SINUMERIK 840DE SL CNC SOFTWARE 6-3 WITH SINUMERIK OPERATE 6 LANGUAGES (EN,DE,FR,IT,SP,CHS); EXPORT SOFTWARE 2.7 SP2 ON CF CARD WITH LICENSE	6FC5850-1YG23-2YA0
SINUMERIK 840D SL CNC SOFTWARE 6-3 WITH SINUMERIK OPERATE 6 LANGUAGES (EN,DE,FR,IT,SP,CHS); SOFTWARE 2.7 SP2 ON CF CARD WITH LICENSE	6FC5850-1XG23-2YA0
SINUMERIK 840DE SL CNC SOFTWARE 31-5 WITH SINUMERIK OPERATE 6 LANGUAGES (EN,DE,FR,IT,SP,CHS); EXPORT SOFTWARE 2.7 SP2 ON CF CARD WITH LICENSE	6FC5850-3YG23-2YA0
SINUMERIK 840D SL CNC SOFTWARE 31-5 WITH SINUMERIK OPERATE 6 LANGUAGES (EN,DE,FR,IT,SP,CHS); SOFTWARE 2.7 SP2 ON CF CARD WITH LICENSE	6FC5850-3XG23-2YA0

Installation and operating notes

The CNC Software 2.7 SP2can only be operated on SINUMERIK NCUs 7x0.2 with MLBF ...-_AA2.

The CNC Software 2.7 SP2 (internal version 02.07.02.00.009)
includes the following components:

Component	Version
NCK	V83.04.06
SNCK	V02.06.00.00.001
SINAMICS	V02.60.54.00.001
PLC Op. Sys.	V20.70.45 and V27.90.14 (for 319)
FB15(sl)	V02.07.13
CP	V02.12.00
MCP CLIENT	V01.04.38
SINUMERIK Operate	V04.04.02.00.013
Linux basic system	V04.40.40.00
NCK files ystem driver	V04.04.00.00.004
Profinet FW	04.01.33.51
Cycles	V04.04.23.01
SNC	V01.05.00.00.002

Functional improvements in NCK SW 2.7 SP2 compared to SW 2.7 SP1:

PR number	AP01308507	AP01299881	AP01294647	AP01294273
AP01290468	AP01288622	AP01286754	AP01286259	AP01285752
AP01285701	AP01279051	AP01275311	AP01273519	AP01272690
AP01272640	AP01265438	AP01263880	AP01262737	AP01262332

AP01262326	AP01261452	AP01255196	AP01254190	AP01253582
AP01252296	AP01251098	AP01249135	AP01246929	AP01242550
AP01239037	AP01238335	AP01238168	AP01237304	AP01233063
AP01232442	AP01231593	AP01231346	AP01230144	AP01226569
AP01222588	AP01221972	AP01217097	AP01216816	AP01216752
AP01214876	AP01213751	AP01213628	AP01210027	AP01205113
AP01203908	AP01203808	AP01203239	AP01202893	AP01199448
AP01195627	AP01195627	AP01194644	AP01193953	AP01190296
AP01190002	AP01189158	AP01183680	AP01179718	AP01179166
AP01176397	AP01172012	AP01171649	AP01161598	AP01149887
AP01149887	AP01145253	AP01100950	AP01100950	

Functional improvements in Operate V2.7 SP2 compared to V2.7 SP1:

PR number	AP01308596	AP01272452	AP01261065	AP01276107
AP01276344	AP01305123	AP00933238	AP00932460	AP01255860
AP01198681	AP01243393	AP01259817	AP01275210	AP01227679
AP01224484	AP01100950	AP01220163	AP01221865	AP01237261
AP01214120	AP01212189	AP01259976	AP01288110	AP01017172
AP01228166	AP01252364	AP01249724	AP00987008	AP01292479
AP01304350	AP01232001	AP01224312	AP01275957	AP01290596
AP01132693	AP01229066	AP01228157	AP01272240	AP01203286
AP01194909	AP01239639	AP01219901	AP01278907	AP01237597
AP01240990	AP01251106	AP01114681	AP01248804	AP01253536
AP01253819	AP01265817	AP01292198	AP01300463	AP01228180
AP01278488	AP01281937	AP01283538	AP01294550	AP00840691
AP01238974	AP01241067	AP01286364	AP01247310	AP01248701
AP01257071	AP01194329	AP01276862	AP01209360	AP01223489
AP01222624	AP01246235	AP01246437	AP01261307	AP01274128
AP01278746	AP01294520	AP01300974	AP01228682	AP01224468
AP01274551	AP01150786	AP01205876	AP01236776	AP01285128
AP01291364	AP01261630	AP01295132		

Functional improvements in the Siemens cycles V2.7 SP2 compared to V2.7 SP1:

PR number	AP01187683	AP01188504	AP01200279	AP01201698
AP01203208	AP01203208	AP01203944	AP01203948	AP01206495
AP01211766	AP01215314	AP01217565	AP01219363	AP01226616
AP01226763	AP01229813	AP01230121	AP01231416	AP01239425
AP01239623	AP01242809	AP01245205	AP01247925	AP01252661
AP01254032	AP01254287	AP01255441	AP01259279	AP01259482
AP01264377	AP01269822	AP01276717	AP01276733	AP01277506
AP01279736	AP01280025	AP01280517	AP01281924	AP01282852
AP01283506	AP01284167	AP01285009	AP01285188	AP01290428
AP01292614	AP01293242	AP01296336	AP01297738	AP01298066
AP01304792				

Installation of the CNC software:
The NCU Service System V04.40.35 (or later) is required on USB stick to install the CNC software.

New installations can be performed via the NCU service system and TCU or with WinSCP / RCS-Commander.

Installation of the NCU service system on USB stick:
The NCU service system is stored as USB stick image in the directory emergency_bootsys_ncu of the DVD delivered. The copy program installdisk.exe is also included. Connect a USB stick >= 2 GB to your PG resp. PC (with Windows XP) and determine the relevant drive letter. Call up installdisk to copy the NCU service system onto the USB stick:
installdisk --verbose --blocksize 1m <image file> <drive letter:>
We recommend that you execute this command in a DOS shell.
For this, you require administrator rights on your PG/PC.
Upon completion of installdisk, remove the USB stick from the PG / PC, boot once an NCU 7x0 from this stick (this boot process takes a little bit longer; a "P" in the 7-segment display of the NCU indicates that the FAT partition of the USB stick is partitioned) and reinsert the stick in the PG / PC. Windows now displays an empty USB stick. Copy the requested CNC software (file with the extension .tgz) from the DVD delivered, the directory ncu_sw.
The USB stick can be inserted in one of the two USB plugs of the NCU 7x0.2. After being switched on, the NCU is booted from this USB stick. The system is operated either via a TCU which is connected to the NCU or via PG / PC using WinSCP under "Open Terminal".
The use of the NCU service system is described in the documentation /IM7/ on the DVD delivered.
We recommend that you use the USB stick "SIMATIC IPC USB Flashdrive, 6ES7648-0DC50-0AA0".

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

1.2 Upgrading an NCU 7x0.2 with CNC-SW 2.6 SP1 or 2.7

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

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. Notes:

- After installing the CNC Software Version SW 2.7 SP2on 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.
- The NCU does not boot if either the fan module is not available or if it is defective (both fans fail).
  The 7-segment display shows "8", all LEDs are on.
- 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 transferred to the PLC input image

  Example of the 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 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: The upper key row is displayed in the PLC as follows:
  The key on the top left (SK1) is mapped in EBn+2 Bit 1
  SK2 = EBn+2 Bit 6
  SK3 = EBn+3 Bit 7
  SK4 = EBn+4 Bit 4

  The LEDs of the upper key row can be activated by the PLC as follows:
  Signals to handheld unit ABn+0 Bit 0 correspond to LED top-left
  Bit 1 for 2nd LED from top left, etc.

- 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
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
- 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.
- 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 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 NCU710.2, 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.
  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. 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
  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]).
- In order to have sufficient reserves for program processing, the maximal load with typical user operations should not exceed 75%. The current average load should not exceed 50%. The current load can be checked under Diagnosis / System resources.
- 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.
- The "Parking axis" function with an SMI encoder is not possible without power off/on.
- 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.
- Alarm 8025: Advanced Surface 6FC5800-0AS07-0YB0 is an option and is interlocked using an option data.
- 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.
- 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 2.7 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.
- The Software Version 2.7 SP1 requires the PLC Toolbox V02.07.03 of the 2.7 Software Version V02.07.05 (or later).
- The NCU Link has not been released in combination with safety axes.
- The behavior of the NCK with regard to the dynamic offset values ACC, ACCLIMA, VELOLIM, VELOLIMA, JERKLIM, JERKLIMA has changed. The machine data description shows the individual effect.

- Archives with MD 19730.11 = false are incompatible. The machine data 19730.11 (hmi_function_mask) must always be set.
- The machine data 18150 must be increased when defining more than two channels. Archives with 18150=150 (standard) may be incompatible.
- Archives with MD 32250 > 0 are incompatible. With MD32250 > 0, speed setpoint or torque setpoint standardization is no longer transferred from the drive.
- The interface signal "Invert M3/M4" now also acts when tapping with G331/G332.
  In applications in which the interface signal is always set to "1", the spindle is now rotated in the incorrect direction with G331/G332. To achieve a behavior that is compatible with earlier versions, the bit 22 in MD35035 must be set to "1" (the standard default is 0).
- Archives with MD28253=100 (old default value) may be incompatible. Change the MD into 200 (new default).
- Archives with MD19730[0] =0 are incompatible. Softkeys are missing in MDA. Change the MD into 804hex.
- Due to an implementation error, Spline Interpolation could be released without a license alarm. If this feature has been used, you require a new license key.
- So far, the gear stage 1 could be preset with 0 and 1 in the DB[axis].DBB16. Now, a gear stage change is always requested when specifying 0 in the data byte 16. Upon acknowledgement of the gear stage change with "0", the Alarm 22010 "Actual gear stage does not correspond to the setpoint gear stage" is output. It may be necessary to adapt the user PLC program.
- The function TANGON with additional parameter "P" has not been released. Remedy: Use the standard setting "S".
- The Siemens cycles are loaded upon each startup from the CF card to the CPU. This behavior can be switched off / on by using the service commands sc disable s_cycles / sc enable s_cycles and performing a subsequent general NC reset.
- The contents of the Siemens GUD variables (SGUD) have been relocated in PGUD. To be able to load archives comprising SGUD, these must be processed. Otherwise, the contents of the SGUD and PGUD variables may be provided twice.
- Grinding technology:
  The new function "Calculate grinding wheel peripheral speed in diameter" has not been released due to AP01041616.
- The SD43235 has a speed imit of 10,000 rpm as default value. When upgrading to software version NCK 78.00 or later and spindle speeds exceeding 10,000 rpm, the SD43235 must be increased to the corresponding value. The speed is limited by SD43235 if the system variable $AC_SMAXVELO_INFO[n] has the value 21.
- Multitool as manual tool:
  With repeated selection (of tools within this manual multitool), in the sequence
  T="x" M06 --> T0 M06 --> T="Y" M06 --> T0 M06, the identifier “manual tool“ is not set upon the second tool selection. As a result, the system tries to store the multitool in the magazine with the next T0 M06.
- When loading the topology from a Sinumerik 840D sl with NCU730.2 PN (0AA2) into an empty Starter project, errors occur during uploading.
  The online access to the CU_INT functions properly.
- The function N<Nx has not been released.
- Safety
  When upgrading the system from SW 2.6 to 2.7, the following checksum or DCC alarms may occur:
  New machine data / confirmation of the axial checksums requested
  To ensure the future functionality, machine data are already included in NCK 83.02.
  As these machine data are also included in the checksum, you have to confirm the axis-specific checksums when upgrading to the Version 2.7.

  Alarm 27001 Code 231
  Further, the alarm 27001 with code 231 (resp. alarm 201711 with code 232) may occur on rotary axes on account of this new machine data.
  This results from the different values entered in the machine data 36947, $MA_SAFE_VELO_X_HYSTERESIS resp. p9547. The standard value "10" for the parameter p9547 has been entered in the drive; for NCK, this value has been converted to the rotary axis.
  To prevent this alarm during startup, this machine data must be adapted. This is possible either via

the HMI function "Copy SI data " or by manually entering "10" in the machine data 36947 $MA_SAFE_VELO_X_HYSTERESIS.
The machine data 36947, $MA_SAFE_VELO_X_HYSTERESIS, is reserved for future functions. At present, the value entered is not evaluated.
Alarm 27071 SPL parameterization checksum error
To facilitate handling when testing the valid language scope of the program SAFE.SP, the bit 2 of the machine data $MN_SAFE_MODE_MASK has been removed from checksum calculation. This allows to perform the test in the future without confirming the checksum.
On account of checksum recalculation without this bit, the checksum must be confirmed once.

- The configuration comprises one or several DQI encoders. During commissioning, the Alarm 201670 with identifier "6" is output.
  Remedy:
  Set the bit 0 in the drive parameter p9515 (incrementer). Further, check the parameter values p9523=r470, p9524=r471, p9525=r472 and p9529=r475. If necessary, adjust the parameters 95xx according to r47x.
- When working with multitool and multitool location OEM data, you must change the data types ($MN_MM_TYPE_CC_MULTITOOL_PARAM and $MN_MM_TYPE_CC_MTLOC_PARAM) from default=3 (integer) to the value=4 (real).
- When taking data backups from NCU versions lower than Software Version 4.4, the machine data 10185 NCK_PCOS_TIME_RATIO should be checked. The setting of NCK_PCOS_TIME_RATIO should be as follows:
  - With NCU 7x0.1 and 7x0.2: 65% with internal HMI and 90% with external HMI
  - With NCU 7x0.3: 90%
- **Activation of the function "Flat D number"**
  The following machine data must be set to activate the function "Flat D number".

  MD18102 $MN_MM_TYPE_OF_CUTTING_EDGE = 1
  MD18082 $MN_MM_NUM_TOOL >= 1 (or keep default value / the value of zero is **not** permissible)

  With MD18102 = 1 and MD18082 = 0, you must first deactivate MD18102 (zero and NCK reset).
  Then, you must set both machine data and activate them **at the same time**.
  This can also be achieved by creating a data backup with the modified settings and reading this backup in again.
  Note: The machine data are memory configuring.
- The NC machine data 28070 NUM_BLOCKS_IN_PREP should be set to a value grater than or equal to 80.
- New mode:
  The following new bit constellation is supported:
  MD 18080 $MN_MM_TOOL_MANAGEMENT_MASK: Bit0=1, Bit1=0
  MD 20310 $MC_TOOL_MANAGEMENT_MASK: Bit0=0, Bit1=0
  For this, make the following entry in the file sltm.xml:

```
<CONFIGURATION>
    …
  <Settings>
    <RefinedToolmanagementMaskHandling value="true" type="bool"/>
  </Settings>
</CONFIGURATION>
```

  The modified file sltm.xml must be stored in the directory /oem/sinumerik/hmi/cfg or /user/sinumerik/hmi/cfg.

  This bit constellation permits programming in the form “T=TNumber“ in NC programs.

- **Fading out the tool management**
  You can deactivate, that means fade out, the tool management in the Parameter operator area. For this,

change the entry DLG002 in the file systemconfiguration.ini in the paragraph [dialogs] as follows:
[dialogs]
…
DLG002= name:=SlParameter, **implementation:=slpadialog.SlPaDialog**, process:=SlHmiHost1, preload:=true, terminate:=false, cmdline:="-conf slpadialog.hmi"
…
In this case, the softkey “Zero offset“ is selected when selecting the Parameter operating area for the first time.
The modified file systemconfiguration.ini must be stored in the directory /oem/sinumerik/hmi/cfg or /user/sinumerik/hmi/cfg.

- **Sag compensations (CEC)**
  If sag compensation is activated, you must check the setting data
  41300[nn] $SN_CEC_TABLE_ENABLE after reading in the data.

5. SINUMERIK Operate

5. 1 Notes regarding SINUMERIK Operate:

- The SINUMERIK Operate 2.7 SPx software is automatically started as a component of the CNC software 2.7 SPx when NCU 7x0.2 is started. SINUMERIK Operate is also called "internal SINUMERIK Operate" (contrary to the "external SINUMERIK Operate" on the PCU 50).
- 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 operating an NCU without TCU (i.e. only with HMI Advanced or HMI sl on PCU 50), the internal HMI sl must be deactivated. This is realized using a service command "sc disable hmi".
- Operator interface software HMI-Advanced 7.6 SP1 (or later) or the commissioning tool 7.6 SP1 (or later) 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 / PC.
  Exceptions: Commissioning / service; HMI-Advanced for the main operator station with permanently assigned MCP and 1 tool loading station with SINUMERIK Operate without MCP.
- 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) 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 neither 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, a V24 module must be integrated into the NCU.
  Notice: Different V24 modules are required for the NCUs 7x0.2 and 7x0.3.
  The V24 baudrate is limited to max. 19200.
- ShopMill / ShopTurn:
  The function "Block search upon selection / processing from the editor " should be deactivated via the NC MD 51040.3 = 1 ($MNS_SWITCH_TO_MASCHINE_MASK Bit 3) because otherwise a malfunction of the operator interface may result.
- The graphic tool display can only be deactivated (MD=0mm) resp. activated (MD>0mm) via the MD 52271 TM_MAG_PLACE_DISTANCE.

6. Information regarding the 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. 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 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.

6. 1 Networking:

- **Please observe the current documentation "Operator components and networking (IM5) Version 02/2012!**
- **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 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.

6.1.1 Configuration of one NCU with TCUs and MCPs

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

6.1.2 Configuration of one NCU with one PCU 50, 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 Operate) on one NCU is only permissible in special cases.
We recommend that PCU 50.3 base software from V8.6 SP1 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.

6.1.3 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 SP1 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. The data is automatically distributed to all NCUs / PCUs with each change.

7. Cycle packages:

In the CNC Software Version 2.7, the following cycle packages are included on the CF card:

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

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

Important note:
The “programGUIDE“ (previousy cycle support) in SINUMERIK Operate is based on the cycle packages of the

CNC SW 4.4. Cycle calls in part programs for these cycles cannot be recompiled or processed with cycle support in HMI-Advanced 7.x.

1. Technological Cycles – Functional Scope

The following cycles / functions are included as from SW 2.6 SP1:

Drilling

Cycle	Function and entry softkeys
CYCLE801	Drilling / positions / grid or frame
CYCLE802	Drilling / positions / arbitrary 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 / row of holes
HOLES2	Drilling / positions / hole circle
CYCLE840	Thread / tapping with compensating chuck
CYCLE84	Thread / rigid tapping
CYCLE78	Thread / thread milling

Turning

Cycle	Function and entry softkeys

CYCLE92	Turning / cutoff
CYCLE930	Turning / recess
CYCLE940	Turning / undercut - Form E, form F, undercut thread DIN, undercut thread
CYCLE951	Turning / stock removal
CYCLE98	Turning / thread / threaded chain
CYCLE99	Turning / thread - longitudinal, taper, face
CYCLE62	Contour turning / contour / contour callup
CYCLE952	Contour turning / stock removal, plunging, plunge-turning, all with residual material

Milling

Cycle	**Function and entry softkeys**
CYCLE60	Milling / engraving
CYCLE61	Milling / face milling
CYCLE70	Milling / thread milling
CYCLE76	Milling / spigot / rectangular spigot
CYCLE77	Milling / spigot / circular spigot
CYCLE79	Milling / spigot / polygon
POCKET3	Milling / pocket / rectangular pocket
POCKET4	Milling / pocket / circular pocket
LONGHOLE	Milling / groove / slotted hole
SLOT1	Milling / groove / longitudinal groove
SLOT2	Milling / groove / circular groove
CYCLE899	Milling / groove / open groove
CYCLE72	Contour milling / path
CYCLE62	Contour milling / contour
CYCLE63	Contour milling / pocket, spigot, all with residual material
CYCLE64	Contour milling / pre-drilling
CYCLE800	Various / plane swiveling, tool swiveling
CYCLE832	Various / >> / HighSpeed Settings

Compatibility cycles with 802Dsl:

- These cycles can be recompiled and modified in screenforms.

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 – Recess
CYCLE94	Compatibility 802Dsl – Undercut Form E and F
CYCLE95	Compatibility of stock removal (re. SW 2.5 and 802Dsl)
CYCLE96	Compatibility 802Dsl – Thread undercut form A, B, C, D
CYCLE97	Compatibility 802Dsl – Thread cutting

Other cycles – internal:

Cycle	**Function**

CYCLE861	Reverse countersinking, only for ISO compatibility, no input screen
CYCLE202	Auxiliary cycle for retraction
CYCLE203	Auxiliary cycle for chamfering
CYCLE204	Auxiliary cycle calculate pitch for metric thread
CYCLE206	Auxiliary cycle tool change in JOG
CYCLE206	Auxiliary cycle tool change in JOG
CYCLE207	Auxiliary cycle SERUPRO
CYCLE208	Auxiliary cycle for multi-channel application (joblist treatment)
CYCLE209	Auxiliary cycle for multi-channel application
CYCLE210	Auxiliary cycle machine configuration analysis
CYCLE211	Auxiliary cycle blank definition for simulation
CYCLE809	Auxiliary cycle multi-channel editor
GROUP_ADDEND	Auxiliary cycle multi-channel editor
GROUP_BEGIN	Auxiliary cycle multi-channel editor
GROUP_END	ASUP to support special operating modes
PROG_EVENT	Manufacturer cycle for swiveling (previously TOOLCARR)
CUST_800	Manufacturer cycle for HighSpeed Settings (previously CYC_832T)
CUST_832	Manufacturer cycle for tool change tracking with SERUPRO
CUST_M6	Manufacturer cycle for multi-channel editor
CUST_MULTICHAN	Manufacturer cycle for tool change tracking with SERUPRO
CUST_T	Manufacturer cycle for technological cycles (previously ST_CUST)
CUST_TECHCYC	Reverse countersinking, only for ISO compatibility, no input screen

Important new functions compared to 840D sl SW 1.5 (cycles SW 7.5):

General
- Cycle functions resp. cycle input screens are configured in the configurable machine resp. setting data. Part programs using the old setting data in the GUDs (field _ZSD[x]) can still be executed.
- The multi-channel editor is supported as from this software version.

Drilling
- Drilling cycles support depth selection referred to shaft/tip
- New drilling pattern frame
- New cycle thread milling (CYCLE78)
- Fading out of drilling position is supported with position patterns

Turning
- Contour transfer via CYCLE62 with stock removal
- New cutoff cycle
- Extended recess functionality
- Extended undercut functionality
- Extended thread cutting functionality
- New contour turning cycle (CYCLE952) with the following functionalities:
  - Stock removal
  - Stock removal, residual material
  - Plunging (contour plunging)
  - Plunging, residual material
  - Plunge-turning
  - Plunge-turning, residual material

Milling
- Contour transfer via CYCLE62 with path milling
- Extended face milling functionality (boundary limit)
- Extended thread milling functionality
- New polygon cycle
- Milling cycles can be executed for single positions or position patterns
- Milling cycles use the new Chamfer function
- New contour milling (CYCLE63) and pre-drilling (CYCLE64) cycles with the following functionalities:
  - Contour pocket with or without islands (currently maximally 10 islands possible)
  - Contour pocket, residual material
  - Contour spigot
  - Contour spigot, residual material
  - Pre-drilling
- Commissioning screenform for swivel data records in the commissioning area

Difference list for 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 softkey
POCKET1 POCKET2	Rectangular pocket / circular pocket[1]

[1] POCKET1/POCKET2 function is covered by the newer pocket milling cycles POCKET3/POCKET4

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

Special notes:
- To activate the cycle support, set the machine data
  - MD 52200 MCS_TECHNOLOGY > 0 (1 = turning or 2 = milling).

---

2. Measuring Cycles – Functional Scope

The following cycles / functions are included as from SW 2.6:

Measuring in JOG
The measuring cycles support the following setup functions in JOG (SK "Workpiece zero"):
- Adjust probe (radius, length)
- Set edge
- Align edge
- Distance between 2 edges
- Measure one hole
- Measure 2 or 3 or 4 holes
- Measure rectangular pocket
- Measure one circular spigot
- Measure 2 or 3 or 4 circular spigots
- Measure rectangular spigot
- Measure corner (3 points, rectangular corner)
- Measure corner (4 points)
- Align plane

The "workpiece zero" functions are only provided in the milling technology.

The following measuring functions are provided for tool measurement in JOG (SK "Tool measurement"):
- Tool probe adjustment
- Tool length measurement
- Tool radius measurement
- Tool measurement on turning machines with B-axis at an arbitrary angle

Measuring in the automatic program
- Milling machine tool measurement with the following functions:
  - Calibrate workpiece probe in hole or on surface (CYCLE976)
  - Paraxial measurement of hole, spigot, groove, fillet, rectangular pocket, rectangular spigot (CYCLE977)
  - Edge measurement (CYCLE978)
  - Measure circular segment outside or inside (CYCLE979)
  - Measure groove or fillet at an angle (CYCLE979)
  - Corner measurement in the automatic program (CYCLE961)
  - Angular measurement in the automatic program (CYCLE998)
  - Kinematic measurement (CYCLE996)
- Milling machine tool measurement with the following functions:
  - Tool measurement in the automatic program (CYCLE971)
- Turning machine tool measurement with the following functions:
  - Workpiece probe calibration in groove or on surface (CYCLE973)
  - One-point measurement (CYCLE974)
  - Two-point measurement (CYCLE994)
- Turning machine tool measurement with the following functions:
  - Tool measurement in the automatic program (CYCLE982)

- General functions of the measuring cycles:
  - Measuring cycle support in the editor
  - Measurement result screen
  - Optionally correction of the measuring difference in a ZO or a tool offset data record or measurement only
  - Use of monoprobe or multiprobe

Prerequisites for application
- The measuring cycles require an electric touch trigger probe.
- According to the default machine data for measuring cycles, the workpiece probe must be connected to the first measurement input of the controller, the tool probe to the second measurement input (these settings can be modified via the machine data
  MD 52600 MCS_MEA_INPUT_PIECE_PROBE and
  MD 52601 MCS_MEA_INPUT_TOOL_PROBE)

General
- All measuring cycle data which are important to machine manufacturers and users are now included in the configurable machine and setting data (e.g. data fields of calibration values). The GUD blocks GUD5, GUD6 and GUD7_MC are no longer required.

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

The following cycles / cycle functions are not included in this cycle package:
- Recording of measurement results
- When measuring workpieces with automatic tool offset, you can no longer select a tool environment TENV – this function is an absolute special case and has not been implemented in SW 4.4.

List of omitted measuring cycles:

Cycle	Function and entry softkeys
CYCLE972	Compatibility of tool measurement on turning machines
CYCLE198 CYCLE199	Auxiliary cycles for machine manufacturers (replaced by CUST_MEACYC)
CYC_JM CYC_JMA CYC_JMC	Auxiliary cycles for measuring in JOG (replaced by CYCLE130, CYCLE131)
CYCLE100 CYCLE101 CYCLE105 CYCLE106 CYCLE113 CYCLE118	Auxiliary cycles for recording

Special notes:
- Observe the two following setting data to ensure the functionality of measuring in JOG:
  - SD 54798 SCS_J_MEA_FUNCTION_MASK_PIECE
  - SD 54799 SCS_J_MEA_FUNCTION_MASK_TOOL

3. Cycles ISO Compatibility – Functional Scope

The following cycles / functions are included as from SW 2.6:

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

The cycle setting data for ISO compatibility have been migrated to the machine and setting data. The data fields for GUD setting are no longer relevant.

4. Configurable Machine and Setting Data – Functional Scope

The following functions are included as from SW 2.6:

Newly configurable machine and setting data have been introduced in SW 2.6. After performing a cold start of the controller, these are automatically read in from the CF card and activated.

- These data replace the GUD variables of the cycle packages as well as the display MDs from JobShop and HMI.
- Like all other machine and setting data, they are displayed in the “Commissioning“ area (depending on the protection level) and are included in the number range starting with 50000.
  - 51xxx General MD
  - 52xxx Channel MD
  - 53xxx Axis MD
  - 54xxx General SD
  - 55xxx Channel SD
  - 56xxx Axis SD
- The data names indicate their meaning resp. use; the corresponding codes are stated directly after the prefix, e.g. DISP for display, ISO for ISO cycles, MEA for measuring cycles, TURN for turning technology, etc.
- The configurable machine and setting data support the data class concept for data backup.

5. Compatibility

5.1. Compatibility with cycle packages 840D sl to SW 1.5

Compared to software versions up to SW 1.5, the cycle behavior shows changed cycles as follows:

Technological cycles:

1. Milling cycles – calculation of the infeed depth with / without consideration of the safety clearance (previously settable in the GUD variable _ZSD[1]):
   In the setting data $SCS_FUNCTION_MASK_MILL_SET Bit2 you can set whether the safety clearance shall be considered when calculating the depth or not. So far, “taking into account the safety clearance“ has been set as default value – now, the default value is “without taking into account the safety clearance“.
   Thus, the individual depth infeeds may change in the existing programs.
   The setting of this setting data has the same effect in G-code programs and ShopMill resp. ShopTurn programs.
2. Milling cycles with callup with MCALL:
   Milling cycles in G-code programs can also be called up on position patterns with MCALL.
3. Tapping cycles CYCLE84, CYCLE840 – technological parameters (previously settable in GUD variables):
   The machine manufacturer must enter the required settings for technological parameters in the relevant cycle setting data. These settings are not changed neither by the cycle nor by the input screen.
4. Tapping cycle CYCLE840 – parameter for reversal of rotation direction:
   The parameter “rotational direction for retraction“ is no longer supported in the cycle input screen. This is suitable because the cycle has now this function with both thread types and only functions if a spindle direction has been programmed prior to the callup.
5. Pocket milling cycles POCKET3 and POCKET4 – parameters for plunge feed:
   The programmed feed for depth infeed is only effective for “drilled“ and “verticular“.“Helical“ and “oscillating“ plunging is performed with the feed for machining in the plane.
6. Milling cycle rectangular pocket POCKET3 – reference to the pocket rotation angle:
   The rotation angle is now always active at the reference point. The programming for the rectangular pocket "Reference point is the corner" and "Rotation angle refers to the center" (with ZSD[2]=1 and ZSD[9]=1) is no longer provided, it leads to the alarm 61109 “Parameter _STA incorrectly defined“.
7. Milling cycles – parameters for infeed width:
   A value > 0 must always be entered for the infeed width. In the screenform, the field is marked as incorrect and the cycle outputs an alarm.
8. Drilling and milling cycles – parameters for safety clearance:
   A value > 0 must always be entered for the safety clearance because this value is also active for switching over from G0 to G1 when approaching for machining in the tool axis. In the screenform, the field is marked as incorrect.
9. HighSpeed Settings CYCLE832: The differences are explained in the documentation:
   Documentation up to and including SW 7.5: Cycles – Programming Manual /6/
   Documentation as from SW2.6: Commissioning Instructions IHsl IM9 /5/
   Milling Operating Manual /7/ resp. Turning Operating Manual /8/
   - As from Software Version 2.6, all functions documented in /6/ “Chap. 3.17“ which are coded in the parameter _TOLM, except the operating mode (_TOLM unit’s place), are only supported in compatibility mode.
   - All GUDs described in /6/ Chap. “3.17.4.3 Adaptation of the machinery manufacturer“ and the documented functionality are no longer included in Software Version 2.6.
   - Note regarding /6/ “ Chap. 3.17.4.4 Adaptation of additional program parameters CYC_832T“:
     As from Software Version 2.6, the user cycle CYC_832T.SPF is replaced by the cycle CUST_832.SPF. To ensure compatibility, the marks _M0 to _M4 are still provided in the cycle CUST_832. The transfer parameter _OVL_on of the CYC_832T no longer applies because the documented machine data are not rewritten in CYCLE832.
   - Note regarding /6/ Chap. “3.17.5 Interfaces“:

As from Software Version 2.6, all documented machine and setting data are not rewritten in CYCLE832.
All documented channel-specific variables GUD7 are no longer included in Software Version 2.6.

- The Advanced Surface (AS) option must always be active when using the CYCLE832.
  Otherwise, the error 8025 "Option AS is not set“ is output.
  According to the application example AS included in the HMI Operate commissioning instructions (03/2010), AS requires specific G commands. These G commands need not be compatible with the settings of the "old" CYCLE832 resp. CYC_832T. These new G commands have been entered as proposal (commented with ;) in the individual marks of the compatibility branch (as from the mark _M1).
  The SOFT command is always activated because BRISK does not fit to AS.
  The old calls CYCLE832 should only be used in exceptional cases. When upgrading the machine, we recommend that you replace the old calls by new calls with CYCLE832. The tolerance and machining type settings (roughing, finishing, ..) can be accepted.
  The parameter _FACTOR should not be used any more.
  Factor settings, see commissioning instructions HMI Operate (03/2010) -> SD55441 to SD55443.
  If the parameter _FACTOR in CUST_832.SPF is described in the compatibility branch, this value is effective.

10. Swivel cycle CYCLE800: (reference to documents, see Chap. 8. re. CYCLE832):
    - The menu screens for the commissioning if swiveling stated in /6/ under Chap. “3.16.7.2 Commissioning of kinematic chain“ are no longer included in Software Version 2.6. The parameters (vectors) of the kinematic chain can be read and written as NC variables -> see /5/ CYCLE800.
      Example: $TC_CARR1[2]=123.456 -> Linear vector I1X of the second swivel data record = 123.456
    - The coding of the parameter $TC_CARR37[swivel data record n] has changed:
      The setting “Direction of rotary axis 1“ or “Direction of rotary axis 2“ at the fourth place (thousand) $TC_CARR37[n] is no longer permissible as from this software version and must be replaced by the setting “Direction of rotary axis 1 optimized“ or “Direction of rotary axis 2 optimized“. This supports the pole position of a rotary axis with both direction selection options (plus or minus).
      The following has been determined for coding the parameters _MODE and _DIR in the call interface of the CYCLE800:
      When using an “old“ program call, the setting direction optimized is taken over at the hundred thousand’s place of the parameter _MODE:
      _DIR = -1 _MODE = 10xxxx Swiveling yes, minus direction
      _DIR = +1 _MODE = 20xxxx Swiveling yes, plus direction
      _DIR = 0 _MODE = 11xxxx Swiveling no, minus direction
      _DIR = 0 _MODE = 22xxxx Swiveling no, plus direction
      When changing the direction setting in $TC_CARR37[n] when upgrading to Software Version 2.6, the relevant NC programs must be loaded anew.
      This applies above all to programs where a rotary axis is in the pole position when swiveling, e.g. swivel table rotary axis A=0 (pole position) swiveling with rotary axis C.
    - The coding retraction at the 7th and 8th digit $TC_CARR37[n] x00xxxxxx to x03xxxxxx has been recoded as from Software Version 2.6.

	Up to SW2.6	As from SW2.6
$TC_CARR37[n]	x00xxxxxx	x01xxxxxx Retraction Z
$TC_CARR37[n]	x01xxxxxx	x03xxxxxx Retraction Z or Z, XY
$TC_CARR37[n]	x02xxxxxx	x02xxxxxx Retraction Z, XY
$TC_CARR37[n]	x03xxxxxx	x00xxxxxx No retraction

    - As from Software Version 2.6, the swivel data records must be marked as active or inactive. This is done in parameter $TC_CARR37[n] at the ninth place (HUNDRED MILLION).
      $TC_CARR37[n] 0xxxxxxxx to 3xxxxxxxx means swivel data record is inactive
      $TC_CARR37[n] 4xxxxxxxx to 7xxxxxxxx means swivel data record is active
      As from Software Version 75, the parameters $TC_CARR38[n] to $TC_CARR40[n] are recalculated by the NCK into the current measurement system (inch, metric). If the parameters shall be written into user-defined cycles, this must be observed when upgrading to the Software Version 2.6 in combination with NCK as from Software Version 75.
      NC programs with the data of the swivel data record can be used again with the Software Version 2.6 if the parameter $TC_CARR37[n] is adjusted accordingly.
    - Note regarding /6/ Chap. “3.16.8 Manufacturer cycle TOOLCARR.SPF“

As from Software Version 2.6, the user cycle TOOLCARR.SPF is replaced by the cycle CUST_800.SPF. In terms of function, the cycle CUST_800.SPF corresponds to the cycle TOOLCARR.SPF, including all marks provided for the machine manufacturer to perform the relevant machine-specific adjustments -> see comments in CUST_800.
If the "tool adjustment" function is activated, the first multi-axis transformation (TRAORI(1)) becomes active in the cycle CUST_800.SPF with tool adjustment.
As from Software Version 2.6, the mark _M35 has been newly introduced in CUST_800.SPF to support semi-automatic and manual rotary axes in block search. With regard to content, the mark _M35 replaces the cycle CYCPE_SC.SPF which is no longer included in Software Version 2.6.

- If the optimized behavior is not requested in the basic position (pole position) of the rotary axes (compatibility), it can be deactivated with the following cycle setting data:

  SD55221 $SCS_FUNCTION_MASK_SWIVEL_SET bit 4

  Bit 4 = 0 Evaluation of the input values CYCLE800 in the pole position of the machine kinematics

  Bit 4 = 1 CYCLE800: Compatibility

11. Circular pocket cycle POCKET4 – plunging: With depth infeed with G0 (drilled), the infeed is now always with G0, irrespective of whether base size has been programmed or not. It is assumed that there is enough space up to (DP+SDIS)
    In case of infeed into the material, the programmed helical radius is active if it is smaller than the tool radius. So for, only the tool radius was active in such cases.
12. Pocket milling cycles POCKET3, POCKET4 – special case of one depth infeed: Only one depth infeed is performed when roughing / finishing if the infeed depth exceeds the max. material depth to be removed.

These changes may require an adjustment of existing programs.

Measuring cycles:

1. With the introduction of new machine and setting data for cycles, the data concept for measuring cycles has been revised. Setting data which have so far been stored in GUD variables have now become machine and setting data.
   The document /4/, Attachment A, includes tables with the relevant information:
   - Comparison of GUD parameters ⇔ machine and setting data
   - GUD variables which are no longer used
   - Changes of cycle and GUD block names.
2. When using measuring cycles with different mesuring systems (basic dystem <> programmed system) and programming G commands G70 or G71, the dimensional unit has changed for the following tolerance parameters – they now refer to the programmed system (previously basic system):
   Confidence region (_TSA), zero offset range (_TZL), averaging with offset (_TMV), dimension difference check (_TDIF).

These changes may require an adjustment of existing programs with measuring cycles.

ShopMill cycles:

1. Up to SW 1.5. the reference point (X0, Y0, Z0) for ShopMill could also be entered as increment. In newly opened screenforms, the reference point can only be entered as an absolute value.
   Recompiled screenforms comprising an incremental reference point include a toggle field abs/inc. Upon checking, an error message "Reference point inc is no longer supported, please convert into abs" is output with inc.

6. Commissioning Notes

6.1. Notes for upgrading (for 840D sl only):

- Upgrading for measuring cycle users (2.6 or 2.6 SP1 only):
  - Create machine data difference with the setting MD 11210 $MN_UPLOAD_MD_CHANGES_ONLY Bit0 = 1
  - Readjust the modified machine and setting data according to the lists included in the Attachment
  - Further, you have to adjust user programs or cycles if these MDs/SDs have been read in therein.
    Example:
    If you have so far accessed the MD
    $MNS_J_MEA_MEASURING_FEED in a user program or cycle, this access must be replaced by $SCS_MEA_FEED_MEASURE (s. list of omitted MD/SD).
  - After upgrading from SW 2.x to SW 4.4 and reading in a series commissioning archive, recalibrate the system to ensure that correct calibration data are used.
  - The functions of the manufacturer cycles CUST_xxx (e.g. CUST_800 or CUST_MEACYC) supplied with the cycle version have been extended in SW 4.4. After upgrading from SW 2.6 or SW 2.6 SP1, these manufacturer cycles must therefore be taken again from the CST.DIR and adjusted accordingly.
  - New functions / functional extensions in manufacturer cycles:
    CUST_TECHCYC
  - New exit 103 for “Cutoff terminated“
    CUST_800
  - Extensions for retraction and turning on milling machines
    CUST_MEACYC
  - Support for deactivating/activating the probe during positioning

7. Changes Compared to Previous Software Versions

7.1. Changes included in SW 2.7 / 4.4 compared to SW 2.6 SP1

Functional changes / extensions for configurable machine and setting data:

- New machine and setting data:

  New MDs for protection levels:
  MD 51071 $MNS_ACCESS_ACTIVATE_CTRL_E
  MD 51072 $MNS_ACCESS_EDIT_CTRL_E
  MD 51073 $MNS_ACCESS_SET_SOFTKEY_ACCESS

  New MD for protection levels for grinding data:
  MD 51199 $MNS_ACCESS_WRITE_TM_GRIND

  New MD for function of the second turret in ShopTurn:
  MD 52248 $MCS_REV_2_BORDER_TOOL_LENGTH

  New MD for multi-channel display:
  MD 52290 $MCS_SIM_DISPLAY_CONFIG

  New MD for magnifier position with tool measurement:
  MD 52751 $MCS_T_MEA_MAGN_GLAS_POS

- Extensions in existing machine and setting data:

  MD 51228 $MNS_FUNCTION_MASK_TECH
  Bit 1: Release editor print function

  MD 52207 $MCS_AXIS_USAGE_ATTRIB
  Bit 7: Offer rotational axis in position patterns

  MD 52214 $MCS_FUNCTION_MASK_MILL
  Bit 0: Release of cylinder jacket transformation (ShopMill)

New functions, functional changes / extensions for configurable MDs/SDs:

- Extended machine data MD 52206 $MCS_AXIS_USAGE
  - 11 = Reserved (for B axis of the main spindle)
  - 12 = B axis of the counterspindle (turning)
  - 13 = Transverse travel X of the counterspindle (turning)
- New machine data:
  - 52253 $MCS_M_CODE_TAILSTOCK_FORWARD[2] M-code for quill forward
  - 52254 $MCS_M_CODE_TAILSTOCK_BACKWARD[2]M-code for quill backward

New functions, functional changes / extensions for the Advanced Contour Cycle:

- Two new LOG files have been introduced for CYCLE63 and CYCLE64
  - LOG_CYCLE63C, LOG_CYCLE64C: Contours used
  - LOG_CYCLE63D, LOG_CYCLE64D: Log of calculation module sequence

  The files are created if the file LOG_CYCLE63A resp. LOG_CYCLE64A exists.

New functions, functional changes / extensions for configurable MD/SDs:

- MD 52006 $MCS_DISP_PLANE_TURN: The write authorization is not included by default
- SD 55216 $SSC_FUNCTION_MASK_DRILL_SET: The non-functional Bit0 and Bit5 have been deleted from the bit description

New functions, functional changes / extensions for technological cycles:

- CYLCE800 Direct swiveling:
  New error code with alarm 61153 "No direct swivel mode rotary axes supported“
  C -> Input value for rotary axis 1 not included in the Hirth tooth system grid
  D -> Input value for rotary axis 2 not included in the Hirth tooth system grid
  The value of the Hirth tooth system and the incorrect input value are displayed in the error message
- Swiveling in JOG deselection:
  When swiveling in JOG deselection of swivel data block, the user has been enabled to traverse the rotary axes to zero or an arbitrary value. For this, the mark _M16 is included in the manufacturer cycle CUST_800.spf.
  If CUST_800 is modified by the manufacturer, the manufacturer cycle CUST_800.spf must be updated when upgrading the software version. To traverse the rotary axes in case of deselection, the calls at the mark _M16 must be activated by removing the comment character ";" (N8006).

New functions, functional changes / extensions for configurable MD/SDs:

- MD 51226 $MNS_FUNCTION_MASK_SIM:
  New Bit 1 = 1 Deactivate simulation

7.2. Changes included in SW 2.7 SP1 / 4.4 SP1 compared to 2.7 / 4.4

New functions, functional changes / extensions for technological cycles:

CYCLE832 / CUST_832

- The ORISON command call has been recoded; the following applies with regard to the setting data:
  A value of 0 can be entered for the following setting data:
  SD55441 $SCS_MILL_TOL_FACTOR_ROUGH
  SD55442 $SCS_MILL_TOL_FACTOR_SEMIFIN
  SD55443 $SCS_MILL_TOL_FACTOR_FINISH
  When entering 0 in the relevant SD, no orientation tolerance (OTOL= -1) is programmed upon the call of CYCLE832 in the corresponding machining type (roughing, finishing, etc.).
  The NC command ORISON is activated depending on the orientation tolerance in the manufacturer cycle CUST_832.
- The command ;OST has been removed

CUST_800:

- Unique N numbers are supported
- G groups can now be modified within the cycle (no longer SAVE); the behavior of specific G groups has been reworked

CYCLE800:

- The deselection of TOOLCARR has been reworked

CYCLE809:

- $MC_TRAFO_RESET_NAME is no longer used due to problems arising during NC runup
- Adjustments for simulation and simultaneous recording

New functions, functional changes / extensions for measuring cycles:

CYCLE996

- The parameter _TR_LIM/2 (_TNVL) has been modified (due to angle bisectors)
- Log file output before / after result display modified
- The evaluation of MD51740 $MNS_MEA_FUNCTION_MASK B_AND Bit1 has been adapted (_CBIT[14])
- Improved tool data read-out

New functions, functional changes / extensions for JobShop cycles:

Multiple clamping for ShopMill

- The new manufacturer cycle CUST_CLAMP has been added.

Functional changes / extensions for configurable machine and setting data:

- New MD 51198 $MNS_ACCESS_READ_TM_ALL_PARAM
- Default value for MD 51215 $MNS_ACCESS_WRITE_TM_ALL_PARAM modified from 3 to 1

New functions, functional changes / extensions for technological cycles:

CUST_832

- The NC command ORISON can be programmed by the user after calling the CYCLE832 following the radius or face milling application with TRAORI.

CYCLE800

- Permit reselection of DIAMON/OF in CUST_800(40 + 70) and reselection of plane in CUST_800(40) (to deselect turning)

Functional changes / extensions for configurable machine and setting data:

Tool display:

MD 52271 $MSC_TM_MAG_PLACE_DISTANCE has been activated

MD 52272 $MCS_TM_TOOL_LOAD_DEFAULT_MAG has been activated

MD 52273 $MCS_TM_TOOL_MOVE_DEFAULT_MAG has been activated

New functions, functional changes / extensions for technological cycles:

CYCLE800

Identify kinematic transformations and reselect them, if required

New functions, functional changes / extensions for measuring cycles:

CYCLE996

- The calculated angle of the angular segment of the rotary axis is monitored in the parameter TVL
- After approaching the start position, the distance (A) to the calibrating ball should correspond to the measurement path parameter (FA).

7.3. Changes included in SW 2.7 SP2 / 4.4 SP2 compared to 2.7 SP1 / 4.4 SP1

New functions, functional changes / extensions for technological cycles:

- CYCLE60: Modified form of the digits “4“ and “7“
- CYCLE800 Direct swiveling:
  Error code %4 extended with alarm 61153 “No swivel mode "direct rotary axes" supported“
  %4 = F ROT ? G54 ..
  Direct swiveling + head kinematics + turning in G54.. active -> not allowed
  %4 = F ROT ? SETFRAME
  Direct swiveling + head kinematics + turning in basic reference active -> not allowed
  %4 = F ROT ? CHBFRAME
  Direct swiveling + head kinematics + turning in basis active -> not allowed

Functional changes / extensions for configurable MD/SDs:

- MD 52210 $MCS_FUNCTION_MASK_DISP extended
  Bit 6,7,8: Fade out tool radius, tool length and tool position (icon) in the T, F, S screen

- MD 52214 $MCS_FUNCTION_MASK_MILL extended
  Bit 5: Release of spindle control tool spindle above surface
  Bit 6: Release of spindle control rotating spindle above surface

- New MD 53220 $MAS_AXIS_MCS_POSITION[3]
  Axis **position** in the MCS.
  The three field elements indicate the position in X, Y and Z.
  With linear axes, the value corresponds to the axis zero in the MCS.
  With rotary axes, the rotary axis position is defined in the MCS.

- SD 55642 $SCS_MEA_EDGE_SAVE_ANG is restricted to the value range 0-10°

7.4. Supplementary notes regarding measuring cycles SW 4.4 SP2

- Only with SD 54760 $SNS_MEA_FUNCTION_MASK_PIECE Bit 12 = 1
  Transfer of an offset value “inverted“ to the tool is not permitted in the measuring cycles, also if this can be selected in the input screen.
- When measuring two angles, the protective zone is not taken into account, even if it can be selected in the screen.
- Only with SD 54760 $SNS_MEA_FUNCTION_MASK_PIECE Bit 13 = 1
  Correction into the radius of a tool is not functional if the measuring axis is the tool axis. Correction is always longitudinal.
- The tool type “L-probe” is only correctly supported when measuring in the tool axis (usually the Z axis) to ensure trailing measurement.

- Measuring in JOG / measuring cycles with TRAORI: When measuring workpieces with active TRAORI, the probe must be calibrated to the ball center.
- Calibrating monoprobe on the ball: Monoprobes cannot be calibrated on the ball. For this, use the measuring versions in the ring and on the surface.