CNC Software 4.4 SP1

Installation and Operating Notes

MLFB	Erzeugnis
6FC58**51**-1XG**40-1**YA0	SINUMERIK 840D SL CNC SOFTWARE 31-3 WITH SINUMERIK Operate, 6 LANGUAGES EN,DE,FR,IT,SP,V-CH SOFTWARE 4.4 **SP1** ON CF-CARD WITH LICENSE
6FC58**51**-1YG**40-1**YA0	SINUMERIK 840DE SL CNC SOFTWARE 31-3 Export WITH SINUMERIK Operate, 6 LANGUAGES EN,DE,FR,IT,SP,V-CH SOFTWARE 4.4 **SP1** ON CF-CARD WITH LICENSE

The CNC Software 4. SP1 (internal version 04.04.01.00.012) comprises the following components:

Component	Version
NCK	V83.03.05
SNCK	V02.06.00.00.001
SINAMICS	V04.40.23.17
PLC Op. Sys.	V32.70.20
FB15(sl)	V04.04.11
CP	V02.05.01
MCP CLIENT	V01.04.36
SINUMERIK Operate	V04.04.01.00.009
Linux basic system	V04.40.35.00
NCK file system driver	V04.04.00.00.003
Profinet FW	04.01.33.50
Cycles	V04.04.20.00
SNC	V01.04.00.00.005

Requirements:

- The CNC SW 4.4 SP1 can only be operated on SINUMERIK NCUs 7x0.3 PN.
- CF Card Smart Modular Xceed 8GB. A specific CF card is required for the CNC software:
  - For the export version 6FC58**51**-1YG**40**-**0**YA8 or 6FC5313-6AG00-0AA0
  - For the standard version 6FC58**51**-1XG**40**-**0**YA8.
- No standard versions may be operated on CF cards which are licensed for the export version because, in this case, NC-Start is disabled.
- Step 7 Version 5.5 HF4 (or later)
- Toolbox 4.4.8
- NCU Service System V04.40.35 (or later)
- NCU 730.3 PN with BIOS Version V16.0.0.0 (with PLC Bootloader V2.0)

- If required, startup tool resp. HMI-Advanced V7.6 SP2 HF2 (or later) for drive commissioning
- If required, SINUMERIK Operate V2.7 SP1 for PCU 50 resp. PC / PG

Installing 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.
The software may only be installed on a Smart Modular XceedCF 8GB card – see prerequisites.

Installing 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 7x0.3. 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".

Notes:
- After installing the CNC SW 4.4 SP1 on CF card, perform a general reset of the NCU and PLC: Set the SIM/NCK switch to the position 1, set the PLC switch to the position 3 and switch the system off/on.
- The NCU does not run up with a missing or defective fan module (both fans fail).
  The 7-segment display shows "8", all LEDs are on.
- You cannot always port an NCK series commissioning archive from 840D powerline without processing on 840D sl.
- The license key must be saved before overwriting a licensed CF card. The key is included in the file 'keys.txt' and stored in the path */card/keys/sinumerik*.
  The key can be saved, for example, with WinSCP from the PG/PC.
- The licenses are permanently linked to the CF card (card ID) and can only be used on this card.
- Data backups should be read in according to the following sequence: NC, PLC, Drive. It is suitable to create the archives NC, PLC and Drive separately.
- The contour handwheel and velocity override with handwheel have only been released for Profibus MCP.
- The reset button on the NCU allows to restart the NCU module. A restart of the entire system cannot be ensured in all configurations. This also applies to the PI service of PLC with FB4.
- The catalog /card/siemens/system/etc. includes a template of the current standard basesys.ini.
- As standard, the port 102 on X130 is disabled. If you wish to access the controller via X130 with Step 7 or SINUMERIK Operate, you must activate the port 102.
  This can be achieved by making the following entry in the file /user/system/etc/basesys.ini:
  [LinuxBase]
  FirewallOpenPorts=TCP/102
  If required, you can also activate the port 5900 for an external VNC Viewer.

- The Ethernet handwheel of the HT2 is configured on the fifth handwheel interface.
  Example for three Ethernet handwheels. The third handwheel is located in the 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 HHU included with the Powerline toolbox can only be transferred to the HT2 when making some modifications. The NCVAR variable descriptions must be adjusted accordingly. The FC13 now supports the visualization of four lines. Thus, the meaning of the “ROW“ parameter has changed as follows:
  0 = No display output
  1 = 1st line
  2 = 2nd line
  3 = Line 1 and line 2 alternately
  4 = 3rd line
  5 = Line 1 and line 3 alternately
  8 = 4th line
  F = Automatic change of all lines

  The content displayed must be stored in the field “ChrArray“ of the string[64] type.
  The upper four keys are now transferred into the PLC input image.

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

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

  * Actuate the rotary switches S1 and S2 on the terminal box or MPP to define which DIP number shall be used to register the node in the system (here S1=0, S2=2). The address “0“ is not permissible because no addresses are assigned by the DHCP server with this setting.
- HT2: The upper key row is displayed in the PLC as follows:
  The key on top left (SK1) is displayed 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 controlled by the PLC as follows:
  Signals to handheld unit ABn+0 Bit 0 corresponds to the LED on top left
  Bit 1 for the 2nd LED from top left, etc.

- HT2: Display of 12 characters incl. decimal point and sign with FC13. If you wish to use the extended display on the HT2, the values (in general position values) must be transferred to the FC13 in the format Real 2. To ensure that the values are correctly processed by the FC13, set the parameter DataType = B#16#30. The values must be transferred as they are read by FB2 / FB5 in the format Real 2. The

values cannot be interpreted between the output at FB2 / FB5 and the entry in FC13. The values read by FB2 / FB5 in the Real 2 format are converted accordingly in the FC13.
The parameter Digits indicates the number of decimal places (value range 1..9). Display updating is managed in the FC13. The signals “Acknowledge digit display“ in the EBm+5 and “New data for selected line“ in the ABm+1 need no longer be wired in the PLC.
Example:
- Display of four axis position values which are read with FB2.
- The results of reading from FB2 are stored in the Real 2 format in DB399.DBX0.0, DB399.DBX8.0, etc.
- Three decimal places are displayed.

```
CALL  FB     2 , DB811           // get data
             Req    :=M999.0
             NumVar :4
             Addr1  :* * *
                      *
                      *
                      *
            RD1    :=P#DB399.DBX 0.0 REAL 2  // result
            RD2    :=P#DB399.DBX 8.0 REAL 2
            RD3    :=P#DB399.DBX 16.0 REAL 2
            RD4    :=P#DB399.DBX 24.0 REAL 2
                      *
                      *
                      *

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; //allways 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 adress 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 adress 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 adress 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 adress 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
- Spindle traversing with limited torque (Focon) or spindle traversing to fixed stop:
  The performance indication on the SINUMERIK Operate is calculated as 100% of the limited torque and displayed.
- As before, the DBSI1 is not released in combination with SINUMERIK 840D sl and integrated drive control.
- The Starter, in combination with 840 D sl, is still only released for diagnostic purposes.
- The interpolatory traversal of positioning axes is prevented with the alarm 8031 “Axis has no IPO functionality“.
- SPI and PW have always been keywords and may not be used as axis identifiers.
- The function G643 (block-internal approximate positioning) has been released for tool change applications (e.g. optimizations when approaching the tool change position). It has not been released for machining applications.
- The function G644 (corner rounding with maximally possible dynamics) has been released for tool change applications (e.g. optimizations when approaching the tool change position). It has not been released for machining applications.
- The software version 6 axes / 2 channels is omitted.
- By default, the number of axes has been set to three. When loading old data back-ups which have been created with “upload changes only“, the fourth axis is missing and must be activated manually.
- The function "extended measurement" with distributed measuring input at the SINAMICS modules is not functional (MEAC, MEASA, MEAWA).
- Access to arbitrary drive data via system variable $nn_nn has not been released. Only the system variables transferred in the telegram 116/136 can be accessed.
- As from 4.4 SP1, the drive-autonomous ESR has also been released with Safety Integrated.
  The machine data MD10089 $MN_SAFE_PULSE_DIS_TIME_BUSFAIL (corresponding to p9580 in the relevant drive) allows to configure the time during which the NCK-SGA are still left in the old version (prior to communication failure) in case of a communication failure between NCK and drive. Afterwards, they are initialized.
  The time provided for the safety drives for retraction is defined in the drive parameters p9697 and p9897.
  Notice: These parameters are not included in the safety copy procedure. Proceed as follows to set these times:
  - Go to the safety drive commissioning mode (p10=95); for this, use the HMI functionality (softkey

"Activate drive commissioning")
- Set the value p9697 to the required value (should be identical to MD10089); Unit: ms
- Set the value p9897 to the required value (should be identical to MD10089); Notice: Unit: us
- Quit the commissioning mode via HMI; as a result, the checksums are confirmed and the data stored
- When working without HMI, you have to confirm the checksums manually (copy 9798 to p9898, p010=0, p0971=1 resp. p977=1)

On account of the interrupted communication, data cross-check errors occur, StopF. If the StopF-StopB time is not sufficient, the pulses are deleted by the StopB-StopA, retraction is not started resp. aborted too early (depending on the values for the StopF-StopB time resp. the StopB-StopA time).

- When using the software on an NCU710.3, you can use 8 out of 31 axes and 2 out of 10 machining channels.
- The MD 10062 $MN_POSCTRL_CYCLE_DELAY must be zero. Check the existing data backup.
- The machine data 32250 $MA_RATED_OUTVAL[] must have the value zero.
- After performing a block search, the auxiliary function M6 is no longer output by default.
  Remedy:
  Change the auxiliary function group in MD 22040 $MC_AUXFU_PREDEF_GROUP[5] into a free group.
- If the value of the MD18210 deviates from the default, increase the MD18210 by at least 21 MB DRAM.
- TOOLMAN: The use of several real magazines must be activated via an option. A real magazine is included in the basic scope.
  To ensure compatibility, the existing option bit $ON_TECHNO_FUNCTION_MASK Bit4=1 must still be set. This is, however, not included in the license handling.
- Deleting the PLC 317-3PN/DP:
  After deleting the PLC via an operator sequence at the PLC rotary switch or from Step 7, the PLC program is automatically reloaded.
  When starting the PLC initialization via the operator handling PLC switch into position 3 and power off/on, the PLC program is not loaded automatically. This is then neither possible via the above-mentioned operator sequence.
- The current cycle time is provided in the DB8.DBB95. The cycle time can also be retrieved via the OB1 start infos.
- The PLC series commissionning archive must be created in the stop status of the PLC.
  Otherwise, it is not ensured that the PLC goes to cyclic mode after loading the data backup.
- Sinamics parameters are required for star / delta switchover with FC17.
  Prerequisites:
  The corresponding DDS/MDS must be set up in the drive!
  P827[0] ⇔ P827[1] ⇔ P827[n] must be unequal!
  The contactor is switched over via the application => P833, Bit0=1
  Pulse suppression is performed via the drive => P833, Bit1=0
  Initial PLC position: Star mode, therefore axis DB Bit21.5 must be set to 1 during startup.
- The service interface X127 of the NCU may only be used for commissioning and service.
- The machine control panel (MCP) is operated on the Profibus or Ethernet. No mixed operation.
- The Profibus 1 must be configured to ensure that the softkeys CU, Infeed, Drives are displayed on the SINUMERIK Operate.
- No parallel use of measuring functions for i-/n-measurement via the Starter and HMI-Advanced. If Starter measurement is required, this requires a Power OFF=>ON and direct measurement with the Starter. Measurements with HMI-Advanced may not be performed prior to Starter measurements.
- When replacing a SINAMICS component resp. performing an upgrade, check the firmware version and restore the original version, if required.
  (Keyword: Macro 150399)
  The firmware versions can be checked in the following parameters:
  Control Unit R18 ➔ Firmware Version of the CU
  Infeed unit R128 ➔ Firmware Version of the infeed unit
  Drive MD R128 ➔Firmware Version of the power unit
  R148 ➔Firmware Version of the sensor module
- The Starter is not yet required to determine the motor codes.

- Line voltages: To ensure smooth operation in the target country, the following parameters must be set in the stated order:
  P010 = 1
  P210 = Rated line voltage in the target country
  P211 = Rated line frequency in the target country
  P340 = 1
  P3410 = 4
  P3900 = 3
  Save RAM to ROM
  Switch off the system, only switch it on in the target network.
  With the next ON command on the ALM, a line identification is performed and the values determined are stored in a powerfail-proof manner.
- The function APC increases the NCU utilization. The performance must be assessed before using this function.
- With the SINUMERIK Software Versions 1.3 and 1.4 / 2.4, you may combine different firmware versions of the components involved in the **safety functions** (NCU, NX, Motor Modules, DRIVE-CliQ motors) without adjusting the firmware versions.
  The following applies as from SINUMERIK Software Version 1.5 / 2.5:
  The firmware versions of the Motor Modules, Sensor Modules and DRIVE-CliQ motors (comprising integrated Sensor Modules) involved in the safety functions must be adjusted to the SINAMICS firmware version of the NCU.
  This is automatically performed during the startup if the parameter p7826 (firmware automatic) = 1 is set (standard setting). When using **Safety Integrated**, the parameter p9826 (firmware automatic) =1 must be set and may not be reparameterized. When performing an acceptance test of **Safety Integrated**, you must read out and record the safety firmware versions of all Motor Modules, Sensor Modules and DRIVE-CliQ motors involved in the safety functions, and check them against the following list:
  http://support.automation.siemens.com/WW/view/de/28554461
  Each line in the tables displays a permissible combination of safety firmware versions.
- Drive system utilization of CU or NX:
  Setpoint values are displayed in V4.4 (or later). The values do not fluctuate any more, in contrast to former times. Now, the statement applies that r9976[1] and r9976[5] must both be less than or equal to 100% to ensure that the utilization is permissible. If one of these two indexes exceeds 100 %, the fault F1054 "CU: System limit exceeded" is output.
- In the reset state, the NCU utilization resulting from position controller and interpolator may not exceed **60%**. The current utilization can be checked under Diagnosis / System Resources.
- Cam output (position switching signals) with modulo rotary axes is not possible in a time frame comprising less than 4 IPO cycles and is rejected with Alarm 21751.
- The MPPs and MCPs PN are operated in Ethernet mode on 840D sl. Switch positions 9 and 10 on “ON“ (PN mode) are not permissible.
- The machine data used to describe the spindle dynamics must be set such that they approximately correspond to the real spindle dynamics. If the values are unnecessarily exceeded, alarms may occur upon the transition from spindle to positioning mode.
- NCU 7xx.3 PN: The NCK cannot access Profinet PLC I/Os via system variables.
- NCU 7xx.3 PN: The two Profinet interfaces are located in the 3-way LAN connector block. The lower of the three connectors is the connection for the system network (X120). The connection for the company network (X130) is located besides the USB interfaces.
- NCU 7xx.3 PN: The “DIAG“ button has no function with 840D sl.
- The initial address of the NCK (slot 4 in the S7 hardware configuration) changes from 256 to 4096 (outside the peripheral image). For this reason, the hardware configuration must be newly created. Here, you must ensure that all properties of the PLC CPU (e.g. remanence ranges, cycles, cycle time monitors, etc.) that deviate from the standard are set as before.
- Alarm 411503: When using existing configurations, the alarm 411503 (incorrect configuring) is output and the PLC set to stop.
- With SMI encoder, the function “Parking axis“ is not supported without voltage ON/OFF.
- Alarm 7300: When using NC archives of Software Versions lower than NCK 75.00.00, the alarm 7300 is output if the MD18235 is 0. The standard setting is MD18235=20000.

- The function “Fixed point approach G75“ is not permissible with active radius offset G41 / G42 (Alarm 14091 Index 9).
- With fixed point approach with G75, the fixed point is now approached with non-linear interpolation (RTLIOF). As a result, the axes are traversed at the jerk set for positioning axes ($MA_JOG_AND_POS_MAX_JERK).
- The machine data 11295[0..9] is now memory-configuring. Values deviating from the standard must be removed from existing archives before performing the upgrade.
- If you wish to use the service interface X127 with PG/PC on an NCU7x0.x with SW V2.6 (or later), please observe that the PG/PC network interface is operated in the mode "DHCP Client", that means "Automatically obtain IP address".
  If you have set the PG/PC network interface to a permanent IP (192.168.215.xx), a ping is sent to X127, but you cannot go online neither with Step7 nor with SINUMERIK Operate. The firewall is activated by default setting.
- NCU Link in combination with safety axes is not released for the functions “Safe limit positions“ and “Safe cams“.
- If you wish to operate more than six axes, use the modules NX10.3 or NX15.3.
- Archives with MD 19730.11 = false are incompatible. The machine data 19730.11 (hmi_function_mask) must always be set.
- Archives with MD18150=150 (old default value / default is 196) may be incompatible.
- Archives with MD 32250 > 0 are incompatible. With MD32250 > 0, neither speed setpoint value nor torque setpoint value standardization are taken from the drive.
- The interface signal "Invert M3/M4" is now also active when tapping with G331/G332.
  In applications where the interface signal is always set to "1", the spindle now rotates in the incorrect direction with G331/G332. To achieve a behavior which is compatible with earlier versions, set the bit 22 in the MD35035 to "1" (default is 0).
- Archives with MD28253=100 (old default value) may be incompatible. Change the MD to 200 (default new).
- Archives with MD19730[0] =0 are incompatible. Missing softkeys in MDA. Change the MD to 804hex.
- In Version V2.6 (or later), the gear stage 1 could be set to 0 and 1 in DB[axis].DBB16. As from Version 2.6, a gear stage change is always requested when setting 0 in the data byte 16. When acknowledging the gear stage change with "0", the alarm 22010 "Actual gear stage does not correspond to the requested gear stage" is output. You may have to adjust the PLC program.
- The function TANGON with additional parameter "P" has not been released. Remedy: Use the standard setting "S".
- Upon each system startup, the Siemens cycles are loaded from the CF card into the NCU. This behavior can be deactivated / activated via the service commands sc disable s_cycles / sc enable s_cycles and subsequent general NC reset.
- Customer-specific macro definitions may not be entered in the SMAC.DEF.
  Before performing a software upgrade, customer-specific macro definitions must be entered in the block MMAC (manufacturer) or UMAC (end user).
- The SD43235 has a speed limit of 10,000 rpm as default value. When upgrading to Software Version NCK 78.00 or later and with spindle speeds exceeding 10,000 rpm, the SD43235 must be increased to an appropriate 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 upon the next T0 M06 to store the multitool in the magazine.
- Safety:
  When upgrading from SW 2.6 to 2.7 / 4.4, the following checksums or DCC alarms may be output:
  New machine data / confirmation of axial checksums requested.
  NCK 83.02 already includes specific machine data to ensure the future functionality.
  As these are also entered in the checksum, the axis-specific checksums must be confirmed when upgrading to the Version 4.4.
  Alarm 27001 Code 231

Further, the alarm 27001 with code 231 (resp. 201711 code 232) may be output when using one of these new machine data with rotary axes.
This is due to the different values of 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 the value has been converted to the rotary axis.
To prevent that this alarm occurs during startup, this machine data must be adjusted. This can either be done by using the HM function "Copy SI data" or by entering "10" in the machine data 36947, $MA_SAFE_VELO_X_HYSTERESIS.
The machine data 36947, $MA_SAFE_VELO_X_HYSTERESIS is reserved for a future functionality. The value entered there is currently not evaluated.

Alarm 27071 checksum error SPL parameterization
To facilitate handling when testing the valid scope of languages of the program SAFE.SPF, bit 2 of the machine data $MN_SAFE_MODE_MASK has not been included in the checksum calculation. This allows to perform tests without confirming the checksum.
Due to the recalculation of the checksum without this bit, the relevant checksum must be confirmed once.

- Safety:
  If safe operating stop is active for an axis and the handwheel is actuated for this axis in this state, the pulses are collected and traversed after deselecting the safe operating stop.
  Proceed as follows to avoid this behavior:
  In MD MD 20624 $MC_HANDWH_CHAN_STOP_COND, set bit 7 to "1" (feed stop)
  In the PLC, select the link "Select feed stop with active handwheel and active SBH".
  Upon deselection of SBH, deselect the feed stop as well.
- When recommissioning the SINAMICS, the default telegram 136 is now selected according to the hardware configuration in Step 7 (in drive parameter P922). In the NC machine data 13060[x], this telegram must be selected manually.
- The CF card includes a directory /siemens/oss-licence which (in subdirectories) includes license texts and copyright notes for Open Source components (in pdf files).
- The terminal assignment of X122/X132/X142 is described in “Commissioning of CNC: NCK, PLC, Drive, Commissioning Manual, 02/2011, 6FC5397-2AP40-0AA0“.
- The alarm “150201 Communication to the PLC failed“ sporadically occurs after the startup.
- If modules are connected via Profibus (e.g. master-slave coupling), problems occur with the online connection in case of a general reset of the PLC.
  Remedy: Open bus, separately load modules, close bus.
- Due to the F-CPU, the I/O addresses of F-DI/F-DO modules must now be included in the PLC process image.
- Multitool:
  A multitool may not be a manual tool
  a. With certain change procedures, the target position for the old tools is not output to the user interface (DB72)
  Tool management: Multitool, incomplete data for the old tool to VDI in case of MT change.
  b. With the end acknowledgement of the T preparation, the multitool position ($TC_MTP_POS) is already set to the new location.
  Thus, the system cannot detect during a following change cycle that the multitool must be positioned.
  Tool management: Multitool $TC_MTP_POS is incorrectly set with MT==manual_tool.
  Tool provision :
  Programming sequence
  ...
  M06
  T="drill";Provision
  ...
  T="drill";repeated tool programming
  M06
  ...
  Can only be used if the Bit11 is set to 0 in $MC_Tool_Management_Mask.

Otherwise, this programming leads to the Alarm 6402 upon the 2nd tool call.
Tool management: Multitool, Alarm 6402 in combination with MD20310 Bit11=1
That means the setting $SCS_FUNCTION_MASK_TECH_SET Bit0=1 is not supported (tool preselection)

Functional improvements in NCK compared to SW 4.4:

PR Number	AP01210027	AP01203908	AP01203239	AP01202263
AP01199448	AP01194644	AP01183680	AP01176397	AP01171902
AP01166900	AP01164249	AP01164119	AP01163181	AP01154601
AP01149864	AP01148160	AP01141537	AP01140972	AP01131853
AP01128387	AP01124036	AP01116485	AP01116290	AP01116030
AP01115493	AP01111648	AP01111302	AP01108675	AP01099388
AP01098780	AP01057841	AP01055441	AP01038533	AP00998187
AP00975035	AP00914458	AP00312426		

SINUMERIK Operate
Functional improvements in Operate and cycles compared to SW 4.4:

PR Number	AP00960977	AP01162116	AP01172299	AP01185669
AP01205512	AP01148050	AP01160914	AP00992361	AP01172912
AP01199616	AP01186236	AP01168594	AP01175659	AP01190102
AP00987008	AP01166519	AP00934607	AP01032664	AP00933176
AP01143323	AP01177980	AP01176527	AP01203095	AP01201535
AP01183826	AP01153381	AP01133017	AP01154100	AP01167222
AP01170101	AP01172871	AP01174505	AP01194909	AP01173229
AP01160940	AP01137183	AP01114681	AP01159558	AP01162995
AP01162627	AP01172305	AP01187602	AP01181442	AP01181467
AP01185425	AP01167172	AP01193947	AP01220493	AP01157227
AP00976957	AP00982650	AP00995404	AP01061138	AP01063398
AP01171613	AP01147179	AP01154508	AP01161319	AP01153332
AP01168923	AP01174748	AP01175218	AP01185164	AP01221538
AP01169737	AP01217755	AP01185024	AP01171077	AP01196116
AP00985891	AP01166202	AP01209483	AP01004074	AP01182077
AP01186611	AP01185756	AP01215324	AP01033435	AP01141362

Notes regarding SINUMERIK Operate:

- The internal SINUMERIK Operate must be deactivated when operating an NCU without TCU (that means, only with HMI-Advanced or SINUMERIK Operate on PCU 50). This is done via the service command “sc disable hmi“.
- The operator interface software HMI-Avanced 7.6 SP2 HF2 (or later) resp. the startup tool 7.6 SP2 HF2 (or later) is required for drive commissioning.
- Only one SINUMERIK Operate may be active on one NCU, either the internal SINUMERIK Operate or SINUMERIK Operate for PCU 50 / PC.
  Exceptions: Commissioning / service; SINUMERIK Operate on PCU 50 for the main operator station with permanently assigned MCP and one tool loading station with internal SINUMERIK Operate without MCP.
- Path names in SELECT instructions of job lists referring to part programs on the CF card or USB data carrier are case-sensitive.
- Handling of special characters in the editor:
  The SINUMERIK Operate editor allows to edit text files which use the character LF (0aH) or the string CRLF (0d0aH) as line resp. end-of-block identifier. Binary files cannot be opened with the editor. Files with the extensions .EXE, .LIB, .ELF, .ARC, .TS, .ZIP, .SO, .PNG, .BMP, .ICO, .CFS, .BIN, .QM, .HMI,

.CFG, .ACX, .EMF, .ALM, .ARD, .TGZ, .PTE, .CYC cannot be opened either.
Files which are newly created with the SINUMERIK Operate editor have been coded in UTF8 and use the character LF as end-of-block identifier. With UTF8-coded files, all special characters are displayed correctly. When opening files, the SINUMERIK Operate editor assumes that the files have been coded in UTF8. When opening files with another coding, e.g. with Windows code-page coding, special characters are only displayed correctly if the SINUMERIK Operate has been switched over to the corresponding system language. This applies, for example, to files which have been created with the HMI-Advanced editor. The coding of such files is not modified when being opened with the SINUMERIK Operate editor. They are not automatically converted into the UTF-8 coding.
If files are created or edited with an external editor (e.g. Notepad under Windows) rather than the SINUMERIK Operate editor, please ensure that the file is saved in UTF8 coding. For this, when using Notepad, enter "UTF-8" as encoding when storing the file in the dialog "Save As". If no special characters are used, you can also state "ANSI" as encoding.
For the V24 data transfer, install a V24 module in 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_MACHINE_MASK Bit 3) because this may lead to an incorrect behavior of the operator interface.

Networking:

- **Please read the current documentation "Operator Components and Networking (IM5) Version 03/2011" !**
- **All settings regarding networking and the TCU configuration can be performed via the "System Network Center" which is provided both on PCU 50 and in the NCU.**
- Important system and network settings of the NCU Base software are specified in the file **basesys.ini** in the directory /card/user/system/etc and may be modified. The original basesys.ini is called "template-basesys.ini" and is included in /card/siemens/system/etc ".
  Each NCU in the plant network should be assigned an unambiguous ("talking") computer name, with the entry "Hostname=..." in the basesys.ini. Upper and lower cases, digits and minus signs are allowed.
- Always switch in the DHCP server of the NCU.
- When changing IP addresses of NCUs / PCUs, execute the service command "sc clear dhcp" to activate the change.

Configuration with 1 NCU with TCUs and MCPs
In this configuration, no specific settings must be made in the basesys.ini. The DHCP server and internal SINUMERIK Operate remain switched on.

Configuration with 1 NCU with 1 PCU 50, MCPs and possibly TCUs
In this configuration, the following settings must be made in the basesys.ini of the NCU:

```
Hostname = ....
SyncModeDHCPD_SysNet = ON_MASTER
```

In general, the internal HMI-sl must be switched off because two HMIs (HMI-Advanced and Operate) may only be operated on one NCU in exceptional cases.
We recommend that you use the PCU Base Software V8.6 SP1 (or later) on the PCU 50.3.
If the system has a TCU (HT8) which shall be connected to the PCU 50.3 as standard, the following entries must be made in the file /card/user/common/tcu/<TCU name>/common/tcu/config.ini on the NCU:

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

Configuration with several NCUs and possibly 1 or several PCUs, TCUs, MCPs
In this configuration, the following settings must be made in the basesys.ini of the NCU:
Unambiguous Hostname = ... for each NCU
Unambiguous InternalIP= ....
Same InternalNetMask= ....
Exactly one NCU with SyncModeDHCPD_SysNet = ON_MASTER
During the first startup after networking, the DHCP master shall run up first, followed by all other stations.
The PCU Base software V8.6 SP1 (or later) is required on the PCU 50.3.
In the System Network Center, tab TCU-Support, keep the default configuration "Complete TCUsupport" unchanged. We recommend to set "Sync mode low priority" in the tab DHCP Settings.
All operator stations in the plant network (TCUs, HT8, PCU 50) are managed on the NCU with "ON_MASTER", that means the config.ini files relevant during the runtime and the .leases file comprising all IP addresses assigned in the system are located here.
The config.ini files are distributed by the master NCU to all other NCUs / PCUs via the service command "sc distribute tcudata".
With the service command "sc clear dhcp" and subsequent switching off / on of the overall system, the NCUs / PCUs are assigned their preset IP addresses, which are newly assigned for TCUs and MCPs, the .leases file is the distributed to all other NCUs / PCUs.
Important: The System Network Center, tab OPs on each PCU 50.3 / PCU 50.5 allows to edit the config.ini files of all operator stations directly on the master NCU. The data are automatically distributed to all NCUs / PCUs upon each change.

Cycle packages:
With CNC SW 4.4 SP1, the following cycle packages are stored on the CF card:
- Standard cycles (technological cycles)
- Measuring cycles
- ISO cycles
- ShopMill cycles
- ShopTurn cycles

All these Siemens cycles are automatically loaded onto the NC during startup of the NCU. 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.

Techological 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
GROUP_ADDEND	Auxiliary cycle multi-channel editor
GROUP_BEGIN	Auxiliary cycle multi-channel editor
GROUP_END	Auxiliary cycle multi-channel editor
PROG_EVENT	ASUP to support special operating modes
CUST_800	Manufacturer cycle for swiveling (previously TOOLCARR)
CUST_832	Manufacturer cycle for HighSpeed Settings (previously CYC_832T)
CUST_M6	Manufacturer cycle for tool change tracking with SERUPRO
CUST_MULTICHAN	Manufacturer cycle for multi-channel editor
CUST_T	Manufacturer cycle for tool change tracking with SERUPRO
CUST_TECHCYC	Manufacturer cycle for technological cycles (previously ST_CUST)

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.

Drilling
- Drilling cycles support depth selection referred to shaft/tip
- New drilling pattern frame
- New cycle thread milling (CYCLE78)

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

Important new functions as from SW 2.6 SP1:
General

- The multi-channel editor is supported as from this software version.

General
Cycle functions resp. input screens for cycles are configured in the
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 softkeys**
POCKET1 POCKET2	Rectangular pocket / circular pocket[1]

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).
- TOOLCARR data cannot be displayed or entered. They can only be set via the NC program.
- The following adjustment must be made in CUST_T and CUST_M for the “Multitool“ application:

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

The following must always be requested when deriving tool data from the magazine location assignment:
Multitool == no --> o.k. continue as before
Multitool == yes --> which tool is in the machining position --> collect this T no.

Example:

```
T_Nr=$TC_MPP6[9998,_LOCATION]
if T_No>=1
  if $P_TMNOIS[T_No]==0 ;Multitool
   _T_Spi=$TC_MTPP6[T_Nr,$TC_MTP_POS[T_No]]
  else
    _T_Spi=T_No
  endif
endif
```

Measuring cycles – Functional scope

The following cycles / functions are included:

Measuring in JOG
The measuring cycles support the following setup functions in JOG (SK “Workpiece zero“):

- Adjust probe (radius, length)
- Set edge
- Align edge
- Measure corner (3 points, rectangular corner)
- Measure one hole
- Measure one circular spigot
- Distance between 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 “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)

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

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

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

(see also /4/ Attachment)

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
- After upgrading from SW 2.x to SW 2.6 SP1 and reading in a series commissioning archive, recalibrate the system to ensure that correct calibration data are used.

Cycle ISO compatibility – Functional scope

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.

Configurable machine and setting data – Functional scope

Newly configurable machine and setting data have been introduced in SW 2.5. 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.

Compatibility

Compatibility with cycle packages 840D sl to SW 1.5

Compared to Software Versions up to SW 1.5, the cycle behavior has 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/ “Chapt. 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/ Chapt. “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/ “ Chapt. 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/ Chapt. “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 Chapt. 8. re. CYCLE832):
    - The menu screens for the commissioning if swiveling stated in /6/ under Chapt. “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/ Chapt. “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.

Commissioning notes

Notes for upgrade (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.
  -

Attachment:
Compatibility list re. machine and setting data for measuring cycles

- 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

Changes compared to SW 2.6 SP1

New functions, functional changes / extensions for technological cycles:
Changed terminal behavior when processing slotted holes: Cycle LONGHOLE extended
Engraving with mirror writing: Cycle CYCLE60 extended
Input screen for swiveling in the commissioning area
Hide drilling positions
New cycle CYCLE809: Tool orientation with static transformation for turning on milling machine
The Siemens cycles no longer use WAIT marks from the user area.

New functions, functional changes / extensions for measuring cycles:
Calibration on reference ball (FDesc: 503.5001)
Measuring cycle support with Operate screen functions
Measuring result screens display tool names
Introduction of new machine and setting data for the measuring cycles:

MD51740 $MNS_MEA_FUNCTION_MASK
MD52740 $MCS_MEA_FUNCTION_MASK
SD54740 $SNS_MEA_FUNCTION_MASK
SD55740 $SCS_MEA_FUNCTION_MASK
In these bit-oriented data, measuring cycle functions are activated or deactivated (previously _CBIT resp. _CHBIT fields). Thus, several individual machine and setting data, which have been introduced in SW 2.6, are no longer included!

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:
New 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)

MD 52218 $MCS_ FUNCTION_MASK_Turn
Bit 6: Release of balance cutting for 2-channel stock removal

All new machine and setting data for measuring cycles as well as changes and compatibility lists, see. 0. Attachment

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.

Notes regarding SW 4.4 SP1

Measuring cycles

- 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 t.
- 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.
- The function Formation of empirical values (EVNUM) isnot supported with edge alignment (with CYCLE998).

Attachment:
Compatibility list re. machine and setting data for measuring cycles

New function-determining MD/SD MEA_FUNCTION MASK

New machine and setting data MEA_FUNCTION_MASK have been introduced to concentrate all activations / deactivations of subfunctions in the measuring cycle package at a central location.
NOTICE: Thus, some of the machine and setting data which are already included up to SW 2.6 SP1 are omitted. Access to these data must be adjusted accordingly in the NC program.

Assignment table:
MEA_FUNCTION_MASK – function - existing MD/SD – GUD parameter

General machine data: **MD51740 $MNS_MEA_FUNCTION_MASK**			
Bit [1)]	**Function**	**MD identifiers up to SW 2.6 SP1**	**GUD_Name Up to SW 1.5**
- Workpiece measurement			
0	Calibration monitoring Default = 1	51616 $MNS_MEA_CAL_MONITORING	_CBIT[16]
1	Probe length reference in the infeed axis 0 = Reference point is the probe ball center 1 = Reference point is the probe ball circumference Default = 1	51614 $MNS_MEA_PROBE_LENGTH_RELATE	_CBIT[14]
2	Consideration of orientable tool holders with correction into a tool Default = 0	51610 $MNS_MEA_TOOLCARR_ENABLE	_CBIT[7]
3	Offset angle for mono workpiece probe Default = 1	51612 $MNS_MEA_MONO_COR_POS_ACTIVE	_CBIT[8]
- Tool measurement			
16	Consideration of orientable tool holders with correction into a tool Default = 0	51610 $MNS_MEA_TOOLCARR_ENABLE	_CBIT[7]

[1)] Bit x=0 means the function is deactivated
Bit x=1 means the function is activated
All non-documented bits are not assigned.

Channel-specific machine data:
MD52740 $MCS_MEA_FUNCTION_MASK

Bit [1)]	Function	MD identifiers up to SW 2.6 SP1	GUD_Name Up to SW 1.5
- Tool measurement			
0	Workpiece probe measurement input 0 = CNC measurement input 1 1 = CNC measurement input 2 Default = 0	51606 $MNS_MEA_INPUT_PIECE_PROBE[0]	_CHBIT[0]
1	Turning measuring cycles use the Y axis as the measurement axis Default = 0	52605 $MCS_MEA_TURN_CYC_SPECIAL_MODE	_CHBIT[19]
	Tool measurement		
16	Tool probe measurement input 0 = CNC measurement input 1 1 = CNC measurement input 2 Default = 1	51607 $MNS_MEA_INPUT_TOOL_PROBE[0]	_CHBIT[1]

[1)] Bit x=0 means the function is deactivated
Bit x=1 means the function is activated
All non-documented bits are not assigned.

General setting data:
SD54740 $SNS_MEA_FUNCTION_MASK

Bit [1)]	Function	MD identifiers up to SW 2.6 SP1	GUD_Name Up to SW 1.5
- Tool measurement			
0	Repeated measurement, Measuring difference > _TDIF or >_TSA Default = 0	54655 $SNS_MEA_REPEATE_ACTIVE	_CBIT[0]
1	Repeated measurement with alarm and cycle stop at M0 Default = 0	54656 $SNS_MEA_REPEATE_WITH_M0	_CBIT[1]
2	With alarm referred to _TUL, _TOL, _TSA, Cycle stop at M0 Default = 0	54657 $SNS_MEA_TOL_ALARM_SET_M0	_CBIT[2]
3	The calibrated probe ball radius is entered in the tool data Default = 1	54660 $SNS_MEA_PROBE_BALL_RAD_IN_TOA	_CBIT[15]
- Tool measurement			
16	Repeated measurement, measuring difference > _TDIF or >_TSA Default = 0	54655 $SNS_MEA_REPEATE_ACTIVE	_CBIT[0]

17	Repeated measurement with alarm and cycle stop at M0 Default = 0	54656 $SNS_MEA_REPEATE_WITH_M0	_CBIT[1]
18	With alarm referred to _TUL, _TOL, _TSA, Cycle stop at M0 Default = 0	54657 $SNS_MEA_TOL_ALARM_SET_M0	_CBIT[2]
19	Milling tool measurement, speed reduction upon the last probing	54762 $SNS_MEA_FUNCTION_MASK_TOOL, Bit6	_CHBIT[22]

[1)] Bit x=0 means the function is deactivated
Bit x=1 means the function is activated
All non-documented bits are not assigned.

Customer-specific setting data:
SD55740 $SCS_MEA_FUNCTION_MASK

Bit [1)]	**Function**	**MD identifiers up to SW 2.6 SP1**	**GUD_Name Up to SW 1.5**
	- Tool measurement		
0	Collision detection Default = 1	55600 $SCS_MEA_COLLISION_MONITORING	_CHBIT[2]
1	Spindle position coupling, with coordinate rotation around the infeed axis in AUTOMATIC Default = 0	55602 $SCS_MEA_COUPL_SPIND_COORD	_CHBIT[13]
2	Rotational direction of the spindle positioning, with active spindle coupling and coordinate rotation = 0 counterclockwise, = 1 clockwise Default = 0	55604 $SCS_MEA_SPIND_MOVE_DIR	_CHBIT[14]
3	Measuring attempts with non-switching probe = 0 5 attempts, = 1 1 attempt Default = 0	55606 $SCS_MEA_NUM_OF_MEASURE	_CHBIT[15]
4	Approach speed to the measuring point = 0 with measurement feed _VMS = 1 with $SCS_MEA_FEED_FAST_MEASURE Default = 0	55610 $SCS_MEA_FEED_TYP	_CHBIT[17]
5	Retraction speed from the measuring point = 0 with $SCS_MEA_FEED_PLANE_VALUE = 1 with $SCS_MEA_FEED_RAPID_IN_PERCENT Default = 0	55608 $SCS_MEA_RETRACTION_FEED	_CHBIT[16]
6	Deactivate / activate workpiece probe before and after the NC command SPOS. See also CUST_MEA_CYC.SPF = 0 No call of CUST_MEA_CYC.SPF = 1 Call of CUST_MEA_CYC.SPF Default = 0	-	-
14	Spindle position coupling, with coordinate rotation around the infeed axis	55770 $SCS_J_MEA_SET_COUPL_SP_COORD	E_MESS_SETT[0]

	With measurement in JOG Default = 1		
15	Calibration in calibration ring when measuring in JOG = 0 Calibration with automatic reference center = 1 Calibration with known reference center Default = 0	55771 $SCS_J_MEA_SET_CAL_MODE	E_MESS_SETT[1]
	- Tool measurement		
16	Collision detection Default = 1	55600 $SCS_MEA_COLLISION_MONITORING	_CHBIT[2]
17	Measuring attempts with non-switching probe = 0 5 attempts, = 1 1 attempt Default = 0	55606 $SCS_MEA_NUM_OF_MEASURE	_CHBIT[15]
18	Approach speed to the measuring point = 0 with measurement feed _VMS = 1 with $SCS_MEA_FEED_FAST_MEASURE Default = 0	55610 $SCS_MEA_FEED_TYP	_CHBIT[17]
19	Retraction speed from the measuring point = 0 with $SCS_MEA_FEED_PLANE_VALUE = 1 with $SCS_MEA_FEED_RAPID_IN_PERCENT Default = 0	55608 $SCS_MEA_RETRACTION_FEED	_CHBIT[16]

[1)] Bit x=0 means the function is deactivated
Bit x=1 means the function is activated
All non-documented bits are not assigned.

MDs/SDs omitted as from SW 4.4

Double setting data of the measuring cycles for measuring in JOG and measuring in the program have been combined into one setting each.
NOTICE: Thus, some of the machine and setting data which are already included in SW 2.6 SP1 are omitted. Access to these data must be adjusted accordingly in the NC program.

Assignment table:
The following machine resp. setting data (measuring in JOG) have been included in Cycle Version SW 4.4 (or later) and are no longer included in the Cycle Version SW 2.6 (or later). They will be replaced by the following setting data **which have the same meaning**.

MDw omitted	**Replaced by SDw**
51609 $MNS_MEA_INPUT_TOOL_PROBE_SUB[0 .. 5]	54652 $SNS_MEA_INPUT_TOOL_PROBE_SUB[0 .. 5]
51755 $MNS_J_MEA_MEASURING_FEED	55630 $SCS_MEA_FEED_MEASURE
51774 $MNS_J_MEA_T_PROBE_TYPE[n]	54633 $SNS_MEA_TP_TYPE[n]
51776 $MNS_J_MEA_T_PROBE_ALLOW_AX_DIR[n]	54632 $SNS_MEA_TP_AX_DIR_AUTO_CAL[n]
51778 $MNS_J_MEA_T_PROBE_DIAM_LENGTH[n]	54631 $SNS_MEA_TP_EDGE_DISK_SIZE[n]
51782 $MNS_J_MEA_T_PROBE_T_EDGE_DIST[n]	54634 $SNS_MEA_TP_CAL_MEASURE_DEPTH[n]
51787 $MNS_J_MEA_T_PROBE_MEASURE_FEED	55628 $SCS_MEA_TP_FEED_MEASURE
SDw omitted	Replaced by SDw (observe the relevant bits)
55761 $SCS_J_MEA_SET_NUM_OF_ATTEMPTS	55740 $SCS_MEA_FUNCTION_MASK Bit 17
55762 $SCS_J_MEA_SET_RETRAC_MODE	55740 $SCS_MEA_FUNCTION_MASK Bit 5
55763 $SCS_J_MEA_SET_FEED_MODE	55740 $SCS_MEA_FUNCTION_MASK Bit 4
55770 $SCS_J_MEA_SET_COUPL_SP_COORD	55740 $SCS_MEA_FUNCTION_MASK Bit 14
55771 $SCS_J_MEA_SET_CAL_MODE	55740 $SCS_MEA_FUNCTION_MASK Bit 15
55772 $SCS_J_MEA_SET_PROBE_MONO	Monoprobe function assigned to the tool Tool type 712

MD/SD with modified numbers

The **number of the setting data** has changed with the following setting data.
Identifiers and functions remain unchanged.

SD number as from SW 2.6	**SD Number as from SW 4.4**
54798	54780 $Sns_J_MEA_FUNCTION_MASK_PIECE
54799	54782 $SNS_J_MEA_FUNCTION_MASK_TOOL
55630	55632 $SCS_MEA_FEED_RAPID_IN_PERCENT
55631	55634 $SCS_MEA_FEED_PLANE_VALUE
55632	55636 $SCS_MEA_FEED_FEEDAX_VALUE
55633	55638 $SCS_MEA_FEED_FAST_MEASURE

General overview of the modified MDs/SDs

General overview of the modified machine data	
Up to SW 2.6 SP1	**As from SW 4.4**
	N51071 $MNS_ACCESS_ACTIVATE_CTRL_E
	N51072 $MNS_ACCESS_EDIT_CTRL_E
	N51073 $MNS_ACCESS_SET_SOFTKEY_ACCESS
	N51199 $MNS_ACCESS_WRITE_TM_GRIND
N51606 $MNS_MEA_INPUT_PIECE_PROBE[0]	
N51606 $MNS_MEA_INPUT_PIECE_PROBE[1]	
N51607 $MNS_MEA_INPUT_TOOL_PROBE[0]	
N51607 $MNS_MEA_INPUT_TOOL_PROBE[1]	
N51609 $MNS_MEA_INPUT_TOOL_PROBE_SUB[0 .. 5]	
N51610 $MNS_MEA_TOOLCARR_ENABLE	
N51612 $MNS_MEA_MONO_COR_POS_ACTIVE	
N51614 $MNS_MEA_PROBE_LENGTH_RELATE	
N51616 $MNS_MEA_CAL_MONITORING	
	N51740 $MNS_MEA_FUNCTION_MASK
N51755 $MNS_J_MEA_MEASURING_FEED	
N51774 $MNS_J_MEA_T_PROBE_TYPE[0 .. 5]	
N51776 $MNS_J_MEA_T_PROBE_ALLOW_AX_DIR[0 .. 5]	
N51778 $MNS_J_MEA_T_PROBE_DIAM_LENGTH[0 .. 5]	
N51782 $MNS_J_MEA_T_PROBE_T_EDGE_DIST[0 .. 5]	
N51787 $MNS_J_MEA_T_PROBE_MEASURE_FEED	
N52605 $MCS_MEA_TURN_CYC_SPECIAL_MODE	
	N52248 $MCS_REV_2_BORDER_TOOL_LENGTH
	N52290 $MCS_SIM_DISPLAY_CONFIG
	N52740 $MCS_MEA_FUNCTION_MASK
	N52751 $MCS_J_MEA_MAGN_GLAS_POS[0]
	N52751 $MCS_J_MEA_MAGN_GLAS_POS[1]

General overview of the modified setting data	
Up to SW 2.6 SP1	**As from SW 4.4**
	N54611 $SNS_MEA_WP_FEED[0 .. 11]
	N54651 $SNS_MEA_TPW_FEED[0 .. 5]
	N54652 $SNS_MEA_INPUT_TOOL_PROBE_SUB[0 .. 5]
	N54740 $SNS_MEA_FUNCTION_MASK
	N54760 $SNS_MEA_FUNCTION_MASK_PIECE
	N54762 $SNS_MEA_FUNCTION_MASK_TOOL
	N54764 $SNS_MEA_FUNCTION_MASK_TURN
N54798 $SNS_J_MEA_FUNCTION_MASK_PIECE	N54780 $SNS_J_MEA_FUNCTION_MASK_PIECE
N54799 $SNS_J_MEA_FUNCTION_MASK_TOOL	N54782 $SNS_J_MEA_FUNCTION_MASK_TOOL
N54655 $SNS_MEA_REPEAT_ACTIVE	
N54656 $SNS_MEA_REPEAT_WITH_M0	
N54657 $SNS_MEA_TOL_ALARM_SET_M0	
N54659 $SNS_MEA_TOOL_MEASURE_RELATE	
N54660 $SNS_MEA_PROBE_BALL_RAD_IN_TOA	
N55600 $SCS_MEA_COLLISION_MONITORING	
N55602 $SCS_MEA_COUPL_SPIND_COORD	
N55604 $SCS_MEA_SPIND_MOVE_DIR	
N55606 $SCS_MEA_NUM_OF_MEASURE	
N55608 $SCS_MEA_RETRACTION_FEED	
N55610 $SCS_MEA_FEED_TYP	
	N55628 $SCS_MEA_TP_FEED_MEASURE
	N55630 $SCS_MEA_FEED_MEASURE
N55630 $SCS_MEA_FEED_RAPID_IN_PERCENT	N55632 $SCS_MEA_FEED_RAPID_IN_PERCENT
N55631 $SCS_MEA_FEED_PLANE_VALUE	N55634 $SCS_MEA_FEED_PLANE_VALUE
N55632 $SCS_MEA_FEED_FEEDAX_VALUE	N55636 $SCS_MEA_FEED_FEEDAX_VALUE
N55633 $SCS_MEA_FEED_FAST_MEASURE	N55638 $SCS_MEA_FEED_FAST_MEASURE
	N55640 $SCS_MEA_FEED_CIRCLE
	N55642 $SCS_MEA_EDGE_SAVE_ANG
	N55740 $SCS_MEA_FUNCTION_MASK
N55761 $SCS_J_MEA_SET_NUM_OF_ATTEMPTS	
N55762 $SCS_J_MEA_SET_RETRAC_MODE	
N55763 $SCS_J_MEA_SET_FEED_MODE	
N55770 $SCS_J_MEA_SET_COUPL_SP_COORD	
N55771 $SCS_J_MEA_SET_CAL_MODE	
N55772 $SCS_J_MEA_SET_PROBE_MONO	