_O=2, a maximum of 5000.
  However, the number (count) of required synchronized action elements should be determined in order to avoid that the performance is unnecessarily reduced.
  The number (count) of the free SI synchronized action elements can be read using the system variable $AC_SAFE_SYNAC_MEM.
  If this variable is called before the start and after SAFE.SPF has run, then the difference is the number (count) of elements that SAFE.SPF occupies. This difference should be entered into MD 28251 with a certain amount of reserve.
  An index 1-5 must be specified when writing to MD 25300 using synchronized actions.
- Safety: Changed checksums for safety functionality integrated in the drive.
  As a result of the new safety functions integrated in the drive, the checksums for these functions must be re-acknowledged.
  To do this, "95" (Safety-commissioning) must be entered into parameter p10.
  The parameters of the actual checksums must then be copied to the parameters of the reference checksums:
  p9798 to p9799
  p9898 to p9899
  Then, p10 must be again set to zero. As a consequence, the values are automatically saved (p971 is set to 1). Wait until the data has been completely saved.
  The operations can also be initiated in HMI-Advanced under IBN/NC/Safety Integrated using the softkeys "Activate commissioning mode" and "Deactivate commissioning mode".
- The help function M6 is no longer output as standard a block search.

© Siemens AG 2009
All rights reserved

Counter-measure:
Change the help function group in MD 22040 $MC_AUXFU_PREDEF_GROUP[5] into a free group.

- If the value of MD18210 deviates from the standard pre-assignment, in order to upgrade from SW1.4 / 2.4 to SW1.5 / 2.5, then as a minimum, MD18210 must be increased by 21 MB DRAM.
- HT2 can be used as handheld operator unit from software release 1.5 HF5 / 2.4 SP1 and higher. Depending on the setting in basesys.ini, the HT2 is booted from the master (refer to the general information on networked systems). The master can be an NCU as well as also a PCU 50.
- Tool manager: There is a new handling option. If several real magazines are being used, then this can be enabled using an option. A real magazine is included in the basic scope.
  For compatibility reasons, the previous option $ON_TECHNO_FUNCTION_MASK Bit4=1 must still be set to 1 - however it is not included in the license handling.
- Toolbox from V01.05.01 and higher and Step 7 from V5.4 SP2 and higher are required.
- The retentive memory of the PLC 319 is limited to 256kB.
- Deleting PLC 319:
  After deleting the PLC, by actuating the PLC rotary switch or from Step 7, the PLC program is subsequently automatically reloaded.
  If PLC initialization is initiated using the operator action - PLC switch in position 3 and power off/on, then the PLC program is not automatically loaded. This is also then no longer realized with the above mentioned sequence of operator actions.
- The actual cycle time is no longer available in DB5 but in DB8. The cycle time can also be called via the start infos of the OB1.
- The PLC module IM153-2 MLFB 6ES7 153-2AA02-0XB0 cannot be used.
  Counter-measure: 6ES7 153-1AA03-0XB0
- The PLC series (standard) commissioning archive must be generated when the PLC is in the stop state. Otherwise, it cannot be guaranteed that the PLC resumes cyclic operation after loading the data back-up.
- The star/delta changeover with FC17 requires Sinamics parameters.
  Prerequisites:
  The appropriate DDS/MDS must be set-up in the drive!
  P827[0] ⇔ P827[1] ⇔ P827[n] must not be equal to one another!
  The contactor is changed over by the application => P833, bit0=1
  Pulses are cancelled by the drive => P833, bit1=0
  PLC output setting: Star operation, this is the reason that when booting (powering-up), a "1" should be set in the axis DB bit 21.5
- The service interface X127 of the NCU should only be used for commissioning and service.
- The machine control panel (MCP) is operated on Profibus or Ethernet. Mixed operation is not permissible.
- Profibus1 must be configured so that the softkeys CU, Infeed, Drives are displayed at the SINUMERIK Operate.
- For PLC 319, Profibus diagnostics using FB125 are not possible.
- PLC 319, ProfiNet functionality: CBA PLC communication has been released.
- Blocks FC1007 AG_LOCK, FC1008 AG_UNLOCK, FC1010 AG_CNTRL have not been released.
- The FORCEN [FORCING] function of the PLC in conjunction with 840D sl has only been released ~~with some restrictions~~. It is possible with the PLC317 from PLC operating system >= V20.70.37 (e.g. 2.4 SP1 HF11) / PLC319 and higher with a PLC operating system >= V26.90.10 (e.g. 2.4 SP1 HF11) and higher.
- For 840D sl, it is not permissible that the VDI interface signal DB10.DBX57.3 is connected up.
- HMI-Advanced software / commissioning tool from 7.6 and higher is required to commission the drive.
- Functions to measure i and n using Starter and HMI Advanced cannot be used in parallel. If Starter measurements are required, this requires a power OFF=>ON and then measurements must be immediately made using Starter. Measurements performed with Starter must not be preceded with measurements performed using HMI-Advanced.
- If a SINAMICS component is replaced or upgraded, then the firmware release must be checked and if required, brought to the original release status.
  (Key word: Macro 150399)
  The firmware releases can be checked in the following parameters:
  Control Unit R18 ➔ Firmware release of the CU
  Infeed R128 ➔ Firmware release of the infeed

© Siemens AG 2009
All rights reserved

Drive MD R128 ➔Firmware release of the power unit
R148 ➔Firmware release of the sensor module

- Initially, Starter is still required to determine the motor code.
- Line supply voltages: In order that the system operates smoothly in the final destination country, the following parameters must be set in the specified sequence
  P010 = 1
  P210 = Rated line supply voltage of the country in which the equipment is to be operated
  P211 = Rated line supply frequency of the country in which the equipment is to be operated
  P340 = 1
  P3410 = 4
  P3900 = 3
  Save RAM to ROM
  Power-down the system, only power-up again when it is actually connected to the line supply in the country in which the equipment is to be finally used.
  A line supply identification is performed with the next ON command at the ALM - and the values determined saved so that they are not lost if the line supply fails.
- The APC function increases the NCU load. The impact on performance should be estimated before it is used.
- p1240, p1244, p1248, p1250:
  If, in SW V2.4, VDC control and DDS changeover were activated, then after an upgrade, the contents of the indices >0 from p1240 should be checked to ensure that they have the correct value.
- Drive clock cycles of 62.5 µs have been released with 840D sl and SW 2.5 SP1. The following conditions then apply:
  - two axes can be operated with this setting on a CU or NX.
  - only the same clock cycles can be set on a double-axis module
- For SINUMERIK software releases 1.3 and 1.4/2.4, different firmware versions of the components involved in the **safety functions** (NCU, NX, Motor Modules, DRIVE-CLiQ motors) can be mixed without having to adapt the firmware versions.
  From SINUMERIK software release 1.5/2.5 and higher, the following applies:
  The firmware versions of the Motor Modules, Sensor Modules and DRIVE-CLiQ motors involved in the safety functions (these include integrated Sensor Modules) must be adapted to the SINAMICS firmware version of the NCU.
  This is realized automatically when booting if parameter p9826 (firmware, automatic) is set to 1 (standard setting). When **Safety Integrated** is used, parameter p9826 (firmware, automatic) must be set to 1 and must not be re-parameterized. For a **Safety Integrated** acceptance test , 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

  Every line in the tables represents a permissible combination of the safety firmware versions.

- The maximum drive system utilization of the CU or NX must not exceed 80% (the utilization level is displayed in parameter 9976[xx]).
- The NCU load level as a result of the position controller and interpolator should not exceed **50%** in the “reset“ state. The actual load can be checked under Diagnosis / System resources.
- Cam output (position switching cycles) for modulo rotary axes is not possible in a time window < 4 IPO clock cycles.
- The MPPs and MCPs PN are operated in the Ethernet mode at the 840D sl. Switch positions 9 and 10 simultaneously "on" (PN mode) is not permissible.
- The machine data to describe the spindle dynamics must be set so that they approximately correspond to the actual dynamics of the spindle. f the values are unnecessarily increased, then alarms can occur at the transition from spindle operation to positioning operation.
- NCU 7xx.2 PN: It is not possible to access Profinet PLC I/O from the NCK via system variables.
- NCU 7xx.2 PN: Profinet interfaces 1 and 2 (upper two) are available.
- NCU 7xx.2 PN: Clock cycle synchronous operation is not permissible.
- The "Parking axis" function with an SMI encoder is not possible without power off/on.

© Siemens AG 2009
All rights reserved

- Alarm 7300: If NC archives from software releases < NCK 75.00.00 are used, Alarm 7300 is output if MD18235=0. MD18235=20000 is the standard setting.
- The function "Approach fixed stop G75" is not permissible for active radius compensation G41 / G42 (Alarm 14091, Index 9).
- During fixed-point approach with G75, the fixed point is approached with non-linear interpolation (RTLIOF). As a result, the axis are traversed with the set jerk for the positioning axes ($MA_JOG_AND_POS_MAX_JERK).
- Machine data 11295[0..9] now configures the memory. When a change is made, this is not signaled by an alarm. Values deviating from the standard must be removed in existing archives before making the upgrade.
- The development work to improve the properties of Ethernet and handwheels has still not been implemented for HT2 and HT8.
- Alarm 8025: Advanced Surface 6FC5800-0AS07-0YB0 is an option and is interlocked using an option data.
  After the setting the option, the function for 3-axis machining operations can be used.
- If you wish to use the X127 service interface with a PG/PC for an NCU7x0.2 with SW 2.6, then it has to be noted that the network interface of the PG/PC operates in the "DHCP client" mode - i.e. the "IP address is automatically allocated".
  If the network interface of the PG/PC has been set to a fixed IP (192.168.215.xx), then although a ping goes to X127, neither with Step7 nor with an HMI is it possible to go online. The firewall is activated as the default setting.
- ~~No SMI motor can be commissioned after the DMC20/DME20.~~
- STEP 7 hardware configuration Sinumerik NCU 7xx.2 PN:
  The different versions V2.4 / V2.6 / V2.7 in the hardware catalog refer to the appropriate PLC operating system of the PLC 319. The PLC operating system is part of the NCU software and thus depends on the NCU software version. The operating system included in the software version can be checked in the diagnostic screen (Diagnosis / Version / PLC / Details).
  As a consequence, the NCU must be selected in the STEP 7 hardware configuration.
  Example:
  The NCU Software Version 2.6 HF2 comprises a PLC operating system 319-3PN/DP Version 27.90.xx.
- ~~Therefore, a NCU 7xx.2 PN (V2.7) must be selected in the hardware configuration.~~

- **Machine data change as a result of the changed Drive ES SlaveOM**
  Drive ES SlaveOM is responsible for allocating addresses of the drive units at the Profibus of the PLC. Addresses are allocated in the PLC HW Config. The addresses allocated here must be entered into the NC machine data MD13050[nn] and when safety is active they must be entered into MD10393[nn] - corresponding to the axis configuration required.
  When Starter from V4.1.n and higher is installed (later, also with the installation of Toolbox V2.6), the "new" Drive ES SlaveOM is linked in. Now - for each CU or NX - an address space is reserved for 6 axes - this is different than previously. This occurs when newly generating or adapting a PLC hardware configuration. As a result, under certain circumstances, addresses can shift or gaps can occur.
  This modified address distribution **must** then be communicated to the NC in machine data 13050[nn] and when safety is active, in MD10393[nn] – as otherwise, the axis assignments are no longer correct. This is especially the case if you are working with the existing NC data backups. The reason for this is that they 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 NC modules, the address is incremented with an offset of 10. Machine data MD13120[1+n] must be appropriately adapted.
  If existing PLC projects are imported into an S7 environment with "new" Drive ES SlaveOM, then the old address configurations are kept. However, drive objects that have been newly inserted in this project have the new address allocation. A standard "new" address allocation is achieved using "reorganized addresses".
  If, for NX objects from an existing configuration, "Standard telegram 1 (PZD-2/2)" is entered, then this must be replaced by telegram 390 in order to achieve a standard, new address allocation.

Example:

© Siemens AG 2009
All rights reserved

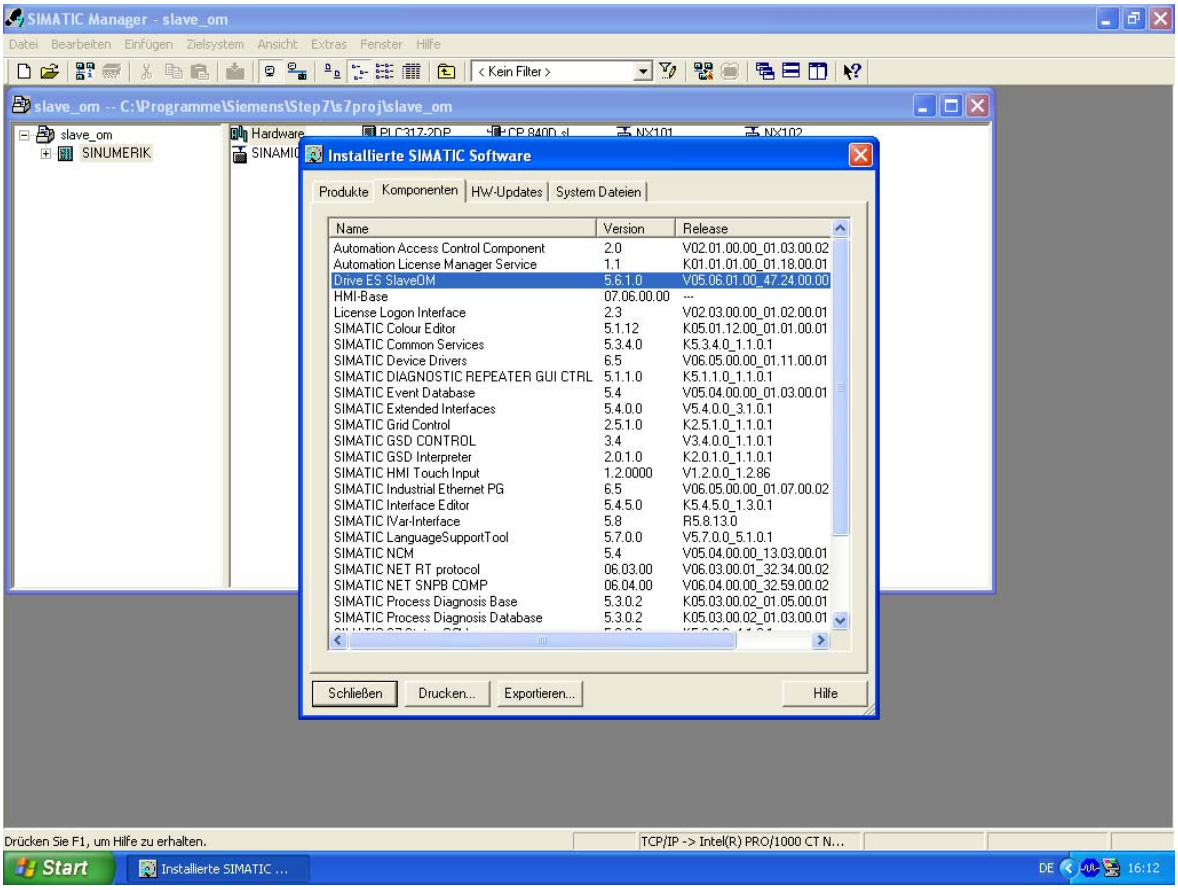
An NCU is configured with a Sinamics Integrated and two NX10 each with 3 axes.

Address distribution with the "old" version of Drive ES SlaveOM:
On the NX10_1, the 3rd axis has address 4420
On the NX10_2, the 1st axis has address 4460

Address distribution with the "new" version of Drive ES SlaveOM:
On the NX10_1, the 3rd axis has address 4420
On the NX10_2, the 1st axis has address 4580

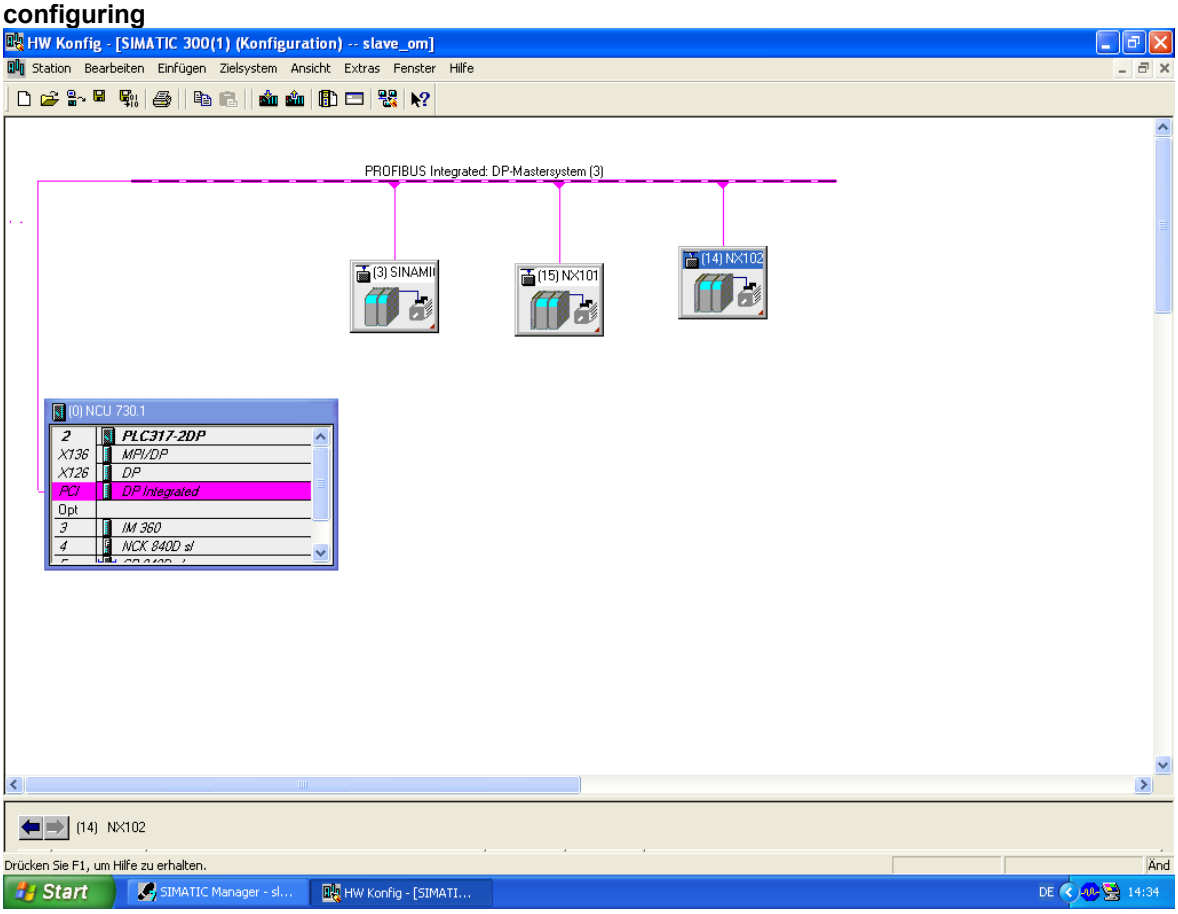
Tip:
When re-configuring existing configurations, in order that a complete change over is made to the new addressing, a start should be made at the lowest slave number to initiate the address allocation using the key "addresses".
All of the slaves have been allocated new addresses, if the note "No change" is output for each slave when initiating the address allocation.

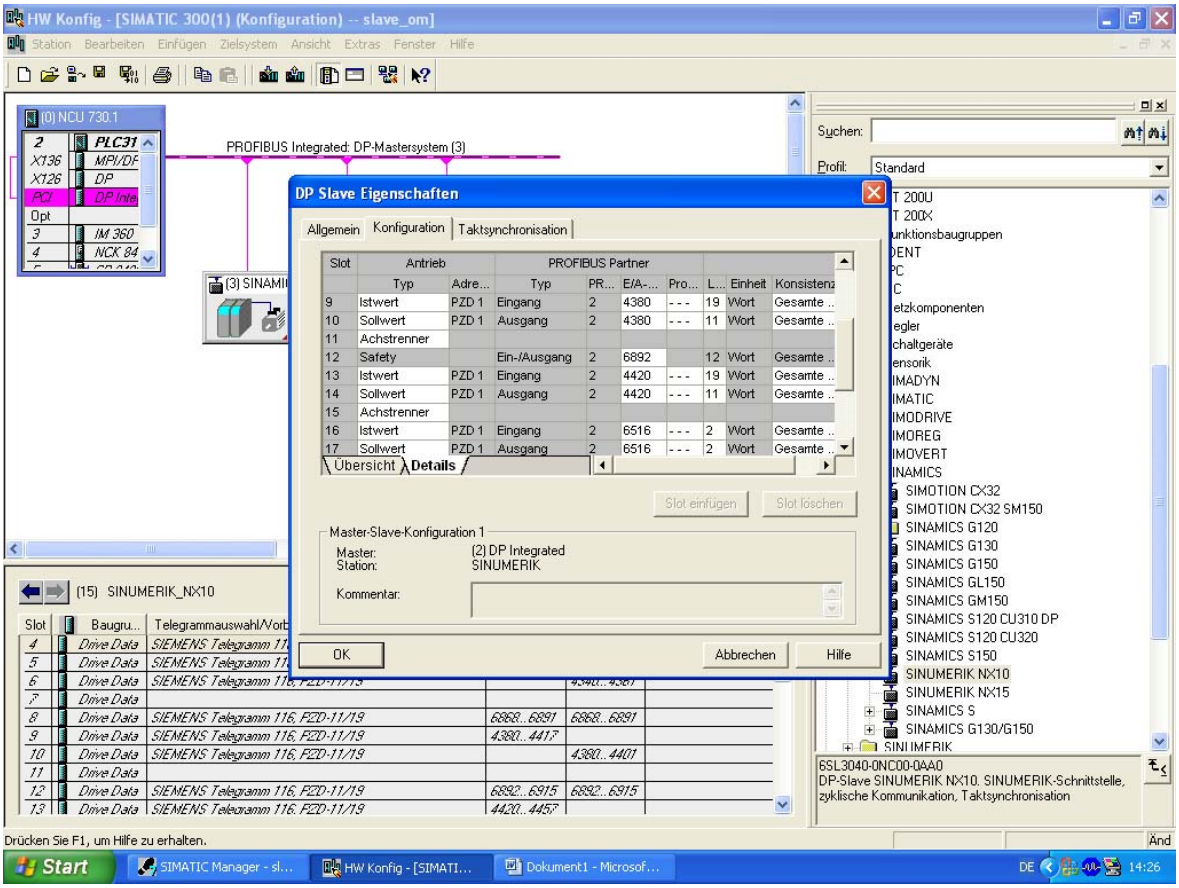
"Old" version of Drive ES SlaveOM



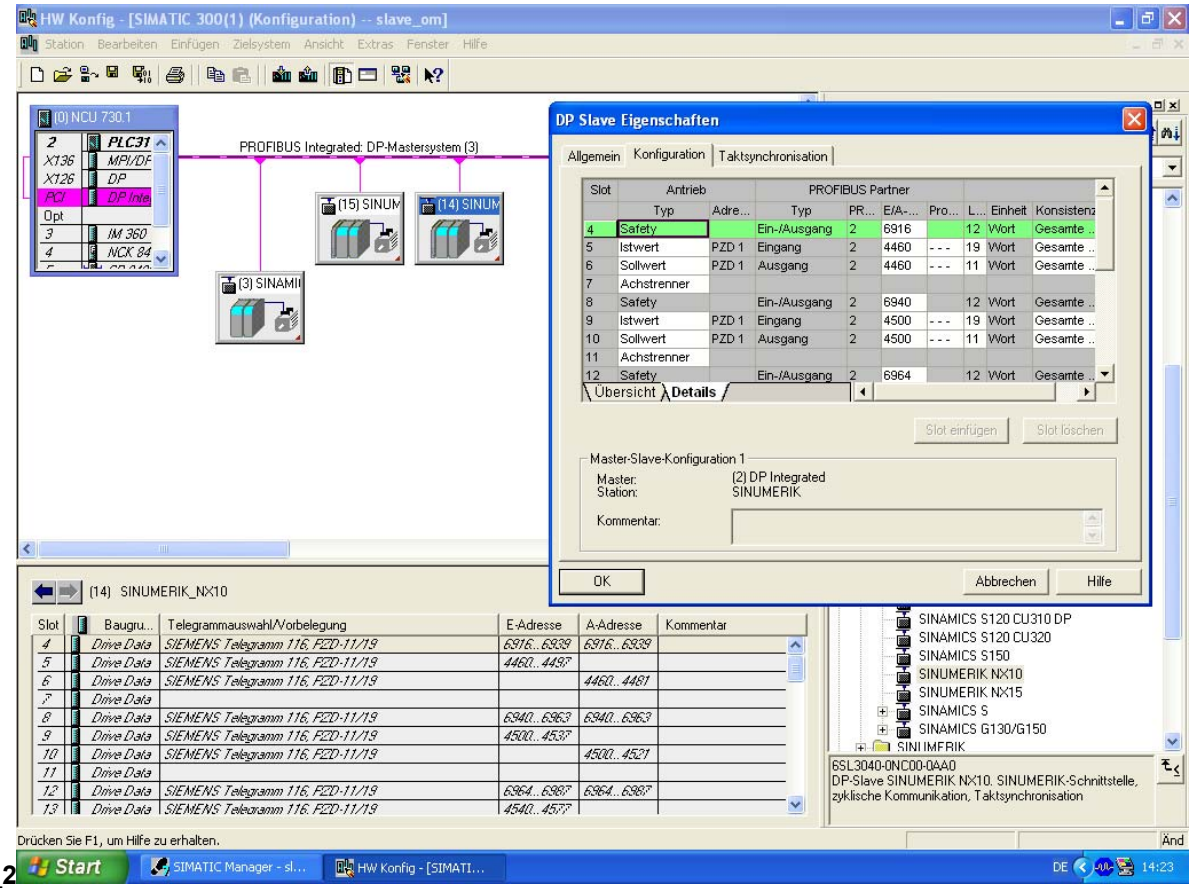
© Siemens AG 2009
All rights reserved

PLC HW configuring




HW Konfig - [SIMATIC 300(1) (Konfiguration) -- slave_om]
PROFIBUS Integrated: DP-Mastersystem (3)
DP Slave Eigenschaften
Allgemein
Konfiguration
Taktsynchronisation
Slot
Antrieb
PROFIBUS Partner
Übersicht
Details
Slot einfügen
Slot löschen
Master-Slave-Konfiguration 1
Master:
Station:
(2) DP Integrated
SINUMERIK
Kommentar:
OK
Abbrechen
Hilfe
(15) SINUMERIK_NX10
Suchen:
Profil:
Standard
SIMOTION CX32
SIMOTION CX32 SM150
SINAMICS G120
SINAMICS G130
SINAMICS G150
SINAMICS GL150
SINAMICS GM150
SINAMICS S120 CU310 DP
SINAMICS S120 CU320
SINAMICS S150
SINUMERIK NX10
SINUMERIK NX15
SINAMICS S
SINAMICS G130/G150
6SL3040-0NC00-0AA0
DP-Slave SINUMERIK NX10. SINUMERIK-Schnittstelle, zyklische Kommunikation, Taktsynchronisation
Drücken Sie F1, um Hilfe zu erhalten.
Start
SIMATIC Manager - sl...
HW Konfig - [SIMATI...
Dokument1 - Microsof...
14:26

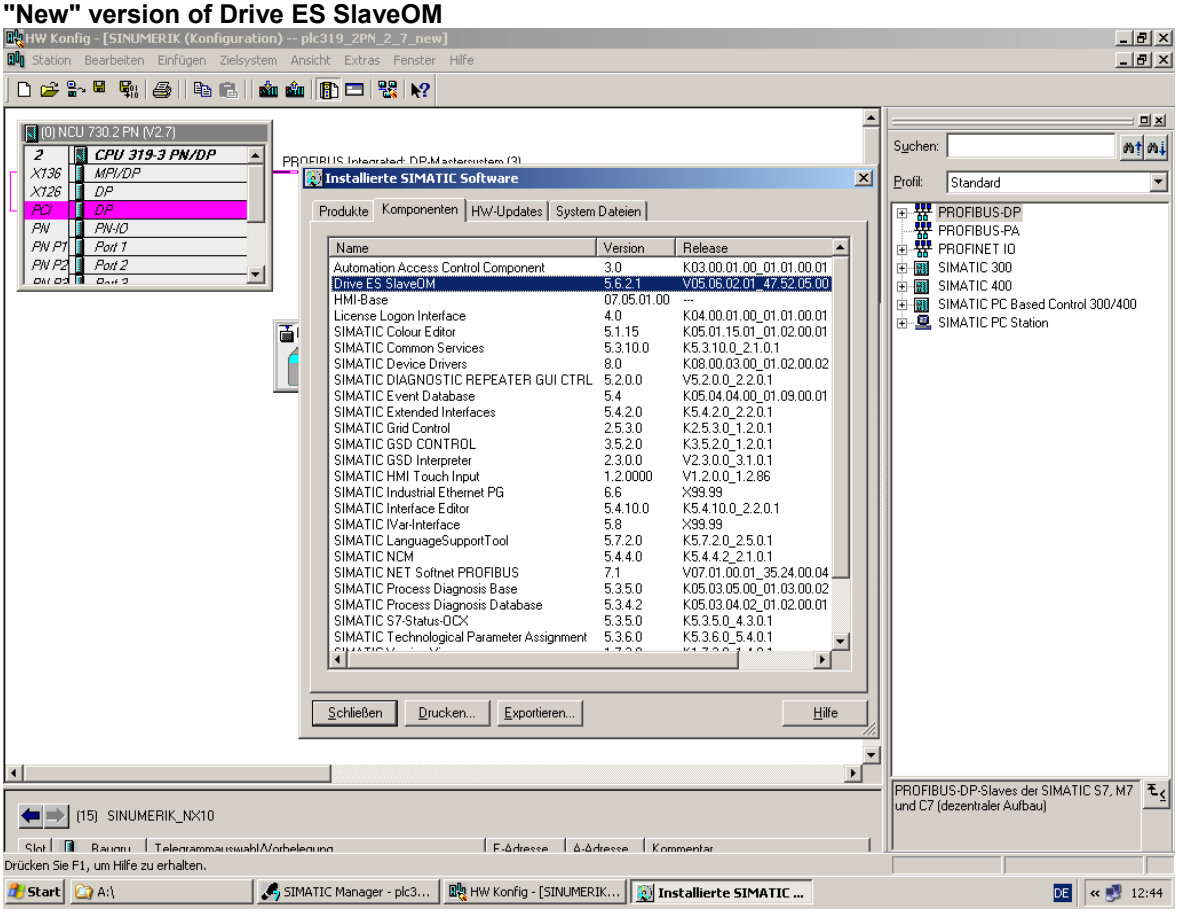
NX10_1



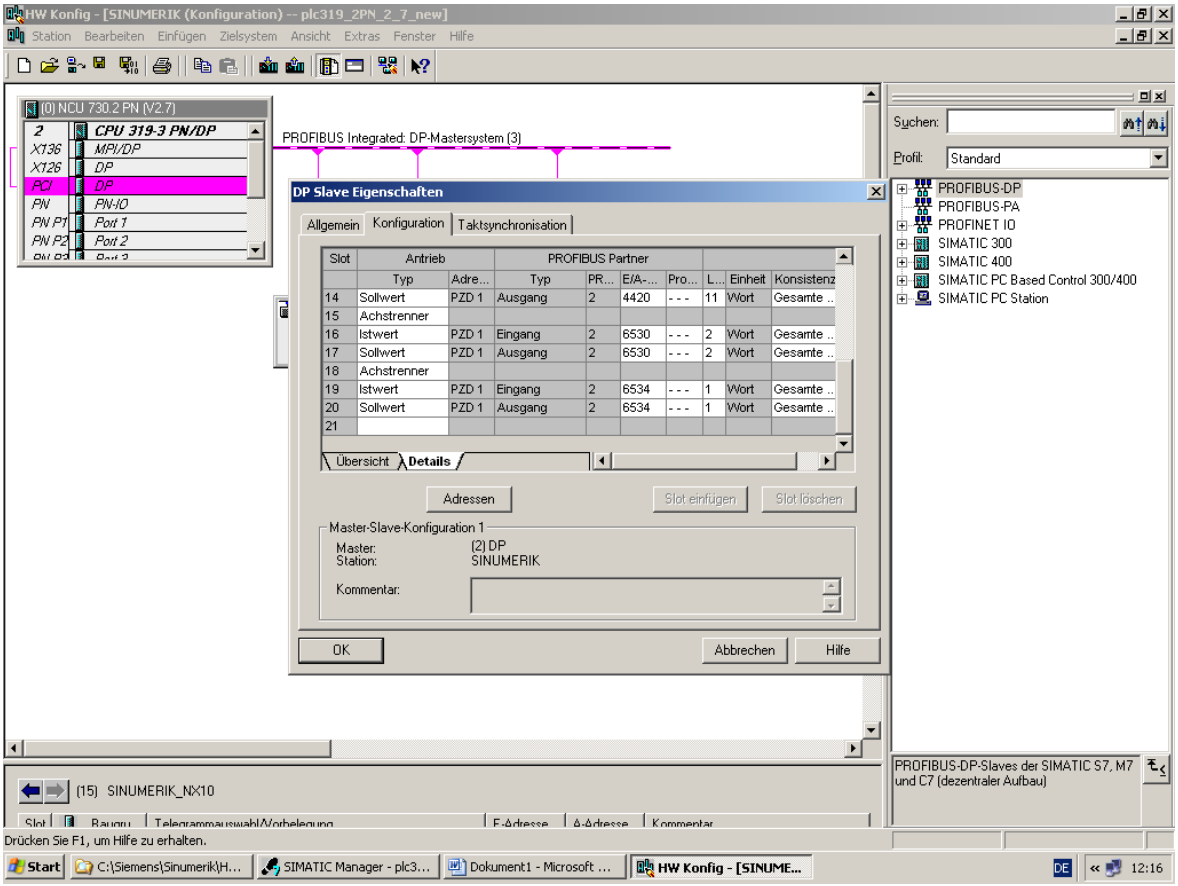
NX10_2

All rights reserved

"New" version of Drive ES SlaveOM

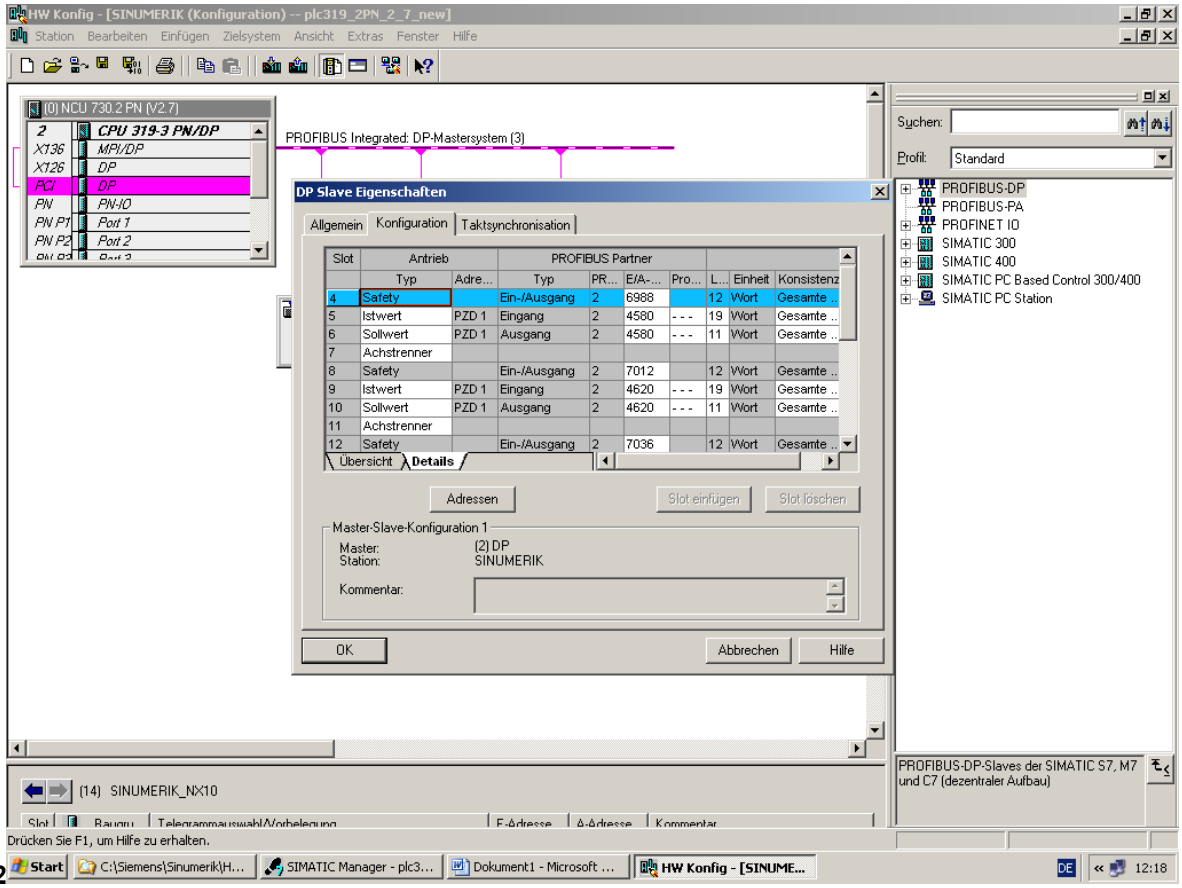


© Siemens AG 2009
All rights reserved



NX10_1

All rights reserved

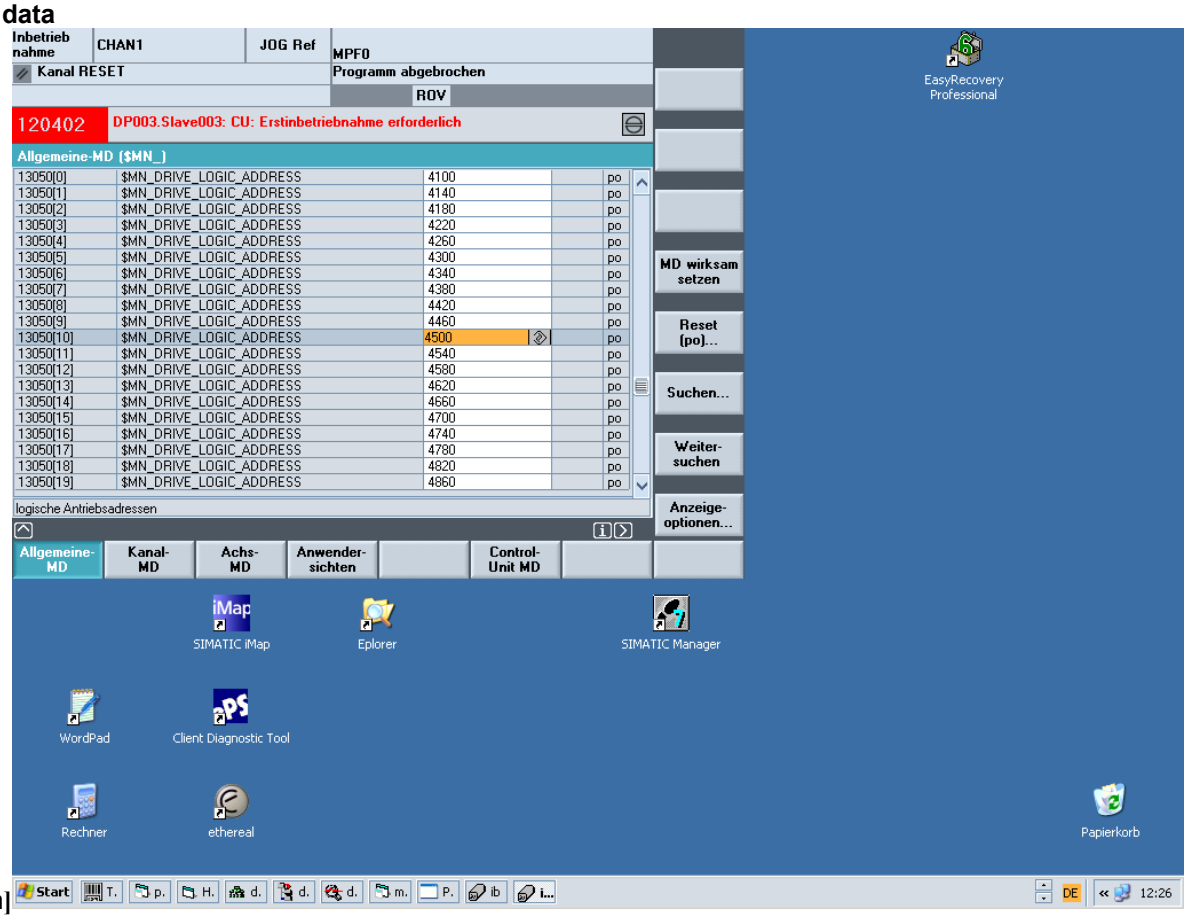


NX10_2

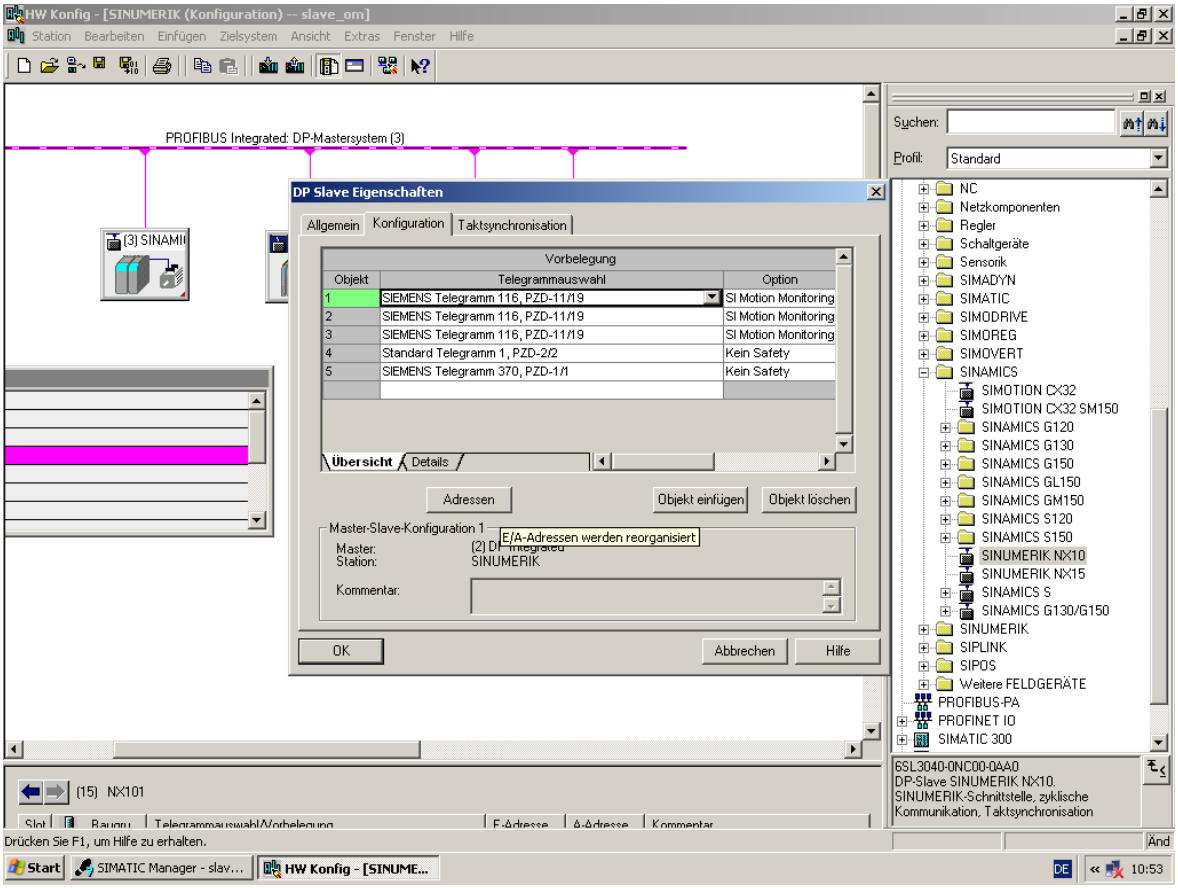
All rights reserved

Machine data

13050[nn]



Reorganizing addresses



7. SINUMERIK Operate I

7. 1 New functions in the SINUMERIK Operate:

- Stock removal resp. face milling in JOG
- Measuring cycles 2.6 are supported
- Option 6FC5800-0AP64-0YB0 SINUMERIK Operate runtime license OA Easy Screen (previously supplement operator interface / Wizard)
- Drive diagnostics
- V24 data transfer for part programs

7. 2 Notes regarding SINUMERIK Operate:

- The SINUMERIK Operate 2.6 software is automatically started as a component of the CNC software 2.6 when NCU 7x0 is started. SINUMERIK Operate is also called "internal SINUMERIK Operate" (contrary to the "external SINUMERIK Operate" on the PCU 50.2).
- The internal SINUMERIK Operate can only be operated from an operator panel with TCU (e.g. OP 012+TCU, HT 8,...) which is connected at X120 of NCU 7x0 using Ethernet. Operator control via PC/PG using the VNC viewer is possible.
- The configuration data such as e.g. PLC message and alarm texts must be saved on the CF card of the NCU in the corresponding directories under /card/user/sinumerik/hmi/... or /card/oem/sinumerik/hmi/... (refer to the commissioning instructions HMI sl). The file system on the CF card can only be accessed "online", i.e. only when the NCU is running. Data can be transferred using a USB stick or network connection via the SINUMERIK Operate operator interface (Commissioning / System data area) or using a network connection and the RCS Commander tool or WinSCP.
  IMPORTANT: Names of configuration files must always be written in lower case letters.
- When upgrading from CNC-SW 2.4, under certain circumstances, display MD values must be manually converted (from the hmi_md.ini file) into the corresponding new NC-MD. Refer to HMI sl commissioning instructions, section 9.1 for the conversion table.
- For storing part programs and archives on the CF card ("local drive"), option 6FC5800-0AP12-0YB0 "additional HMI user memory on CF card" is required. Notice: For a general NCK reset, the option is also deleted and as a consequence, the "local drive" is no longer available until the option is set again. Existing data is kept.
- When operating an NCU without TCU (i.e. only with HMI Advanced or HMI sl on PCU 50.3), the internal HMI sl must be deactivated. This is realized using a service command "sc disable hmi".
- Operator interface software HMI-Advanced 7.6 or the commissioning tool 7.6 is required to commission the drive.
- Only 1 SINUMERIK Operate may be active at 1 NCU - either the internal SINUMERIK Operate or 1 external HMI-Advanced or 1 SINUMERIK Operate for PCU 50.3
  Exceptions: Commissioning / service; HMI-Advanced for the main operator station with permanently assigned MCP and 1 tool loading station with SINUMERIK Operate without MCP.
- The tool manager in the SINUMERIK Operate only supports 1 TO area in the NC.
- The "programGUIDE" for cycles and the contour editor is only released for channel 1.
- Option 6FC5800-0AP17-0YB0 ShopMill/ShopTurn is only released for channel 1.
- Simulation can only simulate NC channel 1.
- After changing the NC configuration or reading-in or downloading an NC series commissioning archive, then an explicit data alignment may be necessary for the simulation. This is initiated in the commissioning / HMI area using the "Simulation Power On" softkey on the 2nd vertical softkey bar.
- Option 6FC5800-0AP22-0YB0 simultaneous recording: The program runtime display can be activated using MD 27860 $MC_PROCESSTIMER_MODE.
- Loadable compile cycles are not backed up using NC series commissioning archive because they are saved on the CF card of the NCU and are kept for a general NC reset. They can be handled the same as HMI configuration data.
- The use of upper case/lower case letters when specifying paths in SELECT instructions of job lists, which refer to the part programs on the CF card or USB stick, is relevant and must be strictly observed.
- Handling special characters in the editor:
  Text files can be edited using the SINUMERIK Operate Editor if the text files use the LF character (0aH)

© Siemens AG 2009
All rights reserved

or the character string CRLF (0d0aH) as line or end of block identifier. The editor cannot open binary files. Files with the following extensions .EXE, .LIB, .ELF, .ARC, .TS, .ZIP, .SO, .PNG, .BMP, .ICO, .CFS, .BIN, .QM, .HMI, .CFG, .ACX, .EMF, .ALM, .ARD, .TGZ, .PTE, .CYC can also not be opened. Files that are newly generated from the SINUMERIK Operate Editor are UTF-8-coded - and have the LF characters as end of block identifier. For UTF-8-coded files, all special characters are correctly displayed. When opening files, the SINUMERIK Operate Editor assumes that the files are UTF-8-coded. If files with another coding are opened, e.g. with Windows page coding, then the special characters are only correctly displayed if the SINUMERIK Operate was changed over to the appropriate system language. This also involves e.g. files that were generated using the HMI-Advanced Editor. When opening with the SINUMERIK Operate Editor, the coding of such files is not changed. There is no automatic conversion into UTF-8-coding. If files were generated or processed with an external editor (e.g. Notepad under Windows) and not the SINUMERIK Operate Editor, then it should be noted that the file is saved, UTF-8-coded. When using Notepad, and when saving the file as encoding select "UTF-8" in the “Save As” dialog box. If special characters are not being used, then "ANSI" can also be specified as encoding.

- For the V24 data transfer, the V24 module 6FC5312-0FA01-0AA0 must be integrated into the NCU. The function is activated in HMI in the file /oem/sinumerik/hmi/cfg/slpmconfig.ini with the entry
  [V24]
  userV24 = true
  A template for this file is provided under /siemens/sinumerik/hmi/template/cfg.
  The V24 softkeys are provided in the ProgramManager under "Archive". The V24 baudrate is limited to max. 19200.

8. Information regarding NCU base software:

- The Linux operating system of the NCU 7x0 is also known as the NCU base software, analog to the base software for the PCU 50.3. The associated documentation is provided on the DVD supplied in IM7. When booting, the NCU base software ensures that the NC, PLC and HMI software start - and if it is being used - that the TCU is also booted. The NCU base software is user-orientated, i.e. you must log-on with a login and password and you are then allocated certain access rights. Presently, this is only relevant for the access to a Linux service shell or for access to the CF card using RCS commander or WinSCP. For commissioning and service, the "manufact" login is used together with the password "SUNRISE" (observe upper and lower case letters!). The Linux service shell is entered at the TCU by simultaneously pressing the "Area changeover" and "Recall" keys (F10 and F9), followed by "Scan for Servers", "2" or by booting the NCU service system from the USB stick.
- After booting the NCU 7x0 from a USB stick with the NCU service system, the "manufact" login and "SUNRISE" password must also be entered assuming that the CF card contains CNC software that can run. If this is not the case (e.g. if the CF card is empty), then in this case, the login is "admin" followed by "SUNRISE" as password.
- Using the command "sc help", in this service shell you obtain a list of the syntax of all of the available service commands.

8. 1 Networking:

- **Please observe the actual documentation "Operator components and networking (IM5) Version 03/2009"!**
- **All of the settings regarding networking and TCU configuration can also be performed using the "System Network Center"; this is available on the PCU 50.3 as well as on the NCU.**
- Important system and network settings of the NCU base software are present in the **basesys.ini** file in the directory /card/user/system/etc. - and can be changed there. The original basesys.ini is located under /card/siemens/system/etc with the name "template-basesys.ini".
  Every NCU in the plant or system network should be allocated a unique ("informative") computer name with the entry "Host name=..." in basesys.ini. Upper case/lower case letters, numbers and minus symbol characters are permitted.
- The DHCP server of the NCU should always be switched-on.
- When changing IP addresses of NCUs / PCUs, the service command "sc clear dhcp" should be run in order that the change becomes effective.

© Siemens AG 2009
All rights reserved

8.1.1 Configuration 1 NCU with TCUs and MCPs

With this configuration, principally, there are no special settings required in basesys.ini. DHCP server and internal HMI-sl remain switched-on.

8.1.2 Configuration 1 NCU with 1 PCU 50.3, MCPs and where applicable TCUs

With this configuration, the following settings are required in the basesys.ini of the NCU:

Host name = ...
SyncModeDHCPD_SysNet = ON_MASTER

Generally, the internal HMI-sl should be switched-off because operating two HMIs (HMI-Advanced and HMI-sl) on one NCU is only permissible in special cases.
We recommend that PCU 50.3 base software from V8.6 and higher is used on the PCU 50.3.
If there is a TCU (HT8) in the system that is to be switched as standard to the PCU 50.3, then the following entries

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

must be entered on the NCU in the file/card/user/common/tcu/<TCU-Nama>/common/tcu/config.ini.

8.1.3 Configuration of several NCUs and where applicable 1 or several PCUs, TCUs, MCPs

With this configuration, the following settings are required in the basesys.ini of the NCU:

unique host name = ... for each NCU
unique InternalIP= ....
same InternalNetMask= ....
precisely one NCU with SyncModeDHCPD_SysNet = ON_MASTER

When first booting after networking, the DHCP master should first boot and then the other stations.
PCU base software from V8.6 and higher is required on the PCU 50.3.
The default configuration "Complete TCUsupport" should be kept in the System Network Center, tab TCU support. "Sync mode low priority" is recommended under the tab DHCP-Settings.
All operator stations in the plant/system network (TCUs, HT8, PCU 50) are managed on the NCU with "ON_MASTER", i.e. the config.ini files - relevant during the runtime - and also the .leases file with all of the IP addresses allocated in the system are located there.
The config.ini files are distributed to all other NCUs / PCUs from the master NCU with the service command "sc distribute tcudata".
With the service command "sc clear dhcp" followed by powering down/powering-up the complete system, the NCUs / PCUs are allocated, their specified IP addresses and these are re-allocated for the TCUs and MCPs; the .leases file is then distributed to all other NCUs / PCUs.
Important: With the System Network Center, tab OPs, the config.ini files of all operator stations can be directly edited on the master NCU on each PCU 50.3. The data is automatically distributed to all NCUs / PCUs with each change.

© Siemens AG 2009
All rights reserved

9. Cycle packages:
The following cycle packages are included in the CNC-SW 2.6 - installed on a CF-card:

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

With SINUMERIK Operate, these packages are in the commissioning area under system data / HMI data / cycle archive.

Upgrading from CNC-SW 1.4 or 1.5 refer to Section 9.6

Important information:
The "programGUIDE" (earlier, cycle support) in SINUMERIK Operate 2.6 is based on these cycle packages 2.6. Cycle calls in the part programs for these cycles cannot be re-compiled and processed with the cycle support in HMI-Advanced 7.x. This also means that the simulation function in HMI-Advanced cannot be used. If the cycle support of HMI-Advanced 7.x is to be used, then the cycle packages 7.x contained in HMI-Advanced - should be downloaded into the NC. This means that no SINUMERIK Operate operation is permissible.

9.1 Technological cycles - function scope
Drilling

Cycle	Function and entry softkeys
CYCLE801	Drilling / positions / grids or frame
CYCLE802	Drilling / positions / any positions
CYCLE81	Drilling / centering
CYCLE82	Drilling / drilling reaming / drilling
CYCLE83	Drilling / deep hole drilling
CYCLE85	Drilling / drilling reaming / reaming
CYCLE86	Drilling / boring
HOLES1	Drilling / positions / series of holes
HOLES2	Drilling / positions / circle of holes
CYCLE840	Thread / tapping with compensating chuck
CYCLE84	Thread / tapping without compensating chuck
CYCLE78	Thread / drill and thread milling

Turning

Cycle	Function and entry softkeys
CYCLE92	Turning / cutting off
CYCLE930	Turning / grooving
CYCLE940	Turning / undercut - Form E, Form F, undercut thread DIN, undercut thread
CYCLE951	Turning / stock removal
CYCLE98	Turning / thread / thread chain
CYCLE99	Drehen / thread - longtitudinal, taper, face
CYCLE62	Contour turning / contour/ contour call
CYCLE952	Contout turning/ stock removal, plunging, plunge turning, all with residual material

Milling

Cycle	Function and entry softkeys
CYCLE60	Milling / engraving
CYCLE61	Milling / face milling

All rights reserved

CYCLE70	Milling / thread milling
CYCLE76	Milling / spigot / square spigot
CYCLE77	Milling / spigot / circular spigot
CYCLE79	Milling / spigot / multi-edge
POCKET3	Milling / pocket / rectangular pocket
POCKET4	Milling / pocket / circular pocket
LONGHOLE	Milling / slot / long hole
SLOT1	Milling / slot / longtitudinal groove
SLOT2	Milling / slot / circular slot
CYCLE899	Milling / slot / open slot
CYCLE72	Contour milling / path
CYCLE62	Contour milling / contour
CYCLE63	Contour milling / pocket, spigot, all with residual material
CYCLE64	Contour milling / pre-drilling
CYCLE800	Various / swivel plane, swivel tool
CYCLE832	Various / >> / HighSpeed settings

Compatibility cycles to 802Dsl:

- These cycles can be re-compiled and changed in screen forms

Cycle	Function
CYCLE71	Compatibility 802Dsl – face milling
CYCLE87	Compatibility 802Dsl – boring 3
CYCLE88	Compatibility 802Dsl – drilling with stop
CYCLE89	Compatibility 802Dsl – boring 5
CYCLE90	Compatibility 802Dsl – thread milling
CYCLE93	Compatibility 802Dsl – groove
CYCLE94	Compatibility 802Dsl – groove, form E and F
CYCLE95	Compatibility stock removal (re SW 2.5 and 802Dsl)
CYCLE96	Compatibility 802Dsl – thread undercut forms A, B, C, D
CYCLE97	Compatibility 802Dsl – thread cutting

Other cycles - internal:

Cycle	Function
CYCLE861	Reverse countersink only for ISO compatibility, no input mask
CYCLE202	Help cycle for retraction
CYCLE203	Help cycle for chamfering
CYCLE204	Help cycle, calculating pitch for metric threads
CYCLE206	Help cycle, tool change in JOG
PROG_EVENT	ASUB to support special operating modes
CUST_800	Manufacturer cycle for swivel (earlier TOOLCARR)
CUST_832	Manufacturer cycle for HighSpeed settings (earlier CYC_832T)
CUST_TECHCYC	Manufacturer cycle for technological cycles (previously ST_CUST)

Important new functions with respect to 840Dsl SW 1.5 (cycles SW 7.5) – details, refer to /1/:

General

- The configuration of cycle functions or the input screen forms for cycles is made in the configurable machine or setting data. However, part programs, which use the old setting data in the GUDs (field _ZSD[x]), can still run.

Drilling

- Support drilling cycles, select depth referred to shaft/drill tip

© Siemens AG 2009
All rights reserved

- New drilling pattern frame
- New cycle, drill and thread milling (CYCLE78)

Turning
- Contour transfer using CYCLE62 when machining (stock removal)
- New cycle, cutting off
- Extended functionality, grooving
- Extended functionality, undercut
- Extended functionality, thread cutting (tapping)
- New cycle, contour turning (CYCLE952) with the following functions:
  - Stock removal
  - Stock removal, residual material
  - Plunging (contour plunging)
  - Plunging, residual material
  - Plunge turning
  - Plunge turning, residual material

Milling
- Contour transfer using CYCLE62 for path milling
- Extended functionality, face milling (edge limit)
- Extended functionality, thread milling
- New cycle, multi-edge
- Milling cycles can be executed at individual positions or a pattern of positions
- Milling cycles have the new chamfering function
- New cycles, contour milling (CYCLE63) and pre-drilling (CYCLE64) with the following functions:
  - Contour pocket with or without islands (presently, a maximum of 10 islands is possible)
  - Contour pocket, residual material
  - Contour spigots
  - Contour spigots, residual material
  - Pre-drilling

List of differences, cycles/cycle functions for 840D sl SW 7.5:
The following cycles/cycle functions are not included in this cycle package:

Cycle	Function and entry softkeys
POCKET1 POCKET2	Rectangular pocket / circular pocket (replaced by POCKET3 / POCKET4)

Supplementary conditions / restrictions:
- The number of islands for contour pockets is limited to 10.

Special notes:
- Activate the cycle supply, the machine data
  - MD 52200 MCS_TECHNOLOGY > 0 (1 = turning or 2 = milling)

must be set.
- For turning, it is possible to set milling in machine data
  - MD 52201 MCS_TECHNOLOGY_EXTENSION = 2

as extension technology.
- There is no display and no possibility of entering data for TOOLCARR data. This can only be set using the NC program.

9.2 Measuring cycles - functional scope
Measuring in JOG
The measuring cycles support the following set-up functions in JOG (SK "zero point workpiece"):

All rights reserved

- Calibrate key (radius, length)
- Set edge
- Align edge
- Measure corner (3 points, right angled corner)
- Measure 1 hole
- Measure 1 circular spigot

New from SW 2.6 (here, option "Extended operator functions" is required, for 840D sl, already included in the basic version)

- Clearance, 2 edges
- Measure corner (4 points)
- Measure 2 or 3 or 4 holes
- Measure rectangular spigot
- Measure 2 or 3 or 4 circular spigots
- Align plane

The functions "zero point workpiece" are only available in milling technology.

The following measuring functions to measure tools in JOG are available (SK "measure tool"):

- Calibrate tool probe
- Measure tool length
- Measure tool radius

Measuring in the automatic program

- Workpiece measurement, milling machines with the following functions:
  - Calibrate workpiece probe in a hole or at a surface (CYCLE976)
  - Measure hole, spigot, groove (slot), rib, rectangular pocket, rectangular spigot in parallel with the axis (CYCLE977)
  - Measure edge (CYCLE978)
  - Measure segment of a circle, outer or inner (CYCLE979)
  - Measure slot or rib under an angle (CYCLE979)
  - Measure corner in the automatic program (CYCLE961)
  - Measure angle in the automatic program (CYCLE998)
  - Measure kinematics (CYCLE996)
- Tool measurement, milling machines using the following functions:
  - Measure tool in the automatic program (CYCLE961)
- Workpiece measurement, lathes using the following functions:
  - Calibrating a workpiece probe in either the slot or at the surface (CYCLE976)
  - Single point measurement (CYCLE974)
  - Two point measurement (CYCLE994)
- Tool measurement, lathes using the following functions:
  - Measuring tools in the automatic program (CYCLE982)
- General measuring cycle functions:
  - Measuring cycle support in the editor
  - Measuring result diagram display
  - Optional correction of the measuring difference in a WO or a WO data set or only measuring
  - Use of either a monoprobe or multiprobe

Prerequisites for use

- The measuring cycles require a switching, electrical probe.
- Corresponding to the pre-assignment of the machine data for measuring cycles, the workpiece probe should be connected at the first measuring input of the control (these settings can be changed using the machine data

© Siemens AG 2009
All rights reserved

MD 52600 MCS_MEA_INPUT_PIECE_PROBE and
MD 52601 MCS_MEA_INPUT_TOOL_PROBE)

Important new functions with respect to 840Dsl SW 1.5 (measuring cycles SW 7.5)

General
- All measuring cycle data that are important for the machine manufacturer (OEM) and user are now in the configurable machine and setting data (e.g. data fields of the calibration values). The GUD blocks GUD5, GUD6 and GUD7_MC are no longer required.

List of differences, measuring cycles/cycle functions for 840D sl SW 7.5:
The following cycles/cycle functions are not included in this cycle package:
- Logging measuring results

Special notes:
- The following two setting data must be carefully observed regarding the functionality of making measurements in JOG:
  SD 54798 SCS_J_MEA_FUNCTION_MASK_PIECE
  SD 54799 SCS_J_MEA_FUNCTION_MASK_TOOL

9.3 Cycles ISO compatibility – functional scope

The functional scope of cycles for ISO compatibility corresponds to those of SW 6.5 for SINUMERIK 840D. The two ISO dialects are still supported.

The setting data of the cycles for ISO compatibility has been changed over to the configured machine and setting data. The data fields for the setting in the GUDs are no longer relevant.

9.4 Configurable machine and setting data – function scope

New configurable machine and setting data have been introduced from SW 2.5. When the control starts (cold start) these are automatically read-in from the CF card and activated.
- This data replaces the GUD variable of the cycle packages and the display MDs of the JobShop and HMI.
- Just like all other machine and setting data, they are displayed in the "Commissioning" area (depending on the protection stage) and are located in the range of numbers starting 50000 and higher.
  51xxx General MD
  52xxx Channel MD
  53xxx Axis MD
  54xxx General SD
  55xxx Channel SD
  56xxx Axis SD
- The names of the data provide information about their significance and use; the appropriate codes are located directly after the prefix, e.g. DISP for display, ISO for ISO cycles, MEA for measuring cycles, TURN for turning, etc.
- The configurable machine and setting data support the data class concept for data backup.

9.5 Compatibility to cycle packages 840D sl to SW 1.5

The behavior of the cycles with respect to software releases up to SW 1.5 has been changed regarding the following points:

Technological cycles:
1. Milling cycles - calculating the infeed depth with / without taking into account the safety clearance (previously this was able to be set in the GUD variables _ZSD[1]):
   In setting date $SCS_FUNCTION_MASK_MILL_SET bit2 it is possible to set whether the safety clearance should be included when calculating the depth - or not. Previously, "taking into account the safety clearance" was set as the default setting – now "without taking into account the safety clearance" is the default setting.

© Siemens AG 2009
All rights reserved

This means that the individual depth infeeds can change in existing programs.
The setting of this setting data acts in the same way in the G code programs and ShopMill or ShopTurn programs.

2. Milling cycles when calling with MCALL:
   Milling cycles in the G code programs can be optionally called also to position patterns with MCALL.
3. Thread drilling cycles CYCLE84, CYCLE840 – technology parameters (previously were able to be set in GUD variables):
   The machinery construction OEM must enter the technology parameter settings he requires in the corresponding cycle setting data. These settings are neither changed by the cycle nor by the input screen form.
4. Thread drilling cycle CYCLE840 – parameter to reverse the direction:
   The "Direction of rotation for retraction" parameter is no longer supported in the input screen form of the cycle. This makes sense, as in the meantime, the cycle has this function for both thread types and also only functions if a spindle direction has been programmed before being called.
5. Pocket milling cycles POCKET3 and POCKET4 – parameters for plunge feeding:
   The programmed feed for the infeed depth is only effective for "pre-drilled" and "vertical". "Helical" and "oscillating" plunging is realized with the feed for machining in the plane.
6. Milling cycle, rectangular pocket POCKET3 – reference, angle of rotation of the pocket:
   The angle of rotation now always acts at the reference point. The programming for rectangular pocket "reference point is a corner" and "angle of rotation refers to the center" (with ZSD[2]=1 and ZSD[9]=1) are no longer available, but lead to Alarm 61109 "Parameter _STA incorrectly defined".
7. Milling cycles – parameter for the infeed width:
   For the infeed width, a value > 0 must always been entered. In the screen form, the field is marked as incorrect and the cycle issues an alarm.
8. Drilling and milling cycles – safety clearance parameters:
   A value > 0 must always be entered for the safety margin as this value is also effective for the changeover from G0 to G1 when approaching to machine in the tool axis. The field is marked as incorrect in the screen form.
9. High Speed settings CYCLE832: The differences are explained using the documentation:
   Documentation up to SW 7.5: Programming Manual, Cycles [1]
   Documentation from SW 2.6: Commissioning Instructions IHsl IM9 [2]
   Operating Instructions, Milling [3] or Operating Instructions, Turning [4]
   - All functions documented in [1] "Point 3.17" that are coded in parameter _TOLM - with the exception of the machining type (_TOLM ones location) will, from SW release 2.6, only be supported in the compatibility mode.
   - All GUDs described in [1] Point "3.17.4.3 Adaptation of the machinery manufacturer" and the documented functionality are no longer available from SW release 2.6 onwards.
   - Refer to [1] "Point 3.17.4.4 Adaptation of additional program parameters CYC_832T"
     From SW release 2.6 and higher, the user cycle CYC_832T.SPF will be replaced by cycle CUST_832.SPF. For compatibility, the marks _M0 to _M4 are still available in the CUST_832 cycle. The transfer parameter _OVL_on of CYC_832T has been eliminated as the documented machine data cannot be re-written in CYCLE832.
   - Note to [1] Point "3.17.5 Interfaces"
     All documented machine and setting data are not rewritten in CYCLE832 from SW release 2.6 and higher. All documented channel-specific variables GUD7 have been eliminated from SW release 2.6 and higher.
10. Swivel cycle CYCLE800: (reference to documents as in Point 8 for CYCLE832):
    - The menu screens, documented in [1] under point "3.16.7.2 Commissioning kinematic chain" to commission swivel, have been eliminated in SW release 2.6. The parameters (vectors) of the kinematic chain can be read and written to as NC variable -> refer to [2] CYCLE800.
      Example: $TC_CARR1[2]=123.456 -> linear vector I1X of the 2nd swivel data set = 123.456
    - The coding of parameter $TC_CARR37[swivel data set n] has changed:
      The setting "Direction rotary axis 1" or "Direction rotary axis 2" in the fourth position (THOUSANDS) $TC_CARR37[n] are no longer permitted from SW release and must be replaced by the setting "direction rotary axis 1 optimized" or "direction rotary axis 2 optimized". This means that the pole position of a rotary axis is supported with both selection options of the direction (plus or minus).

© Siemens AG 2009
All rights reserved

When coding parameter _MODE and _DIR in the call interface of CYCLE800, the following definition applies:
If an "old" program call is accepted, the direction setting is accepted, optimized in the HUNDRED THOUSANDS position of the the parameter _MODE:
_DIR = -1 _MODE = 10xxxx swivel yes, direction minus
_DIR = +1 _MODE = 20xxxx swivel yes, direction plus
_DIR = 0 _MODE = 11xxxx swivel no, direction minus
_DIR = 0 _MODE = 22xxxx swivel no, direction plus
If the setting of the direction is changed in $TC_CARR37[n] when upgrading to software release 2.6, then the corresponding NC programs must be run-in again.
This is especially true for programs where, when swiveling, a rotary axis is in the pole position, e.g. swivel table, rotary axis A=0 (pole position) swivel with rotary axis C.

- The coding, retract in the 7th and 8th position of $TC_CARR37[n] x00xxxxxx up to x03xxxxxx has been recoded from SW release 2.6 and higher.
  up to SW 2.6 from SW 2.6
  $TC_CARR37[n] x00xxxxxx x01xxxxxx retract Z
  $TC_CARR37[n] x01xxxxxx x03xxxxxx retract Z or Z, XY
  $TC_CARR37[n] x02xxxxxx x02xxxxxx retract Z, XY
  $TC_CARR37[n] x03xxxxxx x00xxxxxx no retraction
- From SW release 2.6, the swivel data sets must be designated as either being active or inactive. This is realized in parameter $TC_CARR37[n] in the 9th position (HUNDRED MILLION).
  $TC_CARR37[n] 0xxxxxxxx to 3xxxxxxxx means swivel data set not active
  $TC_CARR37[n] 4xxxxxxxx to 7xxxxxxxx means swivel data set active
  The parameters $TC_CARR38[n] up to $TC_CARR40[n] are converted into the actual dimension system (inch, metric) by the NCK from SW release 75 and higher. If the parameters are to be written in (customers) user cycles, then it is absolutely necessary that this is taken into account when upgrading to SW release 2.6 in conjunction with NCK from 75 and higher.
  NC programs with the data of the swivel data set can be reused with software release 2.6 if parameter $TC_CARR37[n] is appropriately adapted.
- Note to [1] Point "3.16.8 Manufacturer's cycle TOOLCARR.SPF"
  The user cycle TOOLCARR.SPF is replaced, from SW release 2.6 and higher, by the cycle CUST_800.SPF. From a functional perspective, cycle CUST_800.SPF corresponds to the cycle TOOLCARR.SPF, including all of the marks prepared for machinery construction OEMs for the appropriate machine-specific adaptations -> refer to the comments in CUST_800.
  If the "track tool" function has been activated, then the first multi-axis transformation (TRAORI(1)) becomes active in cycle CUST_800.SPF when tracking tool.
  The mark _M35 in CUST_800.SPF is new from software release 2.6 and higher. This supports semi-automatic and manual rotary axes in the block search. From a contents perspective, Mark _M35 replaces cycle CYCPE_SC.SPF, which has been eliminated from software release 2.6 and higher.

These changes may require that existing programs are adapted.

Measuring cycles:

1. The data concept of the measuring cycles was revised with the introduction of new machine and setting data. Setting data, which were previously stored in GUD variables, have now become machine and setting data.
   Attachment A of document /5/ contains tables with the corresponding information about:
   - Comparison of GUD parameters ⇔ machine and setting data
   - GUD variables that are no longer used
   - Changes to the names of cycles and GUD blocks.
2. When using the measuring cycles with different dimension systems (base system <> programmed system) and when programming G commands G70 or G71, the dimension units of the following tolerance parameters have changed. They now refer to the programmed system (previously, basic system): Confidence range (_TSA), zero correction range (_TZL), average value generation with correction (_TMV), dimension difference monitoring (_TDIF).

These changes may require that existing programs are adapted.

© Siemens AG 2009
All rights reserved

ShopMill cycles:

1. Up until SW 1.5, it was also possible to specify the reference point (X0, Y0, Z0) for ShopMill in incremental terms. In the new opened screen forms, now the reference point can only be entered in absolute terms. For re-compiled screen forms, in which an incremental reference point occurs, a toggle field abs/inc is displayed. For inc, the check issues an error message “reference point inc no longer supported, please convert to abs".

9.6 Upgrading the cycles from CNC-SW 1.4 / 1.5

Initial situation:
840D sl with SW V1.4 or V1.5.
These releases include technological cycles V7.3 or V7.5 and measuring cycles V7.3 or V7.5.

Objective of the upgrade:
840D sl with SW V2.6.
This release includes technological cycles V2.6 and measuring cycles V2.6. These cycles are not identical to the cycles from V7.3 or V7.5 - however, they are compatible from a functional perspective. The cycles are always upwards compatible, not downwards compatible. The difference is in the cycle support and the configuration of the cycle functions or the input screen forms are made in the configurable machine or setting data. Part programs, which use the old settings in the GUD parameters, can still run.

Upgrade steps:

1. Back-up the data by reading-out the series commissioning archive for the NC, PLC and drive.
2. Install the CNC-SW V2.6 on CF card (sc restore **-full** ...).
   To do this, a general reset of the NCU and PLC is required: SIM/NCK switch in position 1, PLC switch in position 3 and then power down/power up.
3. Read-in (download) the backed-up series commissioning archive for NC, PLC and drive.
4. Using the program upcycdat.mpf write GUD values to the cycle machine and setting data. The program can be downloaded from eSupport. It is located here under:
   -> Automatisierungssysteme Engineering, Service-Tools [Automation systems Engineering, Service tools]
   -> NC-Engineeringtools [NC engineering tools]
   -> Zyklen-Tools [Cycle tools]
   "Hochrüstung Zyklendaten für 840D sl" ["Upgrade cycle data for 840D sl"]
   The archive file should be read into the NC, whereby the program upcycdat.mpf is saved in the workpiece "TEMP".
   When using the cycles in several channels, the program should be started in each of these channels. After this, the upcycdat.mpf program on the NC should be deleted.
5. Delete the NC data/definitions GUD7_SC, GUD7_MC, GUD7, GUD5, GUD6, SMAC_SC, SMAC. The files can be found under commissioning / system data.
6. Copy the CUST_TECHCYC.SPF and CUST_MEACYC.SPF - that if necessary should be adapted by the machinery construction OEM - from the CF card/Siemens/ Sinumerik/cycles/sc/prog or CF card/Siemens/ Sinumerik/cycles/mc/prog to NC data/cycles/manufacturer cycles (or -/user cycles). The access is realized via Commissioning / system data.
7. Cycle CUST_MEACYC.SPF replaces the CYCLE198 and CYCLE199 cycles from cycle release V07.xx.xx. If changes were made in these cycles, then these changes should also be made in the cycle CUST_MEACYC.SPF. After this has been done, cycles CYCLE198.SPF and CYCLE199.SPF can be deleted.
8. Delete the technological and measuring cycles in the path NC data/cycles/standard cycles. The files can also be found under commissioning / system data.
9. Power down/power up the control.
10. Load the cycles and the definition files from the CF card into the NC. This is realized again under Commissioning / system data - and read-in the archives cycles.arc and defines.arc under the path CF-Card/Siemens/ Sinumerik/cycles/sc/prog - and the archive mcycles.arc under the path CF-Card/ Siemens/Sinumerik/cycles/mc/prog.

© Siemens AG 2009
All rights reserved

11. To activate the cycle support, the following machine data
    - MD 52200 MCS_TECHNOLOGY
    - MD 52201 MCS_TECHNOLOGY_EXTENSION

must be set. The value "1" stands for turning and the value "2" for milling
12. Power down/power up the control.

© Siemens AG 2009
All rights reserved