SIEMENS

Ingenuity for life



NC-84SL-SIP

SITRAIN
Training for Industry

Commissioning, Startup and Service SINUMERIK 840D sl

SIEMENS

SITRAIN
Training for Automation and Industrial Solutions

Service and Start-up SINUMERIK 840D sl

Course NC-84SL-SIP

Name: _____

Course from: _____ to: _____

Trainer: _____

Training site: _____

This document was produced for training purposes. SIEMENS assumes no responsibility for its contents. The reproduction, transmission, communication or use exploitation of this document or its contents is not permitted without express written consent authority. Offenders will be liable to damages. Non-compliances with this prohibition make the offender inter alia liable for damages.

Copyright © Siemens AG 2011. All rights, including particularly the rights created by to file a by patent and/or other industrial property right application and/or cause the patent and/or other industrial property right to be granted grant or registration of a utility model or design, are reserved.

SITRAIN courses on the internet: www.siemens.com/sitrain

Course folder Version: V4.7.3

1 System overview
2 Hardware
3 Operating
4 Networking and tools
5 Data backup
6 Startup general
7 Diagnostics with SINUMERIK Operate
8 Basic startup
9 Startup axes
10 Extended startup
11 PLC interface
12 Basic settings Tool managment
13 Customizing SINUMERIK Operate
14 Synchronized Actions
15 Axis couplings
16 Compensations
17 Training and Support
18 SINUMERIK machine test protocol
19
20

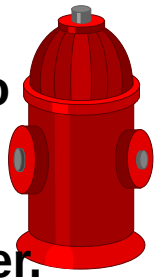
SIEMENS

Safety information for our course participants

- For safety reasons, course participants should only enter the course rooms and areas that have been designated for their use.
- In the event of en emergency, they must follow the instructions issued by their trainer and the Training Center personnel.
- If the building has to be evacuated, follow the emergency exit signs (emergency exit lights).
- In an emergency, only use the Siemens emergency number which you can dial from any internal phone:

Extern: 0911 / 895 – 2050 Int.: 2222

- Please familiarize yourself with the location of the fire extinguishers, wall hydrants and emergency stop buttons.
- Please report any accidents that happen on the way to the Training Center or during course attendance to the Course Office or to your trainer.
- All equipment and tools used by the course participants must be in correct working order; please report any faulty equipment to the trainer.



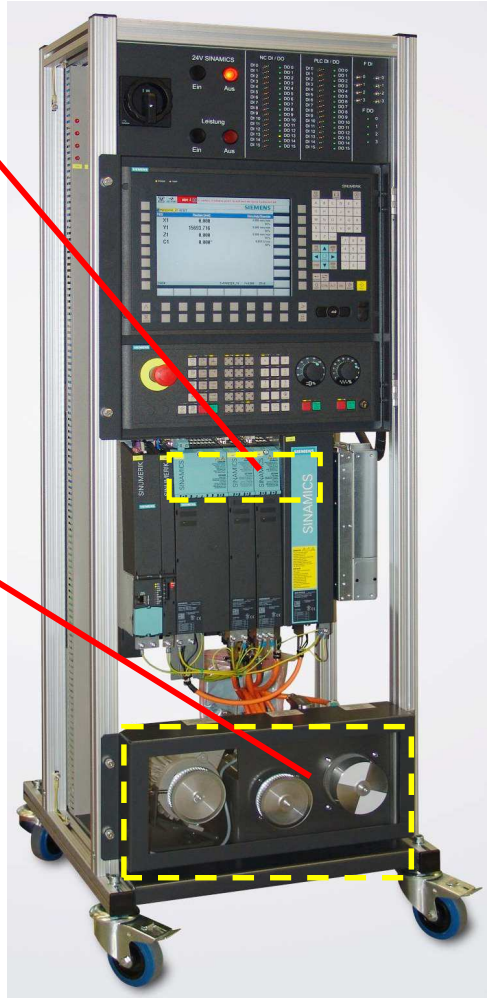
Siemens AG © 2017

Safety information for our course participants

Please use extreme caution when working on devices or parts that are under Voltage (e.g. when checking/testing the 600 V DC link voltage)!

- It is only permissible to use insulated tools/equipment.
- Power capacitors have discharge times of up to 4 minutes. You must not resume operation before this time has lapsed and you have tested they are discharged, as the discharge resistors may be faulty.
- The exercises must never be carried out alone. At least one other person must be present.

When working on the training units, please use caution with rotating parts (motor shafts etc.). It is only permissible to work without protective cover if the control is switched off (emergency OFF).



Siemens AG © 2017

Abbreviation list:

A...	Alarm
AA	Analog Output
AC	Alternating Current
ACX	Compressed Format of XML (Drive data)
ADC	Analog->Digital Converter
ADI4	Analog Drive Interface
AE	Analog Input
AG	Automation Device
AI	Analog Input
AIM	Active Interface Module
ALM	Active Line Module
ANA	Analog Axis (=axis with analog drive)
AO	Analog Output
AOP	Advanced Operator Panel
APC	Advanced Position Control
AS	Automation system
ASC	Armature Short-Circuit (for braking)
ASCII	American Standard Code for Information Interchange
ASIC	Application Specific Integrated Circuit
ASUP	Asynchronous subroutine
BAG	Mode group
BB	Operating Area (german: Bedienbereich)
BCD	Binary Coded Decimal
BCS	Base Coordinate System
BERO	Company name for inductive proximity switch
BG	Professional association (in Germany)
BGIA	German statutory industrial accident insurance institution
BHG	Handheld Unit
BI	Binector Input
BIA	German statutory industrial accident insurance institution
BICO	Binector-Connector-Technology
BLM	Basic Line Module
BO	Binector Output
BOP	Basic Operator Panel
C	Capacitance
CAN	Serial Bus system (Controller Area Network)
CBC	Communication Board CAN
CCF	Common Cause Failure
CDC	Crosswise Data Comparison
CDS	Command Data Set
CF	CompactFlash
CI	Connector Input
CNC	Computerized Numerical Control
CO	Connector Output
CO/BO	Connector Output/Binector Output
COB-ID	CAN Object-Identification
CoL	Certificate of License

COM	Component Objekt Model (Microsoft)
COM	Common contact of a change-over relay
CP	Communication Processor
CPU	Central Processing Unit
CRC	Cyclic Redundancy Check
CTS	Clear to Send
CU	Control Unit
DAC	Digital->Analog Converter
DB	Data Block
DC	Direct Current
DC	Diagnostic Coverage
DCOM	Distributed Component Object Model
DDE	Dynamic data exchange
DDS	Drive Data Set
DHCP	Dynamic Host Configuration Protocol: (automatic assignment of IP addresses)
DI	Digital Input
DIN	German Industry Standard (Deutsche Industrienorm)
DIP	Dual In-Line Package (set of switches assigned in series)
DKE-AK	German electrotechnical gremium (Arbeitskreis)
DMC	DRIVE-CLiQ Module Cabinet (Hub)
DMP	Distributed Machine peripherals
DMS	Direktes Meßsystem
DNC	Direct Numerical Control (MCIS)
DO	Digital Output
DO	Antriebsobjekt (Drive Objects)
DO	Digital Output
DP	Distributed peripherals
DPR	Dual-port-RAM (Dual Ported Random Access Memory)
DRAM	Dynamic Random Access Memory
DRF	Differential Resolver Function (Offset via handwheel)
DRIVE-CLiQ	Drive Component Link with IQ
DRY	Dry run feedrate (for testing)
DSB	Distribution Board
DSC	Dynamic Servo Control
DSP	Digital Signal processor
DVI	Digital Visual Interface,
DVI-D	Digital Visual Interface, only digital
DVI-I	Digital Visual Interface integrated, i.e. analog and digital
DWORD	Double word
ECS-Spindle	engineered catalogue spindle (former ECO-Spindle)
EDS	Encoder Data Set
EG	Electronic Gear
EGB	Electronically endangered (via static voltages...) Unit
EMC/EMV	Electro magnetic compatibility
EN	European standard (Norm)
EnDat	Encoder data interface
EP	Enable Pulses
EQN	Absolute Encoder with Sine/Cosine Signals
ESR	Extended Stop and Retract
FAQ	Frequently asked questions

FB	Function Block
FC	Function call
FCC	Function Control Chart
FCC	Flux Current Control
FDD	Feed drive
F-DI	Failsafe Input
F-DO / F-DQ	Failsafe Output
FFT	Fast fourier transformation
FG	Function generator
FI	Earth leakage circuit breaker
FIFO	First in first out
FP	Function chart (Drives)
FUP	Function chart (Programming language for Step-7)
FW	Firmware
FXS	Travel to Fixed Stop
G	Grinding
GC	Global-Control-Telegram (Broadcast-Telegram)
GSD	Hardware data for Profibus components (Gerätestammdaten)
GSV	Gate Supply Voltage
GUD	global user data
HD	Hard Disc
HF	High Frequency
HFD	High Frequency (Line) Reactor
HHU	Hand held unit
HLA	Hydraulic Linear Drive
HLG	ramp function generator
HMI	Human Machine Interface
HPU	Hand programming unit
HSC	High speed cutting
HT8	Handheld Terminal 8
HTL	High-Threshold Logic
HW	Hardware
I	Input
I/O	In/Output
I/O	Input/Output
IEC	International Electrotechnical Commission
IF	Interface
IGBT	Insulated Gate Bipolar Transistor
IL	Pulse suppression
IM	Interface Module
IP54	Protection against dust deposit and splash water
IP65	Protection against intrusion of dust and water jet
IPO	Interpolator
IT	Insulated three-phase supply network, not grounded
JOG	Operating mode JOG: manual mode for setup
K-Bus	Communikation bus
KTY	Specific Temperature sensor
L	Inductivity
LAD	Ladder diagram (Programming language for Step-7)
LAN	Local Area Network

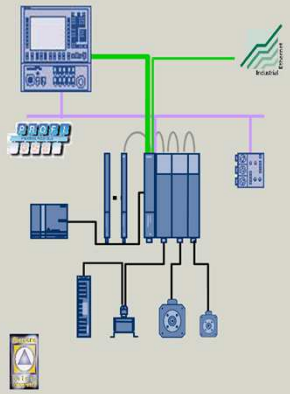
LCD	Liquid Crystal Display
LED	Light Emitting Diode
LSB	Least Significant Bit
LSS	Line Side Switch
LUD	Local user data (user Variable defined in NC program)
M	Milling
M	Mass, inertia
MAC	Media Access Control
MB	Megabyte
MCC	Motion Control Chart
MCI-Board	Motion Control Interface-Board (840Di)
MCIS	Motion control information system
MCP	Machine Control Panel
MCS	Machine Coordinate System
MCU	Motion Control Unit
MDA	Machine data aquisition (MCIS)
MDA	Manual Data Automatic (operating mode)
MDI	Manual Data Automatic (operating mode)
MDS	Motor data set
MLFB	Machine readable product name
MM	Motor Module
MMC	SIMATIC Micro Memory Card
MoMo	Motor Module
MPF	Main Program File
MSB	Most Significant Bit
MSCY_C1	Cyclic Communication between Master (Class 1) And Slave
MSD	Main spindle drive
MTTFd	mean time to dangerous failure
Multi-Turn Encoder	Absolute encoder (absolute within several revolutions)
NAMUR	Standardization association for measurement and control in chemical industries
NC	Numerical Control
NC	Normally Closed (contact)
NCK	Numerical Control Kernel (for block preparation, interpolation, etc)
NCU	Numerical Control Unit
NEMA	National Electrical Manufacturers Association (USA)
NO	Normally Open (contact)
NST	Interface signal (Nahtstellensignal)
NURBS	Non uniform rational base Splines
NX	Numerical Extension
OA	Open Architecture (Packages for HMI and NCK)
OB	Organization block
OEM	Original Equipment Manufacturer
OLP	Optical Link Plug
OP	Operator Panel
OPC	OLE for Process Control
p…	Parameter (of SINAMICS)
PAA	Process image of Outputs
PAE	Process image of Inputs

P-Bus	Peripheral Bus
PCI-Bus	Peripheral Component Interconnect Bus
PCMCIA	Personal Computer Memory Card International Association
PCU	Panel Control Unit (802D), Personal Computer Unit (PCU50)
PDA	Production data acquisition (MCIS)
PDS	Power Module Data Set
PE	Protective Earth
PELV	Protective Extra Low Voltage
PFHD	probability of dangerous failure per hour
PG	Programming PC
PI-Services	Program Invocation Services
PL	Performance Level
PLC	Programmeable Logic Control
PLL	Phase Locked Loop
PM E-F	Power module electronic Failsafe
PN	PROFINET
PNO	PROFIBUS user organisation
PP	Push Button Panel
PROFIBUS	Process Field Bus
PRT	Program test
PS	Power Supply
PSA	Power Stack Adapter
PTC	Positive Temperature coefficient
PTP	Point to point
PUD	Program global User Data
PWM	Pulse width modulation
PZD	PROFIBUS Process data
QFK	Quadrant Error Compensation
r…	Read parameter
RAM	Random Access Memory:
RAS	Remote access service
RC	Robotic Control
RCCB	Residual Current Circuit Breaker
RCD	Residual Current Device
RCS	Remote control service (MCIS)
RDY	Ready
REF	Reference point
REPOS	Re-positioning
RES	Reset
RISC	Reduced Instruction Set Computer
RLG	Rotor position encoder
ROD	Rotary incremental encoder system
ROV	Rapid override
RPC	Remote procedure call (MCIS)
RPDO	Receive Process Data Object
RTCP	Remote tool center point
RTCP	Real Time Control Protocol
RTS	Request to Send
SAR	Safe Acceleration Range (IEC 61800)
SBC	Safe Brake Control (Safety Integrated)

SBH	Safe Operating Stop (Safety Integrated)
SBL	Single block
SBM	Safe Brake Management (Safety Integrated)
SBR	Safe Brake Ramp (Safety Integrated)
SBT	Safe Brake Test (Safety Integrated)
SCA	Safe Cam (IEC 61800) (Safety Integrated)
SCL	Structured control language
SD	Setting data
SDB	System data block
SE	Safe Software limit switch (Safety Integrated)
SFB	System function block
SFC	System Function Call
SG	Safely Reduced Speed (Safety Integrated)
SGA	Safety related Inputs (Safety Integrated)
SGE	Safety related Outputs (Safety Integrated)
SH	Safe Stop == STO (Safety Integrated)
SI	SINUMERIK Safety Integrated®
SIL	Safety Integrity Level
SIM	Single in Line Module
SIMAG	toothed wheel encoder
Single-Turn Encoder	Absolute encoder (absolute within one revolution)
SIZAG	toothed wheel encoder
SK	Softkey
SKP	Skip block
SLA	Safely-Limited Acceleration (IEC 61800)
SLM	Smart Line Module
SLP	Safely-Limited Position (IEC 61800): == SE
SLS	Safely-Limited Speed (IEC 61800): == SG
SLT	Safely-Limited Torque (IEC 61800)
SLVC	Sensorless Vector Control
SM	Sensor Module (SINAMICS)
SM	Signal Module (SIMATIC)
SMC	Sensor Module Cabinet
SME	Sensor Module External
SMM	Safe Motion Monitoring
SN	Safe Software Cam (Safety Integrated)
SOS	Safe Operating Stop (IEC 61800): == SBH
SPC	Setpoint Channel
SPF	Sub Program File
SPL	Safe Programmable Logic (Safety Integrated)
SRAM	Static RAM
SS1	Safe Stop 1 (IEC 61800): == Stop A
SS2	Safe Stop 2 (IEC 61800): == STOP C/D
SSFK	Spindelsteigungs-Fehlerkompensation = Leadscrew Error Compensation
SSI	Serial Synchron Interface
SSR	Safe Speed Range (IEC 61800)
STL	Statement List
STN	Super Twisted Nematic
STO	Safe Torque Off (IEC 61800): ==SH

STOP A	Stop reaction: Pulse suppression; final state: SH
STOP B	Stop reaction: Braking at the current limit; final state: SH
STOP C	Stop reaction: Braking at the current limit; final state:SBH
STOP D	Stop reaction: Braking along the path; final state:SBH
STOP E	Stop reaction: delayed pulse suppression (für ESR)
STOP F	Stop reaction: triggered from crosscheck; variable reaction
STR	Safe Torque Range (IEC 61800)
STW	Steuerwort = Control Word
SW	Software
T	Turning (drehen)
TB	Terminal Board
TCP/IP	Transfer Control Protocol / Internet Protocol
TCU	Thin Client Unit
TDI	Tool data information system (MCIS)
TFT	Thin Film Transistor (Display)
TIA	Totally Integrated Automation
TM	Terminal Module
TN	Grounded three-phase supply network
TP	Touch operator panel
TPM	Totally productive maintenance (MCIS)
TRANSMIT	Transform milling into turning
TT	Grounded three-phase supply network
TTL	Transistor transistor logic
UL	Underwriters Laboratories Inc.
UP	Subroutine
USB	Universal Serial Bus
VC	Vector Control
Vdc	DC link voltage
VGA	Video graphic adapter
VO	Voltage Output
VPM	Voltage protection module für 1FE1-Motoren
VSM	Voltage Sensing Module
WCS	Workpiece coordinate system
ZSW	Zustandswort = Status Word
β	susceptibility to common cause failure

SIEMENS



Chapter 1

System overview of SINUMERIK 840D sl

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
The SINUMERIK family of CNCs 3
840D sl NCU 7x0.3 4
SINAMICS S120 5
Overview of functions 8
Quantity structure 9
Commissioning tools for 840Dsl 10
840D sl basic version with TCU 11
840D sl networking/terminology 12
SINUMERIK Operate 13
SINUMERIK Integrate 15
Run MyScreens 19
HMI Pro 20
What do I find where in the folder? 21

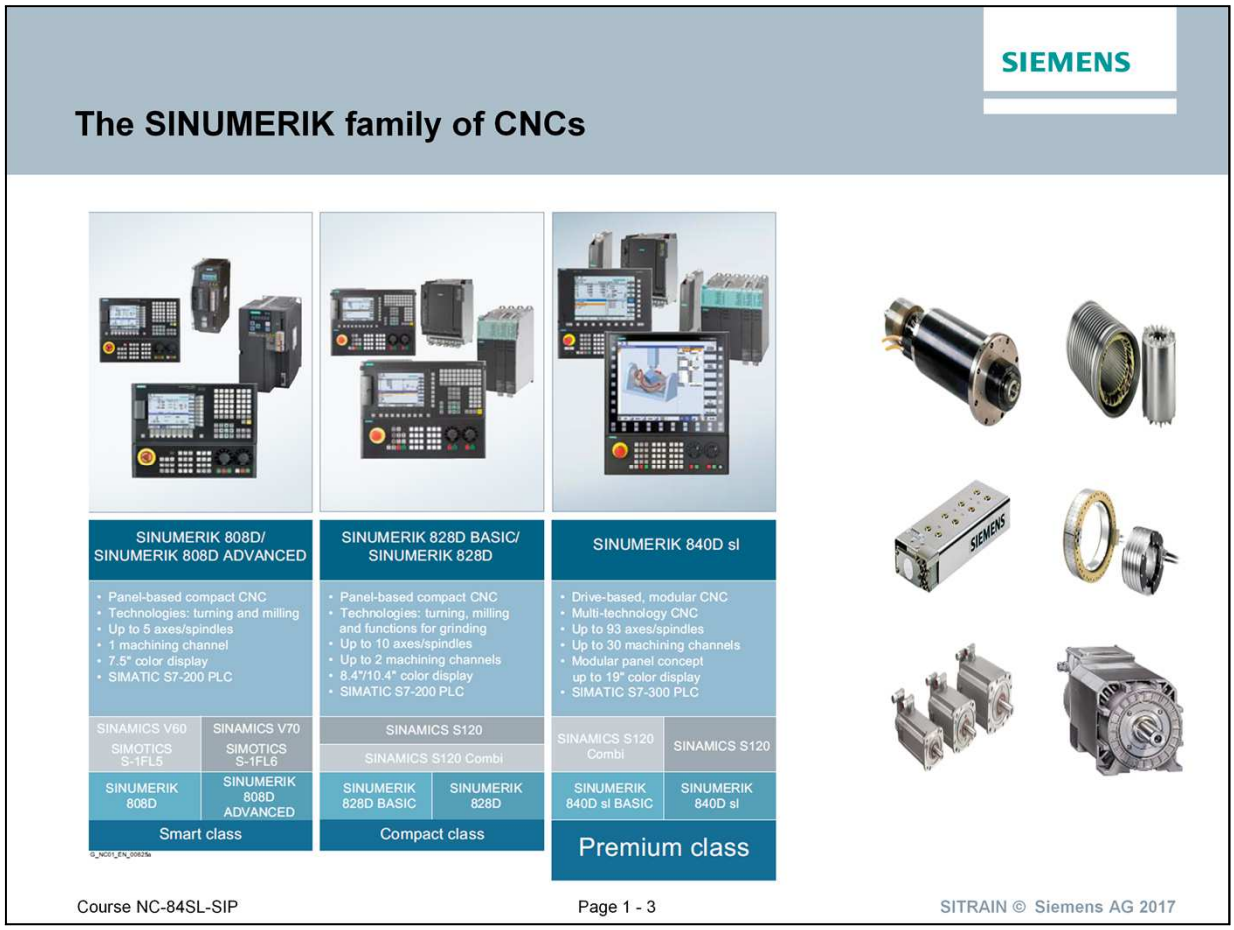
Learning objectives

- You will have an overview of the SINUMERIK 840D sl system
- You will be familiar with the relationship between SINUMERIK 840D sl and SINAMICS S120



SITRAIN © Siemens AG 2017

Notes



808D

The SINUMERIK 808D is a panel-based CNC for the lower performance range. This compact and user-friendly entry-level solution is used for basic turning and milling applications.

828D

This panel-based CNC system is precisely tailored for sophisticated and demanding turning and milling machines, such as those typically used in a workshop - from vertical and simple horizontal machining centers up to inclined-bed turning machines with a machining slide, equipped with a Y axis and counterspindle.

Operator panel and CNC electronics are combined into a rugged and maintenance-free unit. SINUMERIK 828D offers a wide range of powerful CNC functions – and is very compact.

840D sl

SINUMERIK 840D sl offers modularity, openness, flexibility, and uniform structures for operation, programming, and visualization. It provides a system platform with trend-setting functions for almost all technologies.

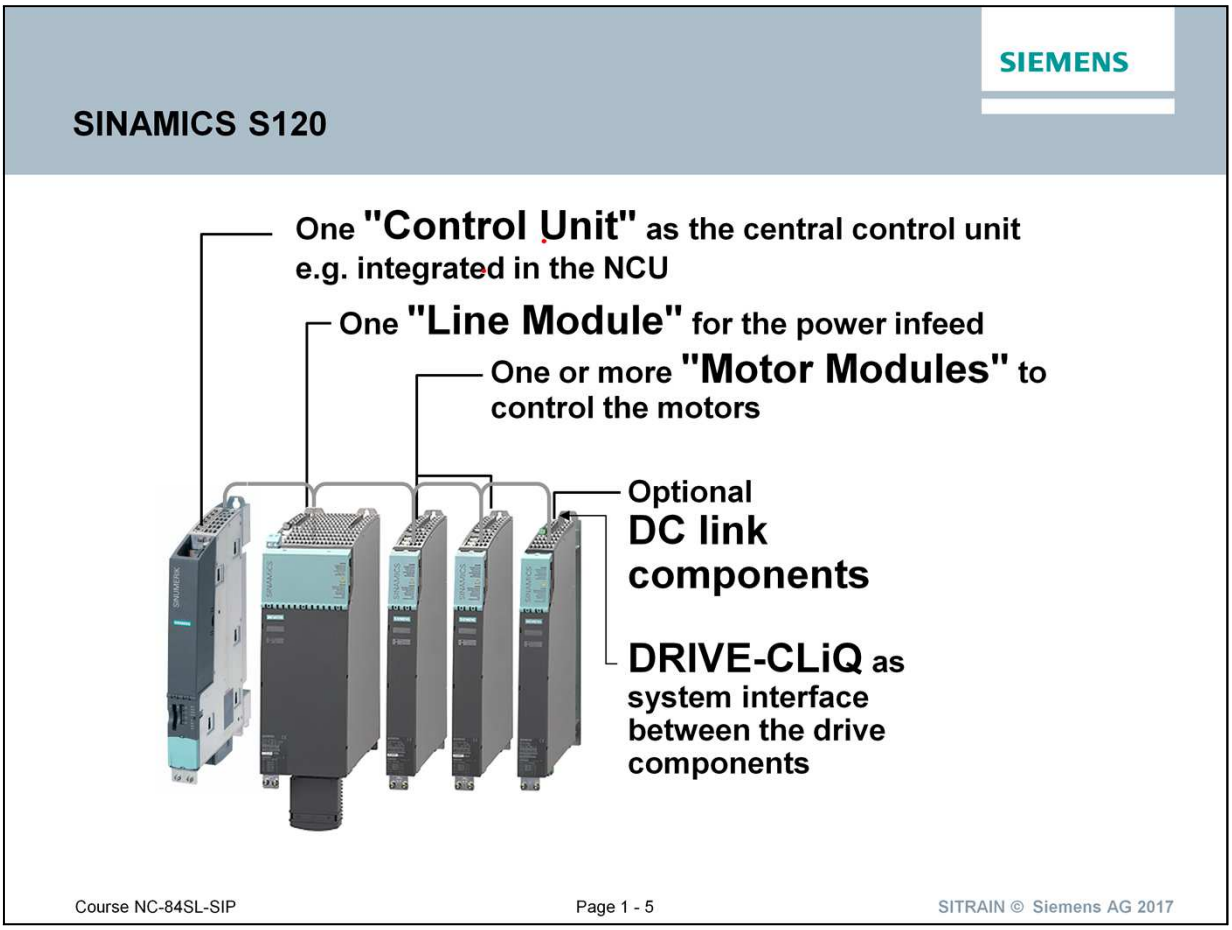
The SINUMERIK 840D sl sets itself apart due to its high degree of flexibility, very high dynamic response, precision, and optimum integration in networks.

840D sl NCU 7x0.3

840D sl BASIC	840D sl
▪ 6 axes/spindles **NCU 710.3 PN**	▪ Up to 31 axes/spindles, in 10 machining channels per NCU ▪ Maximum performance with multicore technology ▪ NCU 720.3 PN ▪ NCU 730.3 PN
	
NC62 catalog	NC62 catalog

SITRAIN © Siemens AG 2017

Notes



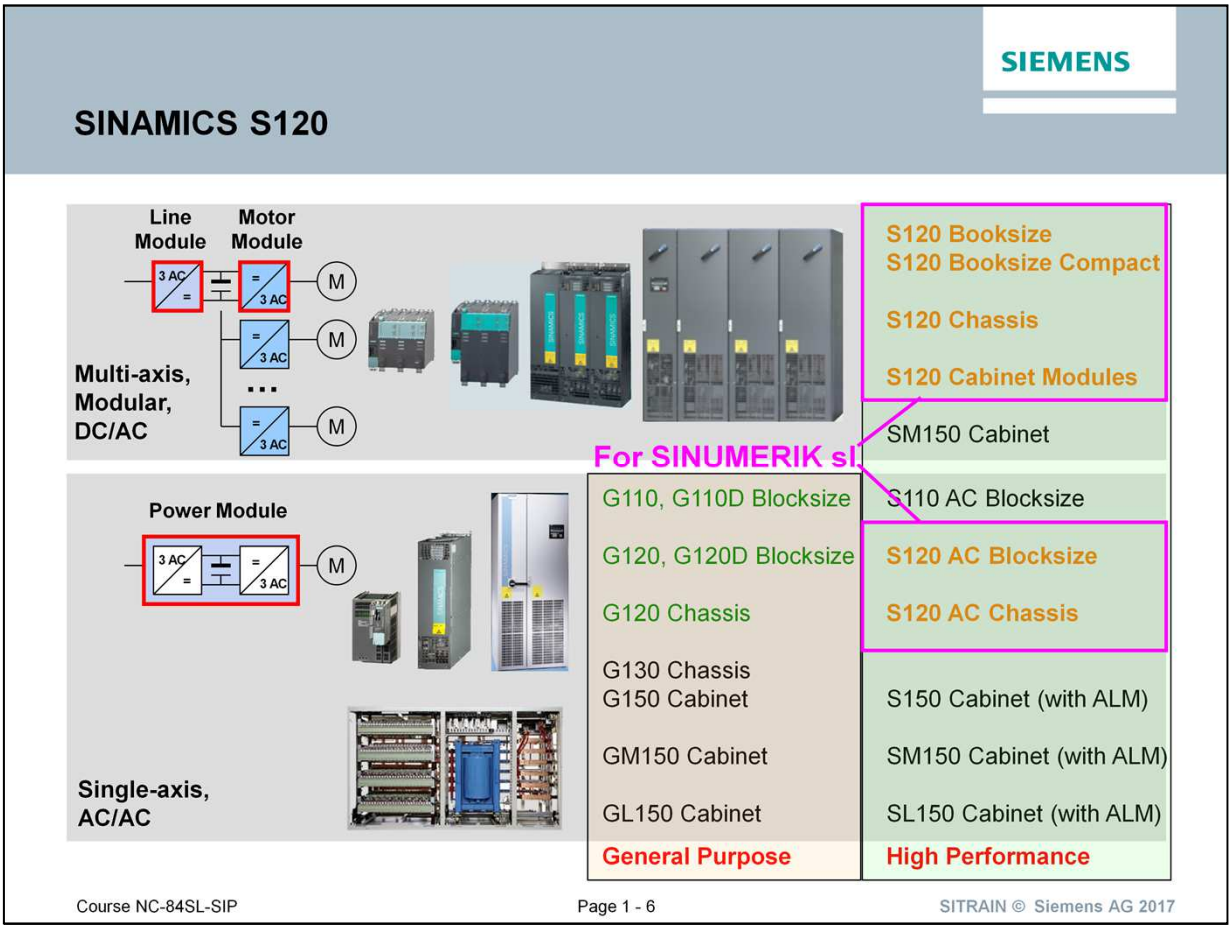
Control Unit

In the case of a drive system for the SINUMERIK 840D sl, the Control Unit is integrated in the NCU or is available as an expansion in the form of an NX module.

It is also possible to use a separate module (CU320-2 DP) as the Control Unit for a SINAMICS drive unit. The corresponding axes are then operated as PLC axes on PROFIBUS.

Restriction

SINUMERIK NCUs cannot be used to drive PLC axes on PROFINET (via CU320-2 PN).



Variants

Depending on the application, the SINAMICS range offers the ideal variant for any drive task.

- SINAMICS G is designed for standard applications with induction motors. These applications have less stringent requirements regarding the dynamic performance of the motor speed.
- SINAMICS S handles complex drive tasks with synchronous and induction motors and is able to meet stringent requirements regarding
  - Dynamics and accuracy
  - The integration of extensive technological functions in the drive control system

The S150 cabinet unit differs from the G150 cabinet unit in that an Active Infeed is used.

AC/AC compact

SINAMICS G drive units are always supplied as compact AC/AC units.
In this case, the infeed and power supply for the motor are combined in a single Power Module.

The types of construction (formats) are Booksize, Chassis, and Cabinet.

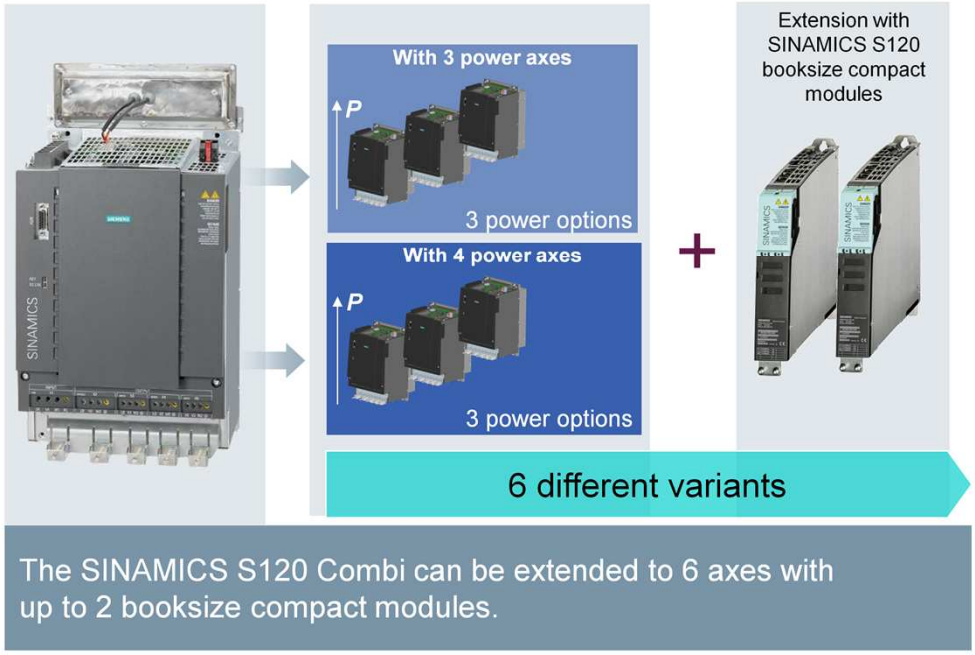
AC/AC cabinet units are not only fully wired up and mounted with customer options, but they also have a predefined software structure to minimize commissioning work.

DC/AC modular

The SINAMICS S120 drive system for high-performance applications is not only available in the form of compact AC/AC units, but also as modular DC/AC units in the Booksize Compact, Booksize, Chassis, and Cabinet Module formats.

Different infeeds (Basic, Smart, Active) are configured separately from the Single/Double Motor Modules.

SINAMICS S120 Combi



SITRAIN © Siemens AG 2017

Notes

Overview of functions
Number of NX/mode groups … / axes

SIEMENS

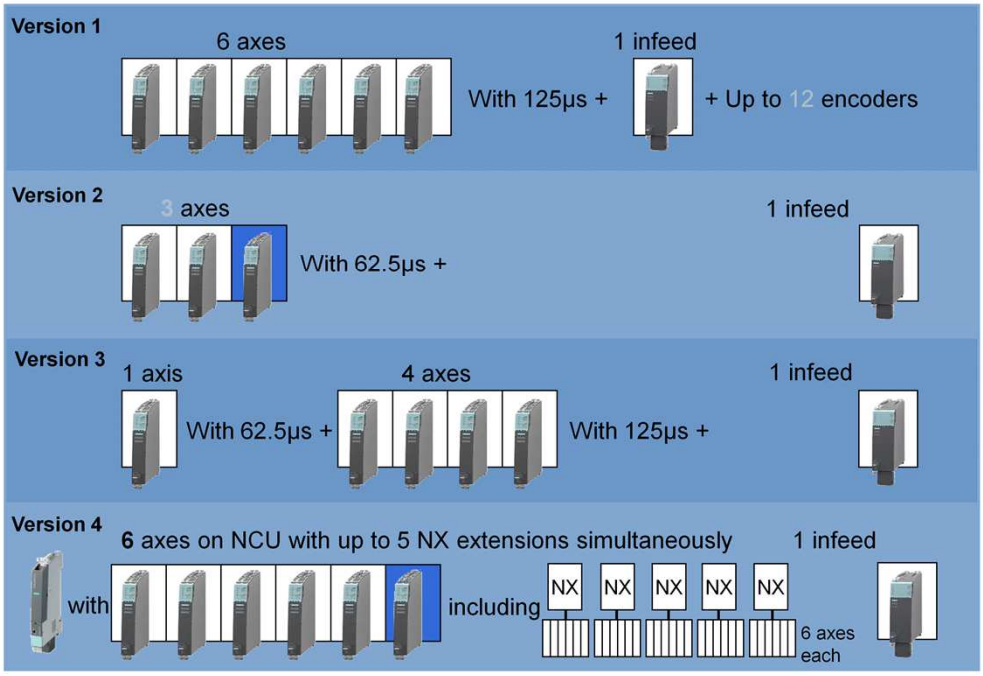
	NX	Mode groups / channels	Axes	Spindles	Axes per channel	PLC axes	Axes total
NCU 710.3 PN + Combi	0	4	6	3	6	9	15
NCU 710.3 PN + S120	3	4	8	8	8	7	15
NCU 720.3 PN	5	10	31	31	20	9	40
NCU 730.3 PN	5	10	31	31	20	19	50



SITRAIN © Siemens AG 2017

Notes

Quantity structure



Course NC-84SL-SIP Page 1 - 9 SITRAIN © Siemens AG 2017

Maximum configuration

The maximum number of drives that can be controlled by the Control Units (CU) is:

	NCU 7x0.3	NX15.3	NX10.3
Axes (125 µs)	**6**	**6**	**3**
Axes (62.5 µs)	**3**	**3**	**1**
Axes (31.25 µs)	**1**	**1**	**-**

Commissioning tools for 840D sl

Tool	Functions	Procurement
• STEP 7 +Toolbox • SINUMERIK Operate • Ghost • Alarm text converter	• PLC start-up • NC and drive commissioning • Back up HD (PCU); PCU eboot flashdrive • Conversion HMI adv. -> operate	• NC software DVD (catalog) • NCU / PCU catalog • PCU base software • NC software DVD (catalog)
Add-on software: • SinuCom NC • SinuCom CFS • Create MyConfig • Access MyMachine P2P • WinSCP • VNC Viewer • TCU menu	• NC commissioning • Back up CF card • Data for modular machines • CF card access / remove maintenance • CF card access / cmd shell • Remove operation / remote maintenance	• SinuCom program package • SinuCom program package • Catalog • Catalog • Freeware / Internet • Freeware / Internet • System software TCU
• Windows cmd shell • Linux command line	• HMI selection / Linux cmd shell • Execute Windows commands • Execute Linux commands	• Windows standard • Linux standard

Course NC-84SL-SIP Page 1 - 10 SITRAIN © Siemens AG 2017

Minimum package for commissioning

The minimum package of tools for commissioning comprises the following:

- STEP 7 on PG
- NC software DVD with: NC software, Toolbox, Alarm Text Converter, AMM P2P

Optional packages

The following additional software options are available to order:

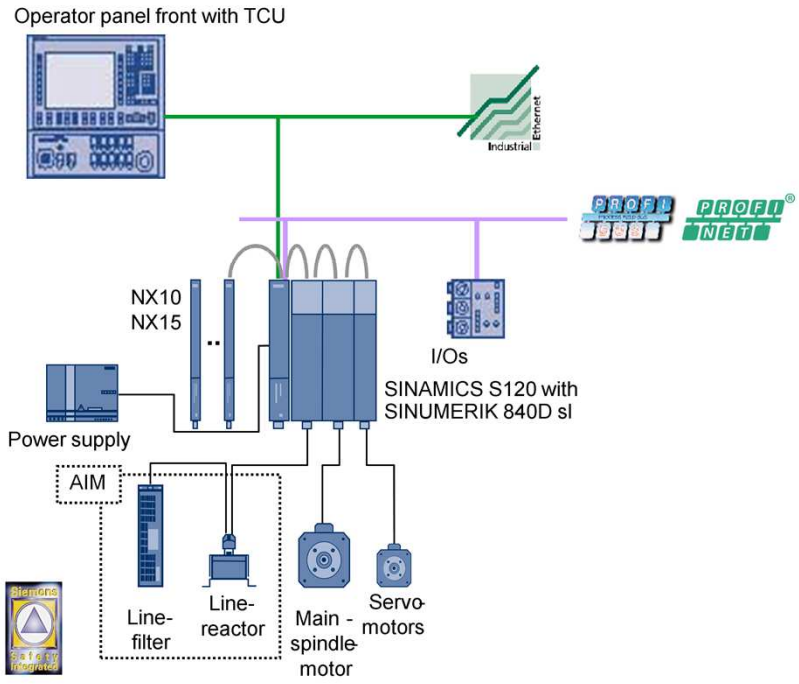
- SinuCom program package
- Create MyConfig (formerly SinuCom Installer or Update Agent)

Other tools

The following other tools are also used:

- **TCU available** Service shell of the TCU
- **No TCU** WinSCP
- **PCU available** ServiceCenter backup/restore + Ghost32 of PCU base software
- **PG/PC (Windows)** Windows cmd (+...) for network settings

840D sl basic version with TCU



Course NC-84SL-SIP Page 1 - 11 SITRAIN © Siemens AG 2017

Basic version Operator panel with adapted **T**hin **C**lient **U**nit - **TCU.**

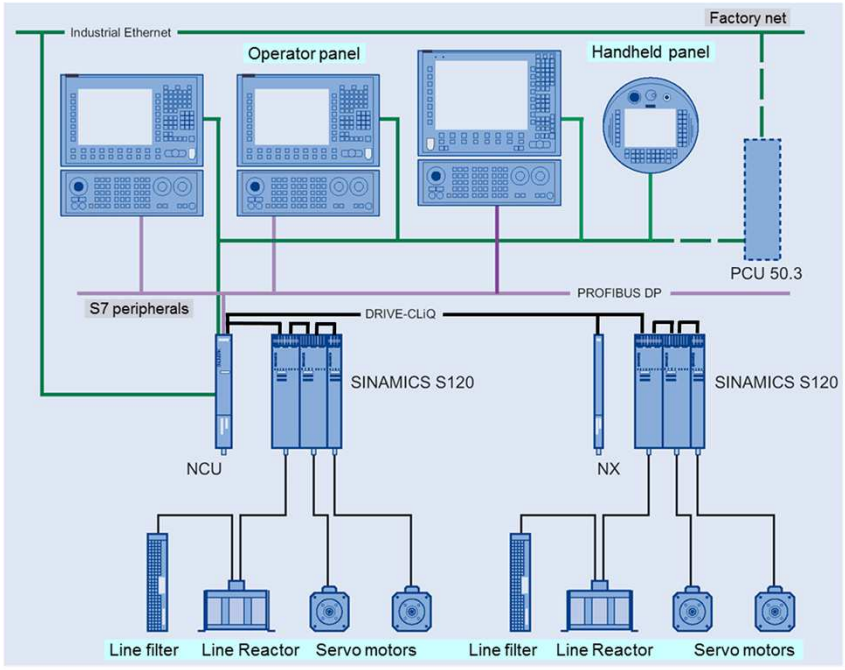
Increased HMI performance

With increased demands on the HMI performance:

Operator panel front with adapted **PCU 50.3** or **PCU 50.5**

Optional **PCU 50.3 / PCU 50.5** in the control cabinet and operator panel with adapted TCU.

840D sl networking/terminology



Course NC-84SL-SIP Page 1 - 12 SITRAIN © Siemens AG 2017

Networking

A SINUMERIK Solution Line automation solution involves several networks. An Ethernet network connects HMI control components such as operator panels, handheld units, etc.
Other Ethernet networks are designed for connection to the factory network and for connecting a programming device for when service work has to be performed.

The connection to the machine I/Os is established via a PROFIBUS or a PROFINET network.

A DRIVE-CLiQ network functions as an intelligent drive bus for the connection of the SINAMICS drive components.

Terms

Drive unit
A drive unit consists of drive objects. Each drive unit has only one Control Unit and no more than one power supply, but one or more drives.

Drive system
A drive system consists of several drive units and therefore has more than one Control Unit. A Control Unit is integrated into the NCU; others are the NX modules (NX=Numerical eXtension).

Drive object
A drive object is part of the drive unit. Drive objects are, for example, the Control Unit, the power supply, or also an axis/drive.

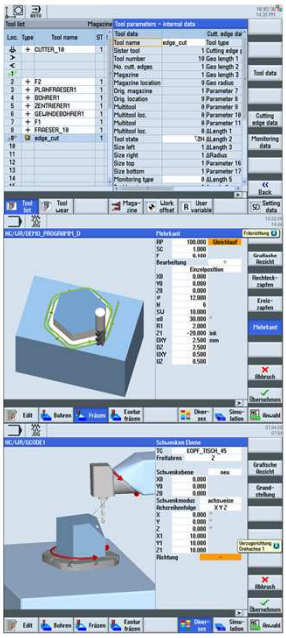
Component
A drive object comprises component(s). The Control Unit is an object but also just one component. That is also the case for the power supply (if it has a DRIVE-CLiQ connection). However, an axis/drive comprises several components such as the motor, the Motor Module, the encoder, and the Sensor Module. If an axis has 2 measuring systems, this drive object also has a 2nd encoder and a 2nd Sensor Module.

SINUMERIK Operate
The SINUMERIK operator interface

SIEMENS

With SINUMERIK Operate, SINUMERIK offers a single standard user and programming interface:

- HMI-Advanced, ShopMill, and ShopTurn combined in a single interface
- Perfect operating and programming usability
- Transparent, can be used intuitively
- Optimum display using a modern Windows style
- With new powerful functions covering all aspects of setup, programming, tool management, and program management
- CNC programming for the highest degree of productivity using programGUIDE
- New functions for complete machining (single-chuck machining)
- New functions such as multi-channel capability with ShopTurn for multi-channel machines, program synchronization with programSYNC



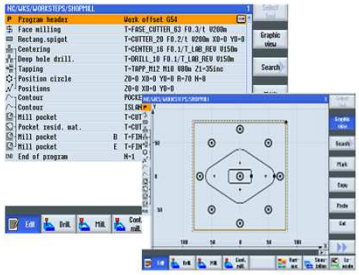
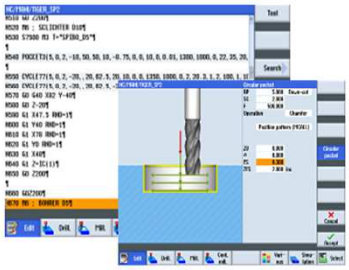
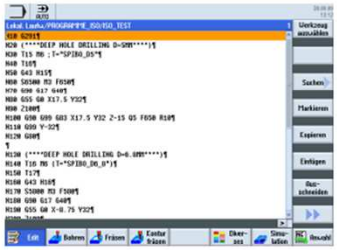
Course NC-84SL-SIP Page 1 - 13 SITRAIN © Siemens AG 2017

Notes

SINUMERIK Operate
Programming as required

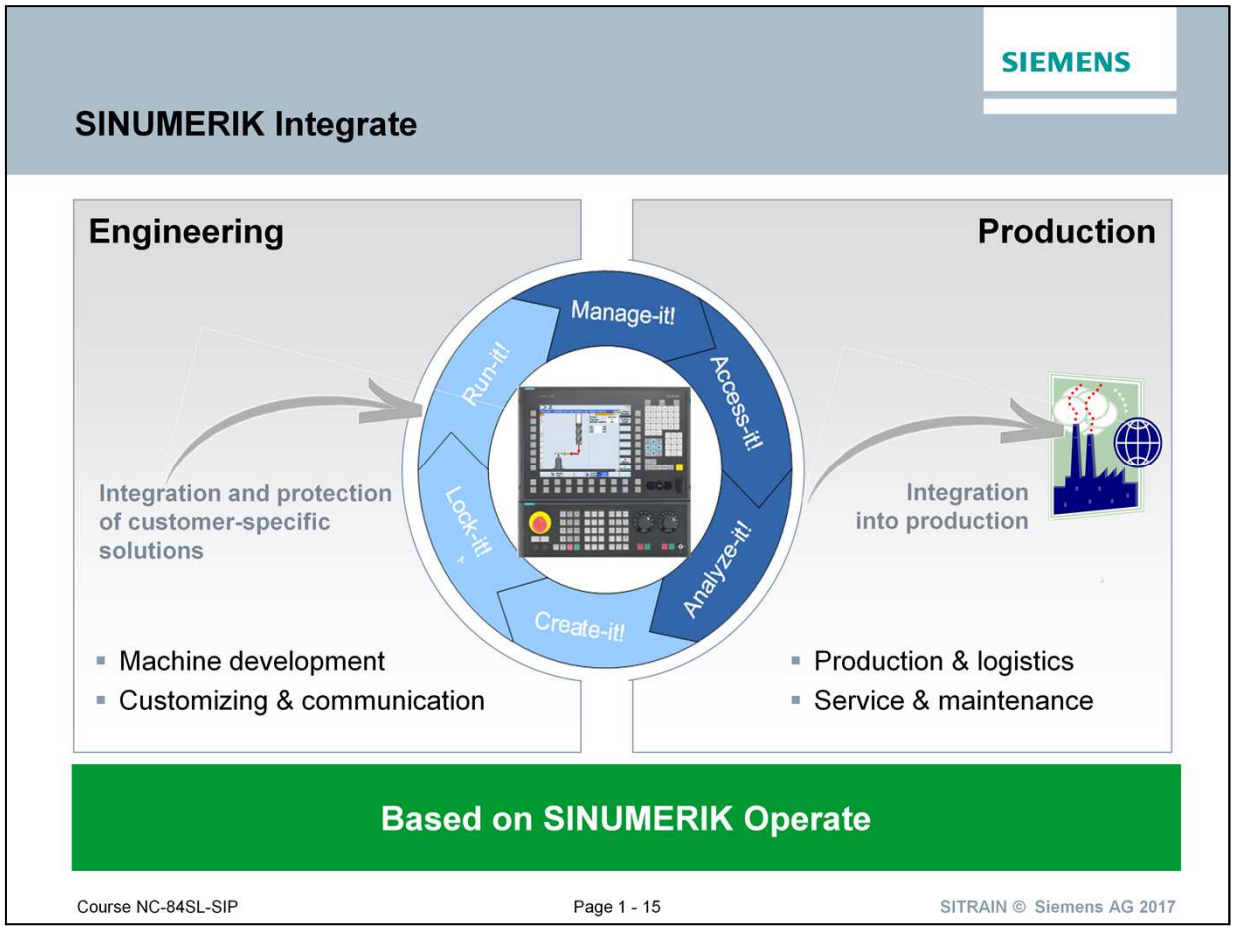
SIEMENS

Unified and simple for all programming types

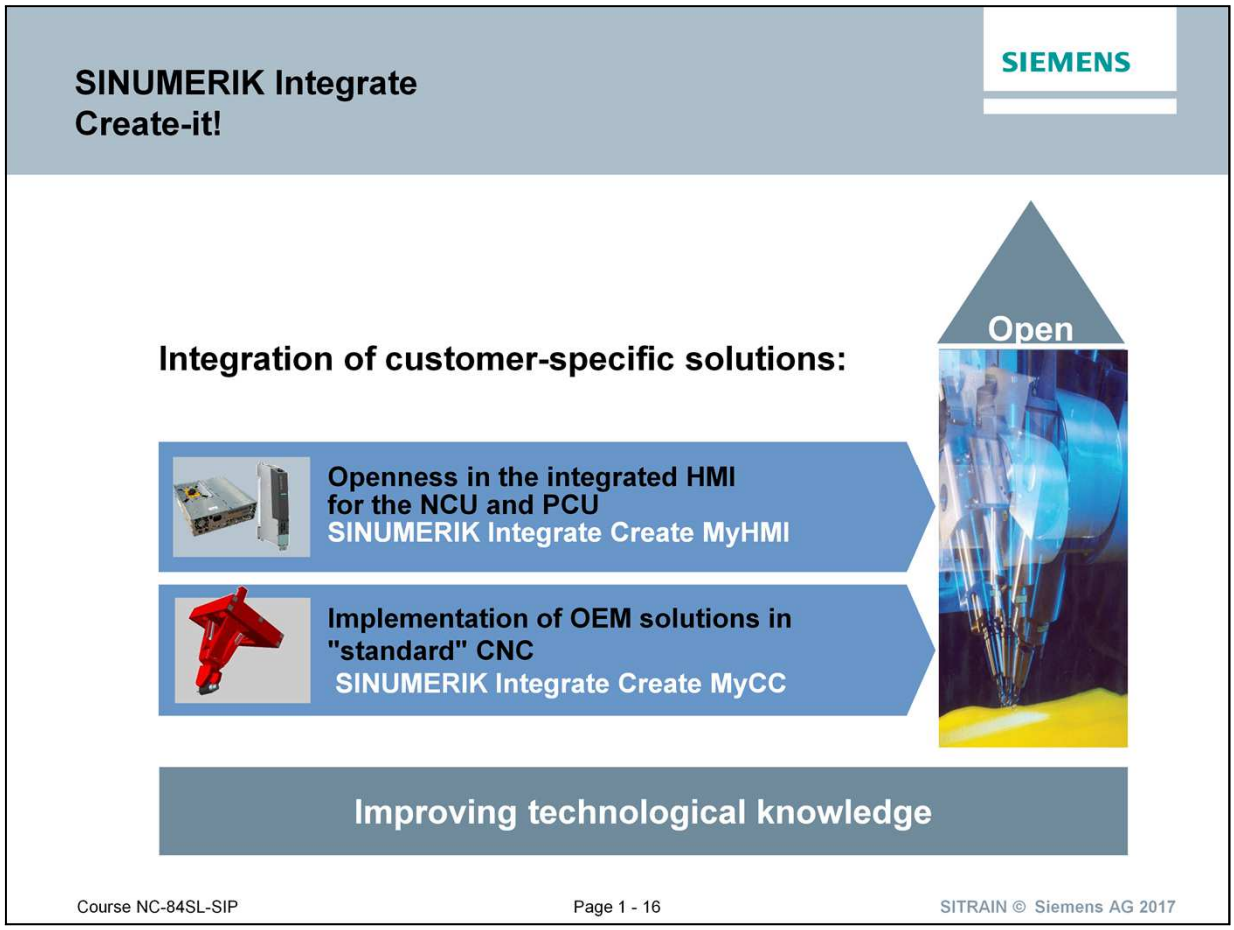
ShopMill/ShopTurn machining step programming	SINUMERIK high-level language with **programGUIDE**	**ISO code** with cycle support
		
Permits extremely short programming times **Suitable for component part and small series production**	**Permits the maximum degree of flexibility and extremely short machining times** **Suitable for average batch sizes and mass production**	**Permits the maximum ISO code compatibility to be achieved** **Suitable for average batch sizes and mass production**

SITRAIN © Siemens AG 2017

Notes



SINUMERIK Integrate	Integration of the machine into communication, engineering, and production processes based on SINUMERIK Operate. This includes the following sub-functions:
Create-it!	Customization (OA=Open Architecture) of the user interface and the NCK (configuration and/or programming) and integration of customer-specific solutions for - The user interface: Integrate Create MyHMI / 3GL (programming package), Integrate Create MyHMI / WINCC (formerly WinCC with option Sinumerik) Integrate RunMy Screens (formerly easyscreen) - NC functions: Integrate Create MyCC (Compile Cycles)
Run-it!	Runtime system for OA
Lock-it!	Know-how protection
CreateMyConfig	Configuration of modular data sets to make start-up series machines easier (previously called: SinuCom Installer).
Manage-it!	Management of programs and tools
Access-it!	Remote access/remote diagnostics; backups via network
Analyze-it!	Condition monitoring for diagnostics



SINUMERIK Integrate Create-it!

Integration of customer-specific solutions for:

- The user interface: Integrate Create MyHMI
- The NC functions: Integrate Create MyCC

SINUMERIK Integrate
Create MyHMI

SIEMENS

Application

- **Dedicated menu areas**
- **Dedicated screens/screen areas**
- **Technological calculations**

- Same project for NCU and PCU
- In many cases, PCU 50 is not required



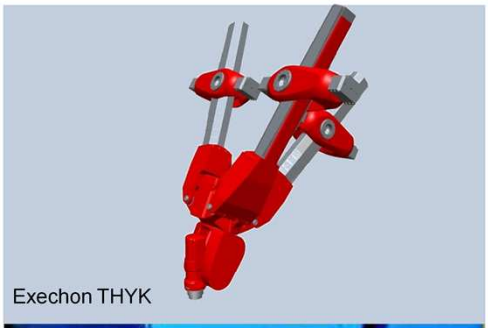
OEM-specific solutions already implemented in the "standard"

SITRAIN © Siemens AG 2017

Notes

Application

- **Special kinematic transformations**
  - Exechon THYK
  - Handling RCTRA
  - Doubletransmit DTRA
  - Crank interpolation CRIP
  - ...



Exechon THYK

- **Technological special functions**
  - Clearance control CLC
  - Tool monitoring TPM-PB
  - Retrace support RESU
  - Axis coupling AXCO
  - ...

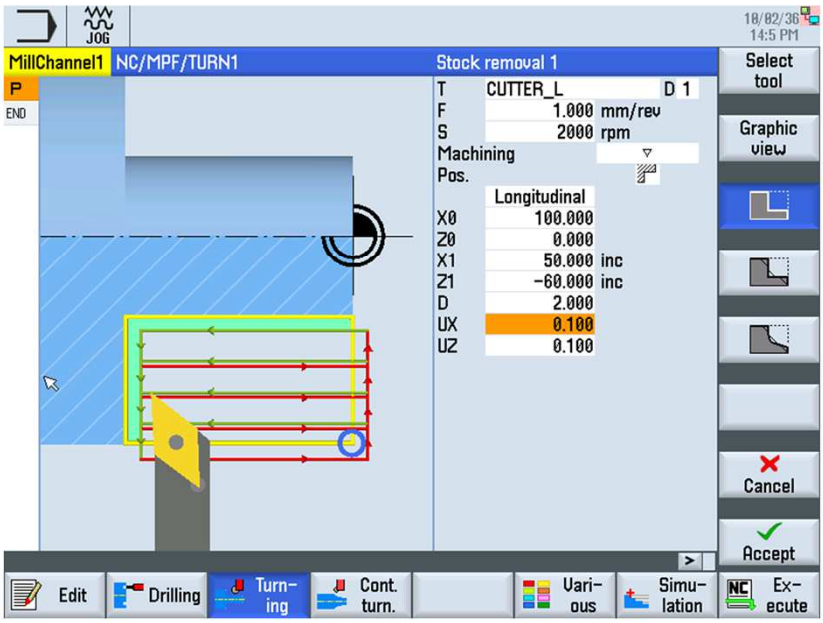


Implementation of OEM-specific solutions in the "standard" CNC

SITRAIN © Siemens AG 2017

Notes

Run MyScreens



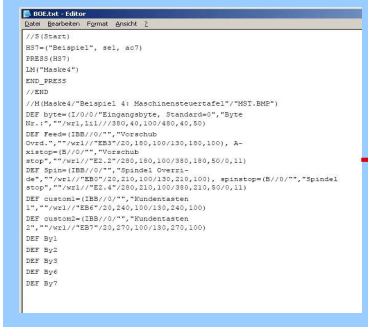
 SITRAIN © Siemens AG 2017

Expand user-specific screens (continuation of "Expand user interface")

- Option P64; for 6 or more configured screens; fewer are free of charge
- In the operating areas (Machine, Parameters, etc.)
- As complete operating area
- For cycle support (programGUIDE)
- Configuration as for "Expand user interface" with ASCII editor
- Forward and backward compatible

Powerline

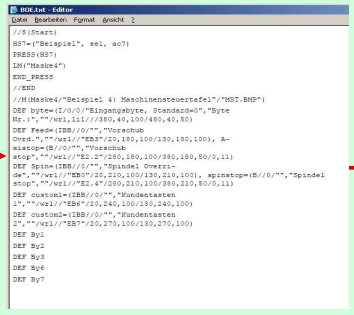
Expand user interface

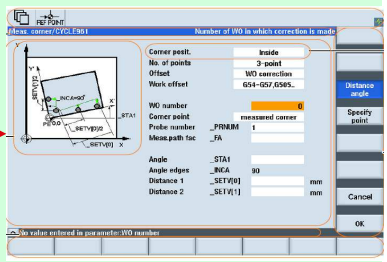


Largely com-patible

solution line

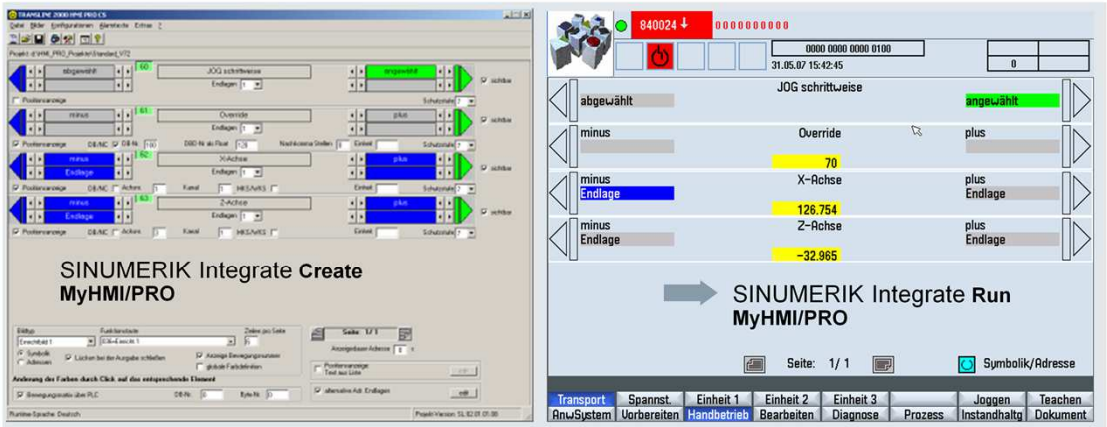
SINUMERIK INTEGRATE
Run MyScreens





HMI Pro
Machine operation in the automotive industry

SIEMENS



- Simple graphical configuration of operating screens with SINUMERIK Integrate Create MyHMI/PRO
- Standardized operating screens and diagnostics
- Graphical editor to create user displays
- Projects are executable on SINUMERIK PCU 50 and NCU 7x0
- Migration of projects from HMI Advanced/HMI PRO

SINUMERIK Integrate Create MyHMI/PRO is the user interface for the Automobile Powertrain

SITRAIN © Siemens AG 2017

Product Machine user interface for operation and diagnostics in mass production

Application Automotive projects

- Transfer lines
- Flexible production lines
- NC series and special-purpose machinery
- Assembly lines
- Linked production
- Rotary indexing machines

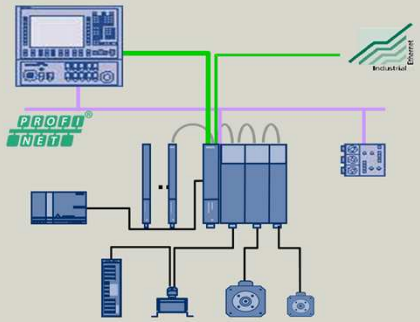
What do I find where in the folder?

Topic	Parts / Examples	Chapter
• Hardware • Networks and tools • Data backup	• NCU/NX, SINAMICS, OP/PCU/TCU • WinSCP, **s**ervice **c**ommand sc • RAM memory (NCU) • CF card (NCU), AMM P2P, hard disk PCU	• Hardware • Networks and tools Data backup with Operate Data backup of cf card Data backup with PCU 50
• Diagnostics • Commissioning	• Diagnostics OA, Setup, OFF1/2/3 • Overview of machine data, BICO, optimization • NX commissioning, topology changes	• Diagnostics with Operate • Commissioning basics • Basic commissioning and commissioning axes • Advanced commissioning
• Sinumerik Operate	• Operation, basic settings, tool management, settings for Operate	• Operation MCP / Operate Basic settings tool management Adaptation Sinumerik Operate
• PLC functions • NC functions	• Basic program, alarm texts • Settings/modifications Leadscrew error comp. • Gantry, synchronous spindle, master/slave	• PLC interface • CNC adaptation • Compensations • Axis couplings

SITRAIN © Siemens AG 2017

Notes

SIEMENS



Chapter 2

Hardware

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 1
840D sl: NCU and NX 2
NCU 3
SINAMICS 15
Operator panel fronts and control computers 43
Machine control panels, hand-held controllers 44
PCU 50.5 45
PCU 50.3 48
Multiple operator panels on n NCUs and m PCUs 49
Machine control panels 50
HT8 53
HT8 – HT2 54
Handwheel connection module 56

Learning objectives

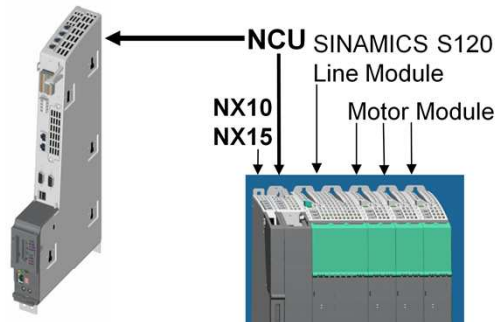
- You will know the hardware components required to build an 840D sl



 SITRAIN © Siemens AG 2017

Notes

840D sl: NCU and NX



System	NCU710.3	NCU720.3	NCU730.3	NX10.3/15.3
Processors	3	3	3	1
Drive	x	x	x	x
NC/HMI	x	x	x	-
PLC	317F-3 PN/DP	317F-3 PN/DP	317F-3 PN/DP	-
Max. number of channels	4	10	10	-
Max. number of NC axes/sp.	8 (6) Max. 2 x NX15	31 Max. 5 x NX15	31 Max. 5 x NX15	6
High-speed I/O terminals	12DI, 8DI/DO NC: 4 DI, 4DO	12DI, 8DI/DO NC: 4 DI, 4DO	12DI, 8DI/DO NC: 4 DI, 4DO	4 DI, 4 DI/DO
DRIVE CLiQ connections	4	6	6	4/6
x available - not available				

Course NC-84SL-SIP Page 2 - 3 SITRAIN © Siemens AG 2017

As of NCUs **7x0.3** and NX modules **NX 10.3/NX15.3** a new drive processor was introduced.

These modules are no longer software-compatible with NCU 7x0.2 / NX10 / NX15!

NCU 7x0.2 Only runs with SW V <= 2.7

NCU 7x0.3 Only runs with SW V >= 4.4

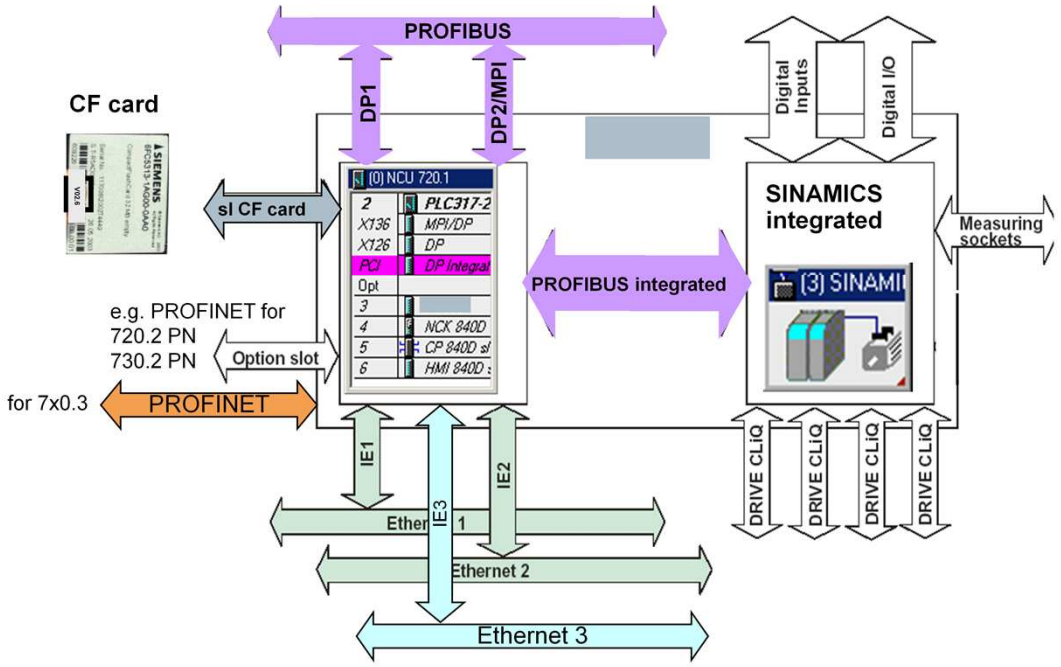
The x.3 NCUs are only available in the version with a PROFINET interface. For accelerated graphics, a dedicated processor kernel is used for the integrated HMI (SINUMERIK Operate). THE NCK/HMI CPU time split changed to 90 %/10 % from the 65 %/35 % split of 7x0.2 NCUs.

All NCU 7x0.2/.3 devices have a controlled dual fan module. This results in reduced air noise and increased service life.

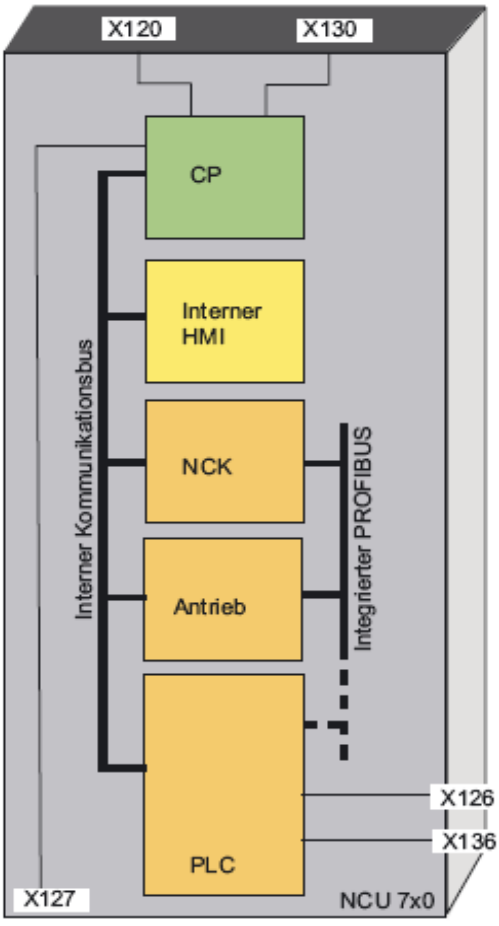
The 720.3 PN and 730.3 PN NCUs have a 317F-3PN/DP PLC for increased performance compared with predecessor modules.

NCU
Internal structure

SIEMENS



 SITRAIN © Siemens AG 2017

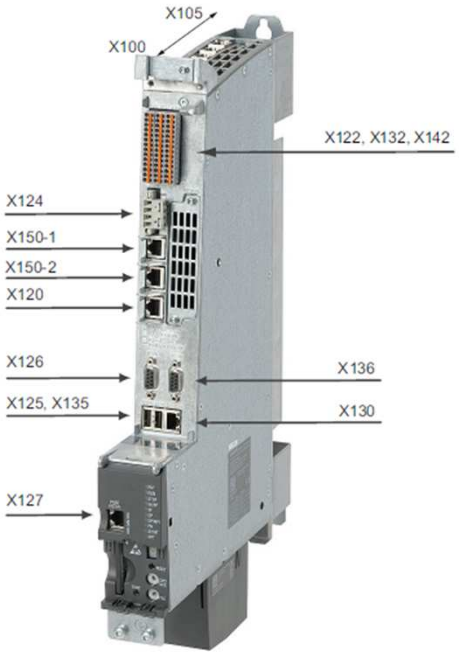


X120 Standard Ethernet – machine network
X127 Standard Ethernet – service network
X130 Standard Ethernet – factory network
X126 Profibus DP 1
X136 Profibus DP 2

NCU
7x0.3 view from above

SIEMENS

X100 ... X105	DRIVE-CLiQ interfaces
X124 +24 V	electronics power supply
X125, X135	2x USB interface, X125(left), X135(right)
X120	Industrial Ethernet to connect the machine network (e.g. TCU and/or PCU)
X130	Industrial Ethernet to connect the company network
X127	Industrial Ethernet service interface for PG/PC
X136	PROFIBUS DP / MPI
X126	PROFIBUS DP (e.g. for PLC-axis)
X150-1, X150-2	PROFINET IO for PROFINET-components
X122, X132, X142	Digital in-/output for peripherie-signals



Course NC-84SL-SIP Page 2 - 5 SITRAIN © Siemens AG 2017

X120 / X130 / X127: **PROFINET not supported!**

Contrary to the designation for X120 / X130 / X127 ("PN/IE-OP", etc.) PROFINET is not supported on these interfaces.

PN ports On PROFINET X150 interfaces, P1 / P2 designate the two ports port 1 and port 2 respectively.

Hardware X150 with integrated 2-port switch

'Autocrossing': Automatic detection of cable type (a crossover cable is not required).

Functions The following functions are supported:

- **PROFINET CBA:**

Communication between controllers as components in distributed systems.

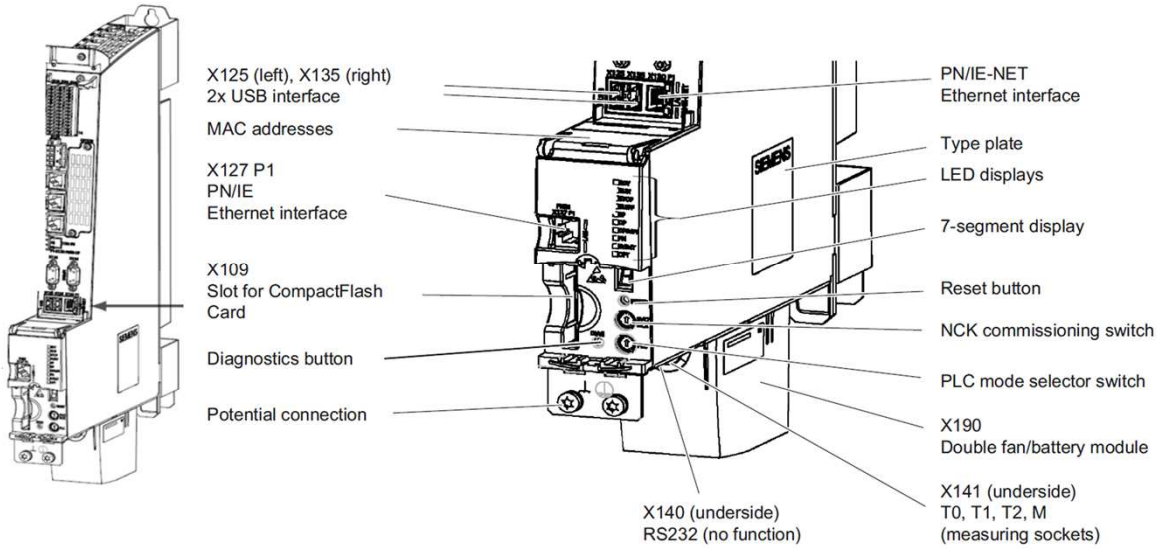
- **PROFINET IO:**

Communication between the PLC, the controller, and field devices;

- Restrictions:
- No access from NCK
- No isochronous mode (PROFINET IRT)

NCU
7x0.3 view from below

SIEMENS



Course NC-84SL-SIP Page 2 - 6 SITRAIN © Siemens AG 2017

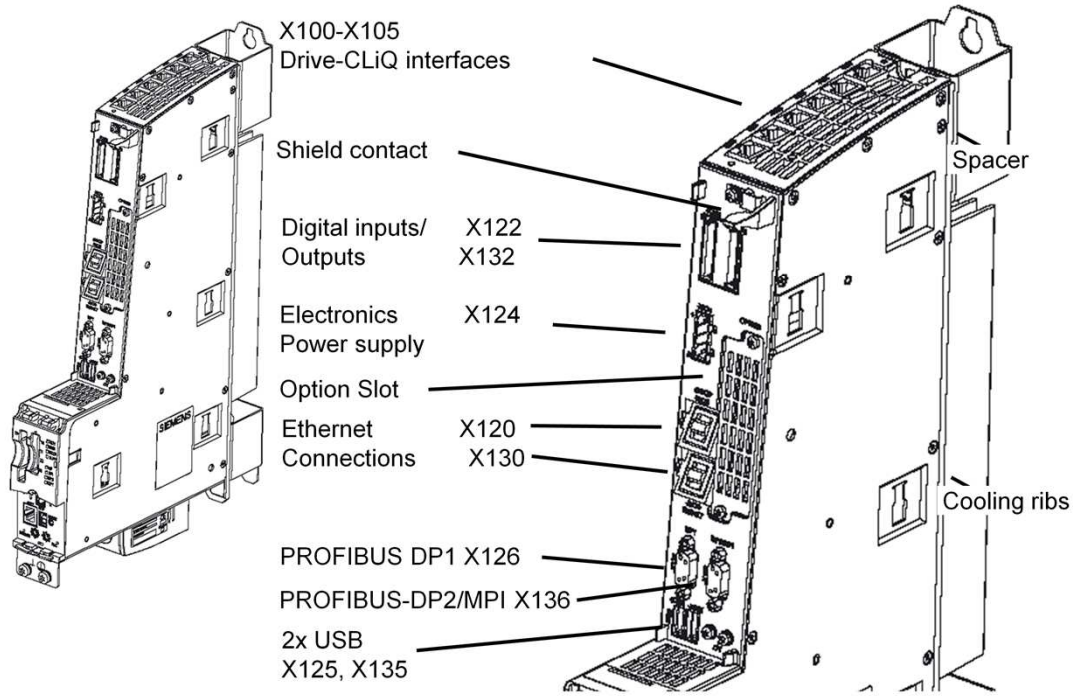
USB X125, X135 USB 2.0 interface
The type A dual USB socket can support a load of up to 0.5 A on each channel.

MAC addresses



NCU
7x0.1 / 7x0.2 view of upper part

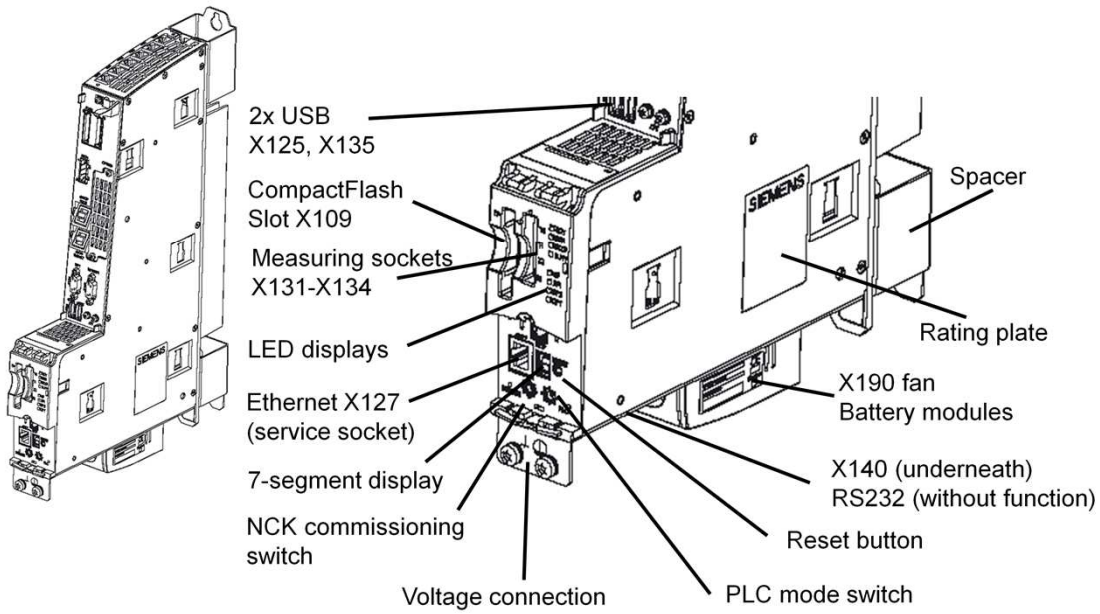
SIEMENS



Course NC-84SL-SIP Page 2 - 7 SITRAIN © Siemens AG 2017

Notes

NCU
7x0.1 / 7x0.2 view of lower part

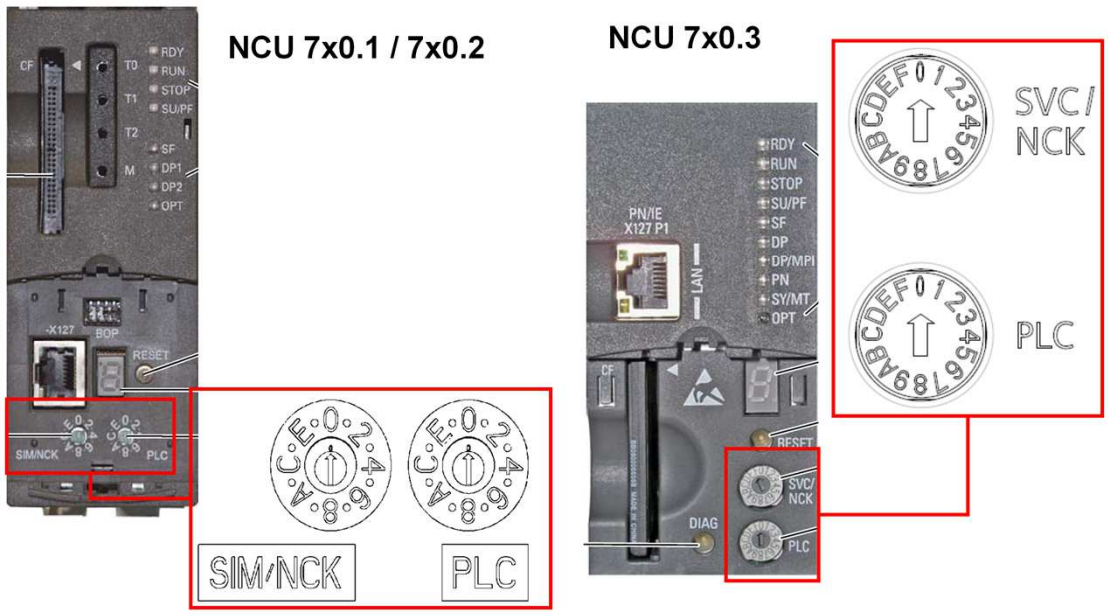


SITRAIN © Siemens AG 2017

Notes

NCU
Commissioning switch and reset

SIEMENS



SITRAIN © Siemens AG 2017

'SIM/NCK' or 'SVC/NCK' switch:

NCK commissioning switch

Position 0:	Normal operation
Position 1:	NCK general reset
Position 2:	SRAM (NC+PLC) is restored to CF card from memory dump
Position 7:	NCK not started during run-up
Position 8:	Display the IP address on X130

'PLC' switch

PLC operating mode switch

Position 0: PLC-RUN-P

Position 1: PLC RUN

Position 2: PLC STOP

Position 3: MRES (general reset request/general reset)

Position 5: Flash PLC bootcode
(if necessary, display PLC on 7-segment display)

'RESET' button

Restart of all systems (warm restart):

- Linux
- SINUMERIK Operate
- NCK
- PLC
- Drives

Changes to hardware-level data may require a complete POWER ON (e.g. cycle times, ALM configuration, loading of SINAMICS default data, firmware update, etc.)

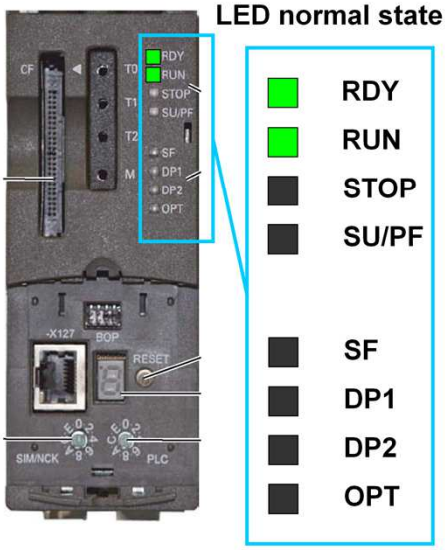
'DIAG' key

Without function

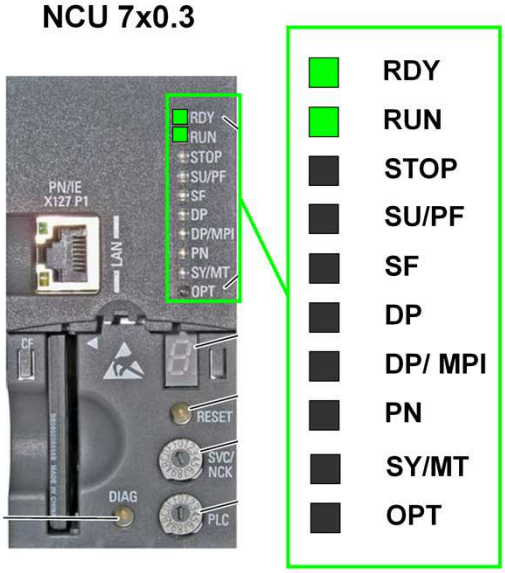
NCU
LED displays

SIEMENS

NCU 7x0.1 / 7x0.2



NCU 7x0.3



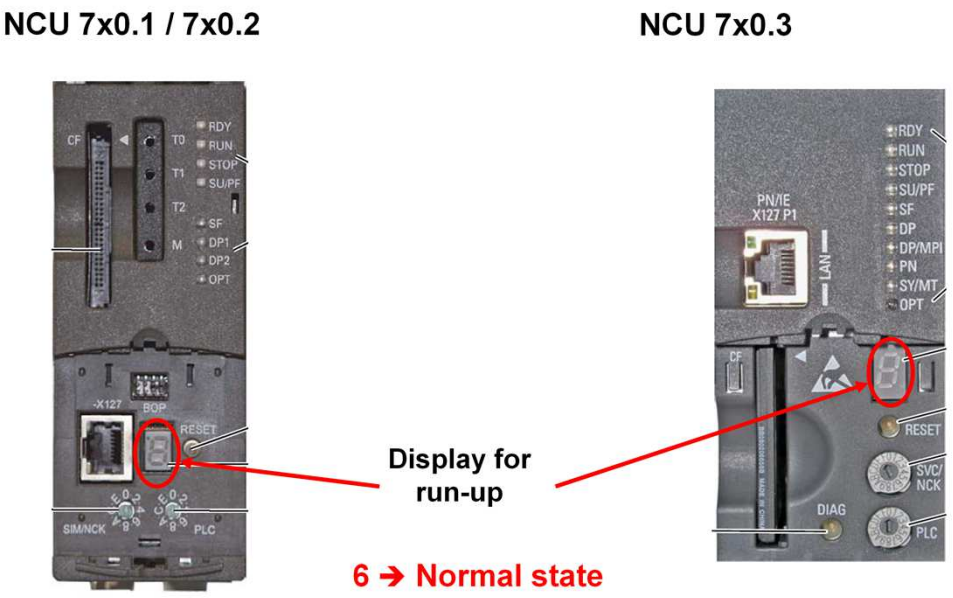
Course NC-84SL-SIP Page 2 - 10 SITRAIN © Siemens AG 2017

LED

Name	Function	Status	Meaning
RDY	Ready	Red	There is at least one fault (e.g. RESET, watchdog monitoring etc.) or the Control Unit is booting up.
		Flashing red/orange (0.5 Hz)	Error accessing CompactFlash Card
		Orange	Accessing CompactFlash Card
		Flashing orange (0.5 Hz)	Updating the firmware of the connected DRIVE-CLiQ components
		Flashing orange (2 Hz)	Firmware update is complete for components. Wait for POWER ON for the components in question.
		Green	NC powered up and everything in cyclic mode
		Flashing green/orange or red/orange (1 Hz)	LED-supported recognition of connected DRIVE-CLiQ component is activated: (p0124[0] = 1).
RUN	PLC RUN	Green	PLC ready to operate
STOP	PLC STOP	Orange	PLC stopped
SU/PF	PLC FORCE	Red	FORCE activated
SF	PLC SF	Red	PLC group error
DP1	BUS1 F	Red	PROFIBUS-DP1 group error
DP2	BUS2 F	Red	PROFIBUS-DP2 group error
OPT *)	BUS3 F	Off	The PROFINET system runs correctly; data exchange to all configured IO devices runs.
		Red	• Bus fault (no physical connection) • Incorrect transmission rate
		Flashing red (2 Hz)	• Failure of a connected I/O device. • Incorrect or no configuration.

NCU
7-segment display

SIEMENS



Course NC-84SL-SIP Page 2 - 11 SITRAIN © Siemens AG 2017

7-segment display

The 7-segment display is located behind the blanking plate of the Control Unit. The most important states are as follows:

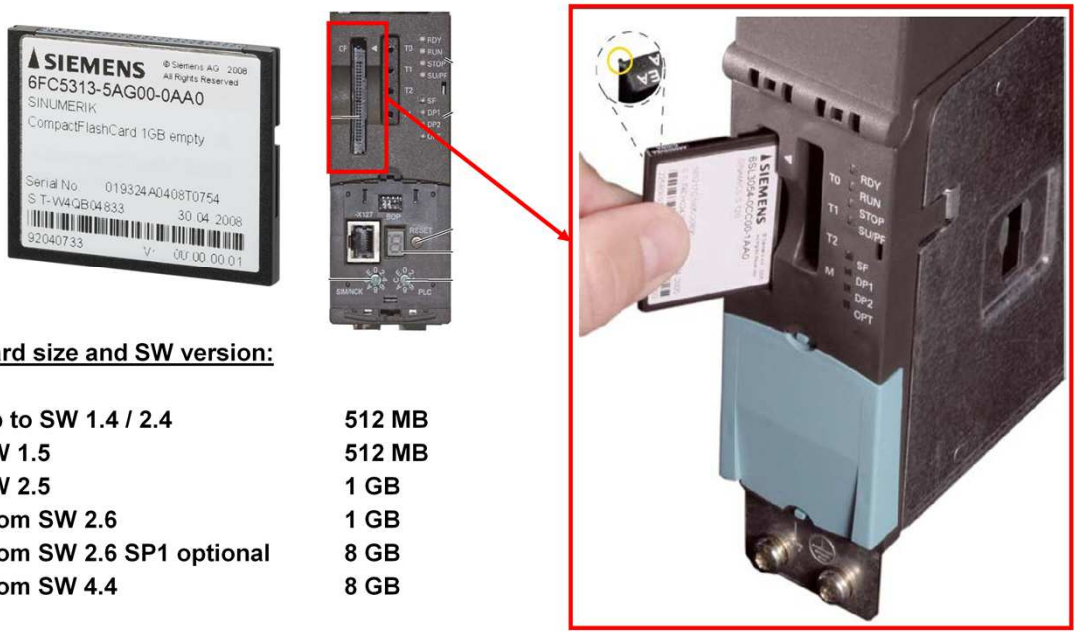
6 Normal state

5 PLC stop

8 Battery/fan module missing or faulty (NCK stopped)

NCU
CF card

SIEMENS



Card size and SW version:

Up to SW 1.4 / 2.4	**512 MB**
SW 1.5	**512 MB**
SW 2.5	**1 GB**
From SW 2.6	**1 GB**
From SW 2.6 SP1 optional	**8 GB**
From SW 4.4	**8 GB**

Course NC-84SL-SIP Page 2 - 12 SITRAIN © Siemens AG 2017

CF card

The CompactFlash card is shipped in a bootable condition. It is not supplied with the Control Unit and must be ordered as a separate component.

The CompactFlash card is inserted in the CF plug-in slot (X109 interface).

Notice

The CompactFlash card must only ever be inserted or removed when the Control Unit is disconnected from the power supply.

Data

The CompactFlash card is necessary for operation of the Control Unit.

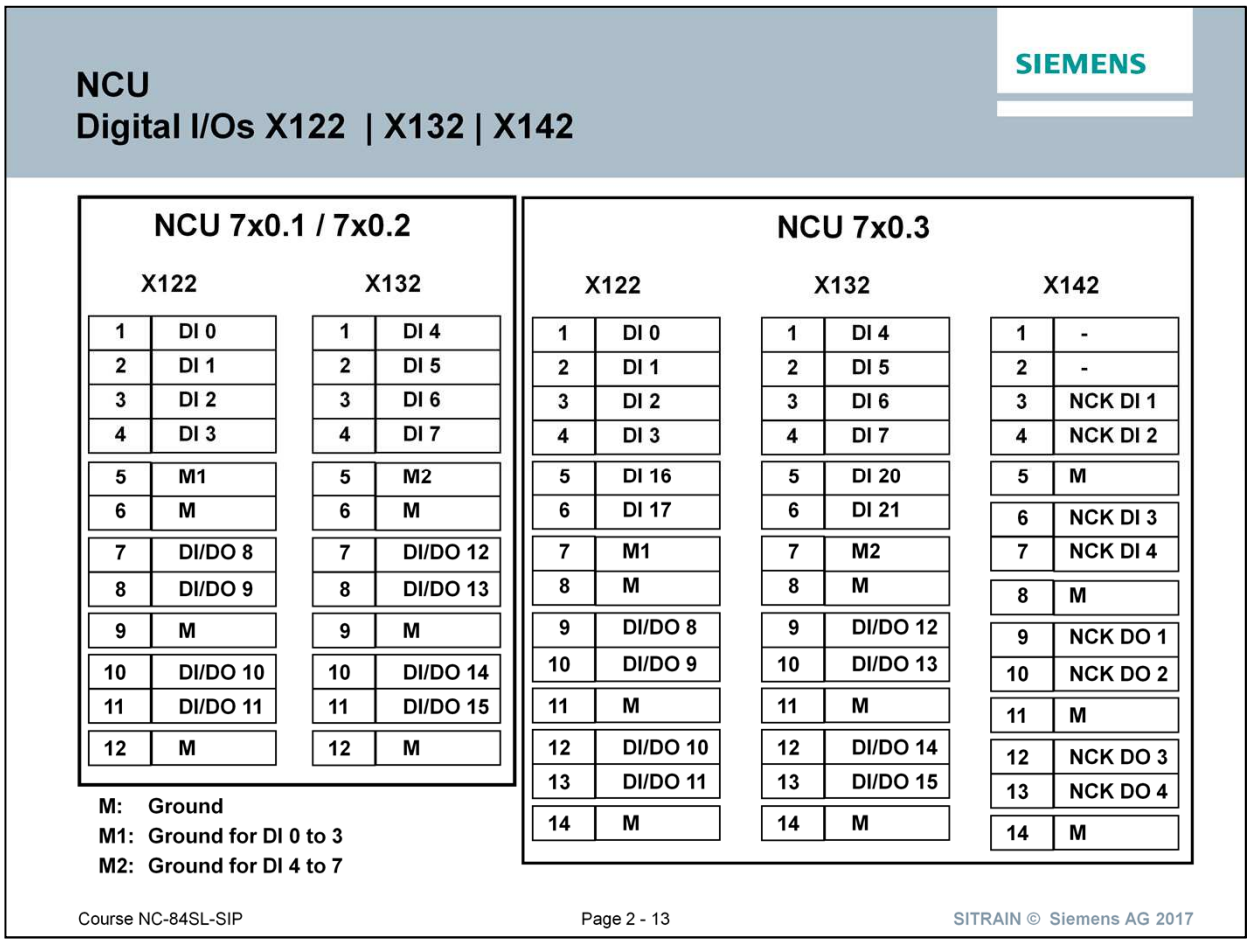
As well as the basic software for SINUMERIK and the firmware for SINAMICS, the CompactFlash card also contains:

- User data (programs, configuration data, parameter settings)
- Version information (serial number, version, type designation)
- License key

This means that the CompactFlash card can be inserted into another NCU without having to change the license.

Inserting the card

1. Switch off the power supply.
2. Gently insert the new CompactFlash card into the plug-in slot until it clicks into place (web is on left-hand side, see photo).
   When properly installed, the card does not protrude beyond the housing.
3. Switch the power supply on again.



Overview of digital inputs/outputs

DI 0 … 7 Digital inputs drive; status: p722 (CU) bits 0 to 7

DI 16/17 Digital inputs drive; status: p722 (CU) bits 16/17

DI 20/21 Digital inputs drive; status: p722 (CU) bits 20/21

DI/DO 8 … 15 Digital inputs/outputs, drive

- Input/output configuration: p728 (CU)

- Signal assignment: p738 … p745 (CU)

NCK DI 1 … 4 Digital inputs NCK $A_IN[1] to $A_IN[4]

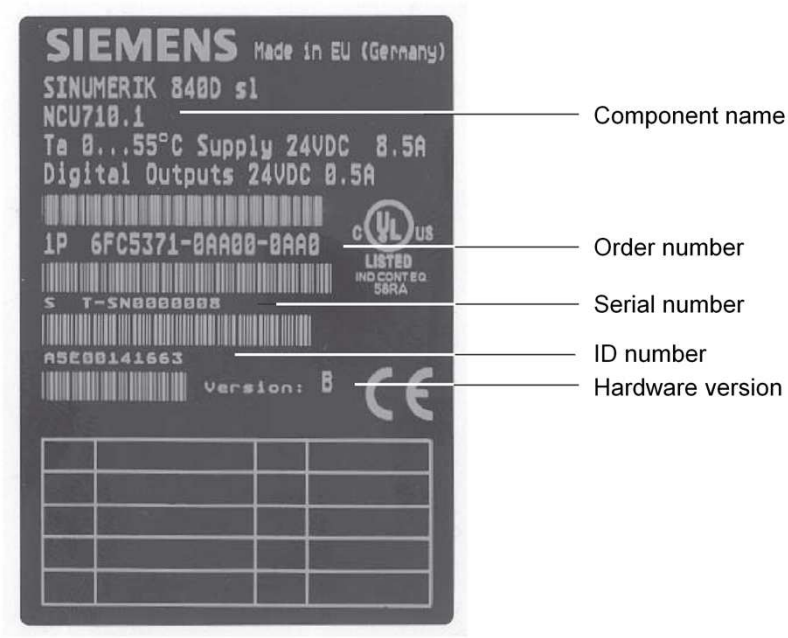
- Permanently assigned (NCU 7x0.3 only)

NCK DO 1 … 4 Digital outputs NCK $A_OUT[1] to $A_OUT[4]

- Permanently assigned (NCU 7x0.3 only)

Note Standard assignment of the inputs/outputs is part of device configuration during commissioning. It is therefore described in the chapters on commissioning.

NCU
Rating plate

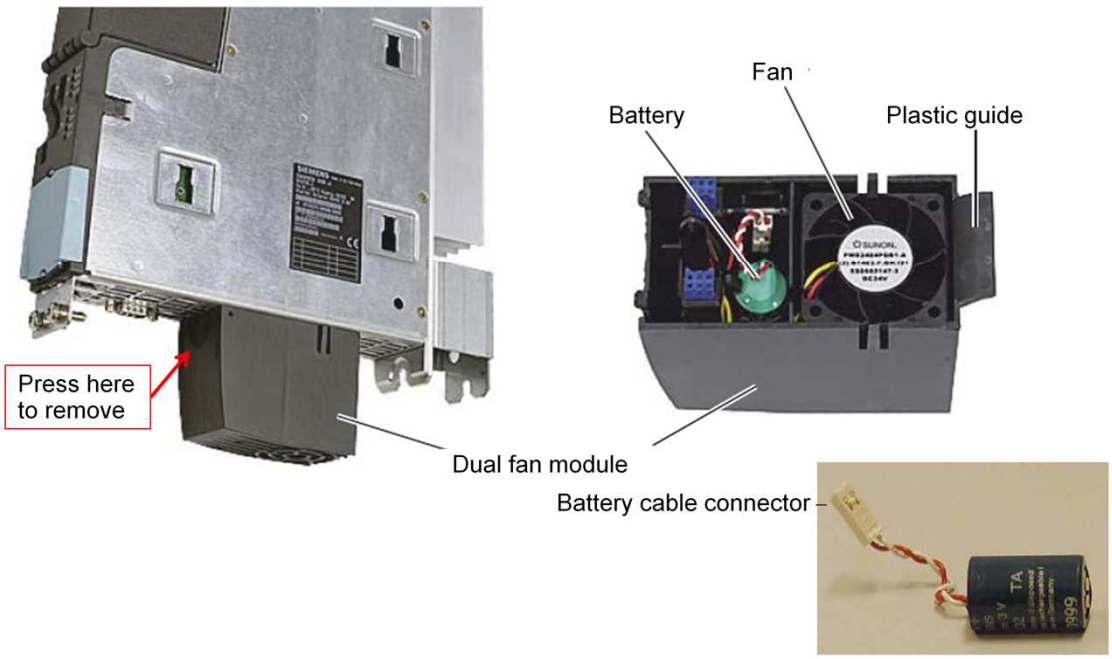


Course NC-84SL-SIP Page 2 - 14 SITRAIN © Siemens AG 2017

Electronic rating plate

All information on the rating plate is also available electronically on the NCU.

NCU
Battery/fan module



Course NC-84SL-SIP Page 2 - 15 SITRAIN © Siemens AG 2017

Tasks

The fan/battery module performs the following tasks:

- Cooling the CPU
- SRAM backup (if the capacitor is insufficient)

The Control Unit monitors the temperature inside the module and the functioning of the fan. The fan is switched on automatically if the temperature exceeds 40 °C. Fan faults are displayed and can be read out by means of the diagnostic buffer.

The **SF LED** shines **"red"** in the event of **temperature or fan faults**.

Replacing the module

The battery and fan module has an elastic lock on the front.
When this is pressed, you can fold the module outward and downward.
Insertion is performed by inserting the rear plastic rail into the corresponding slot into the NCU and folding it up.

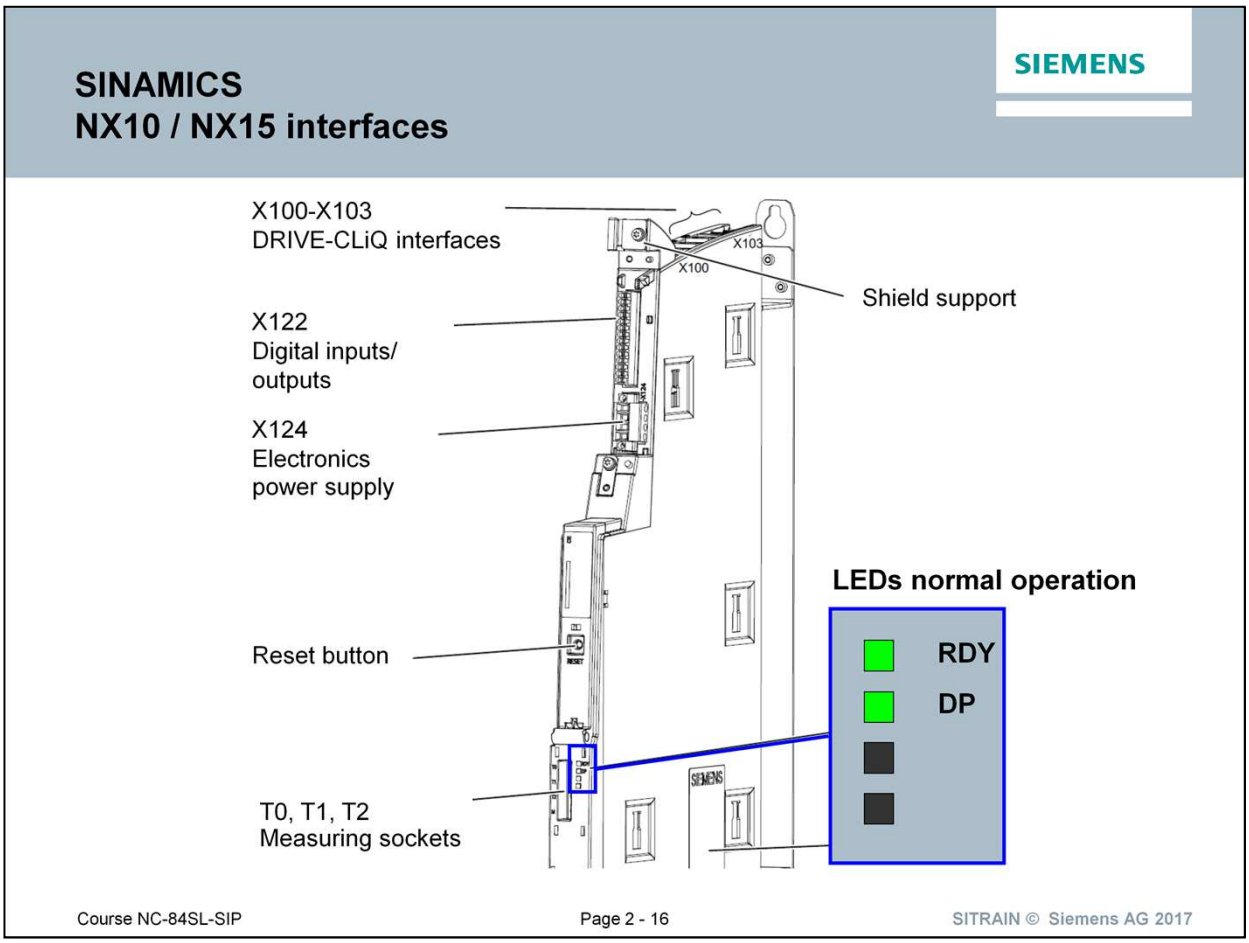
Notice

If the battery/fan module has no contact or is missing completely, all LEDs on the front light up yellow. The Control Unit cannot be operated without the fan, i.e. the Control Unit will not power up without a functioning fan/battery module!

Note: Replacement during operation

The fan/battery module can be replaced during operation.

If you choose to do this, remember that the **Control Unit can only be operated for a maximum of 1 minute without fans**. If this time is exceeded, the Control Unit will shut itself down.



LED (NX 10.3 / NX 15.3)

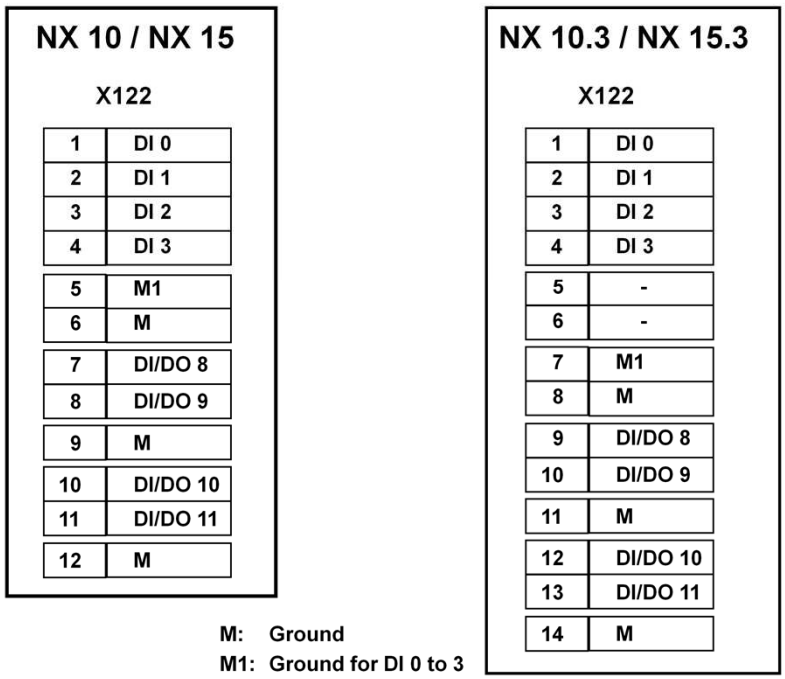
LED	Color	Status	Description
RDY	Off		Electronic power supply outside permissible tolerance range
	Green	Continuous light	NX is ready for operation
		Flashing light 2 Hz	Writing to CompactFlash card of the connected NCU
	Red	Continuous light	NX is presently booting and at least one fault is present (e.g. RESET, watchdog monitoring, basic system fault).
		Flashing light 0.5 Hz	Boot error (e.g. firmware cannot be loaded into the RAM)
	Orange	Continuous light	Firmware loading into RAM
		Flashing light 0.5 Hz	Unable to load firmware into RAM
		Flashing light 2 Hz	Firmware CRC fault
DP	Off		Electronics power supply outside the permissible tolerance range, NX is not ready to run.
	Green	Continuous light	CU_LINK is ready for communication and cyclic communication is running.
		Flashing light 0.5 Hz	CU_LINK is ready for communication and no cyclic communication is running.
	Red	Continuous light	At least one CU_LINK fault is present. CU_LINK not ready for operation (e.g. after switching on)

Note On the NX 10 and NX 15 (instead of the NX 10.3 / 15.3), the LED is still called 'DP1'.

Reset button For rebooting the module.

SINAMICS
NX10 / NX15 pin assignment I/O

SIEMENS



M: Ground
M1: Ground for DI 0 to 3

Course NC-84SL-SIP Page 2 - 17 SITRAIN © Siemens AG 2017

Overview of digital inputs/outputs

DI 0 … 3	Digital inputs drive; status: p722 (CU) bits 0 to 7
DI/DO 8 … 11	Digital inputs/outputs, drive

- Input/output configuration: p728 (CU)

- Signal assignment: p738 to p741 (CU)

Note Standard assignment of the inputs/outputs is part of device configuration during commissioning. It is therefore described in the chapters on commissioning.

SINAMICS
S120 booksize components

SIEMENS

840D sl as the central Control Unit

Line Modules for the power supply:
- Smart Line Module 400 V, up to 55 kW
- Basic Line Module 400 V up to 100 kW
- Active Line Module 400 V, up to 120 kW

One or more **Motor Modules** to control the motors
- Single Motor Modules up to 200 A
- Double Motor Modules up to 2x18 A



Optional DC link components
- Braking Module
- Braking resistors
- Capacitor Module

Electronic options

Sensor Modules:	SMC10 (Resolver) SMC20 (Encoder) SMC30 (TTL/HTL)
Terminal Modules:	TM15, TM31, TM41



Course NC-84SL-SIP Page 2 - 18 SITRAIN © Siemens AG 2017

Active / Smart / Basic Line Module

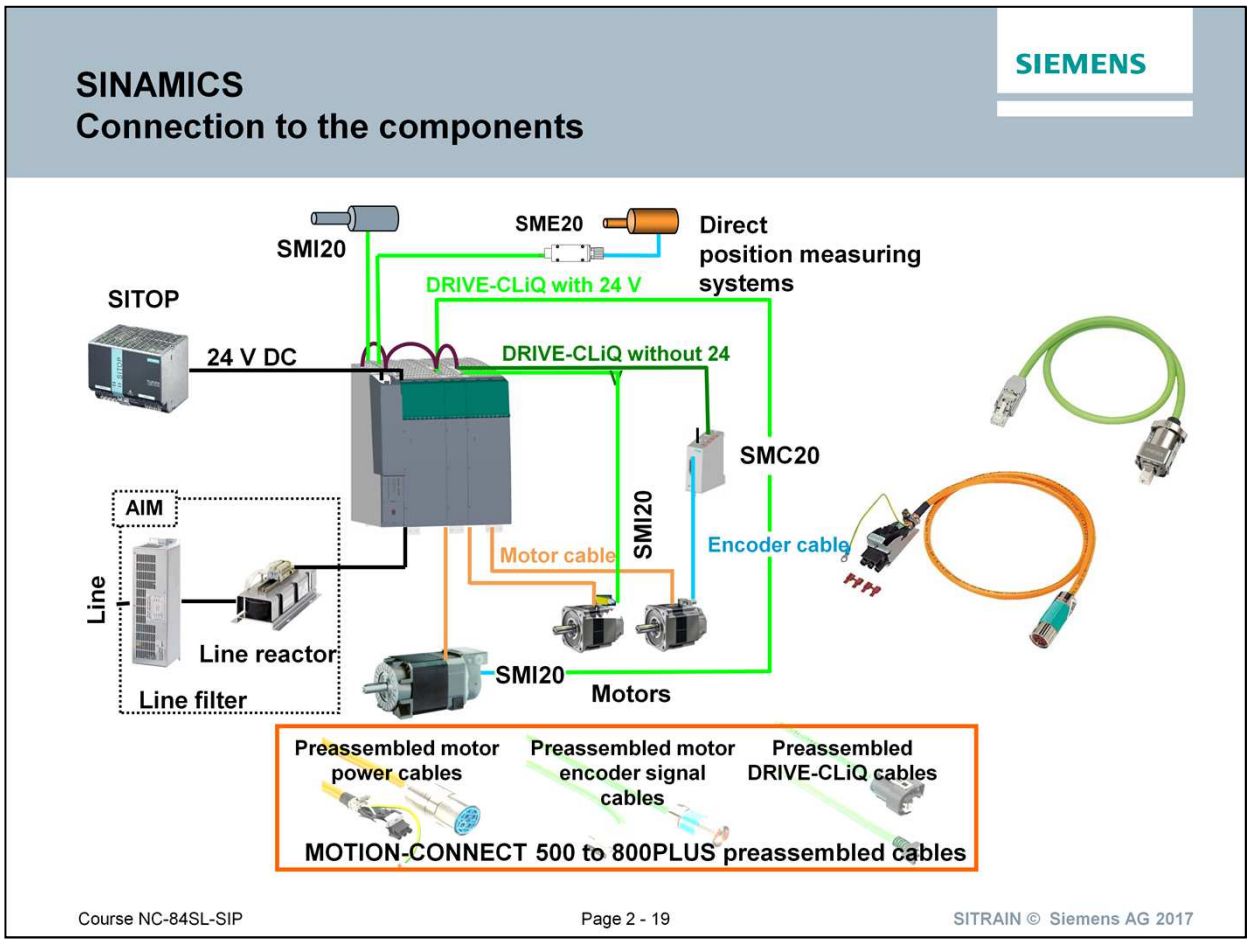
Line Modules provide the central power supply to the DC link. They convert the line alternating voltage (fixed frequency) into direct voltage.

Single/ Double Motor Module

Motor Modules draw their power from the DC link to supply the connected motors with alternating voltage at variable frequency.

SMC10/20/30 Sensor Modules Cabinet-Mounted-are used when a motor with a DRIVE-CLiQ interface is not available and when external encoders are required in addition to the motor encoder.

SME20/25/120/125 Measuring systems outside the cabinet can be connected directly to the Sensor Module External.



Faster and simple commissioning with SMI

Automatic detection of motor and encoder

Storage of technical data and logistics data

Storage of the equivalent circuit diagram and encoder data in the motor

Integration of direct position measuring systems

Use of measuring systems with DRIVE-CLiQ interface

Use of SMC/SME modules for signal conversion on DRIVE-CLiQ

Third-party encoders/ third-party motors

Integration via SMC/SME modules

Parameter assignment via commissioning screens

Motion Connect 500 Proven, economic solution for small mechanical loads.

Motion Connect 800plus

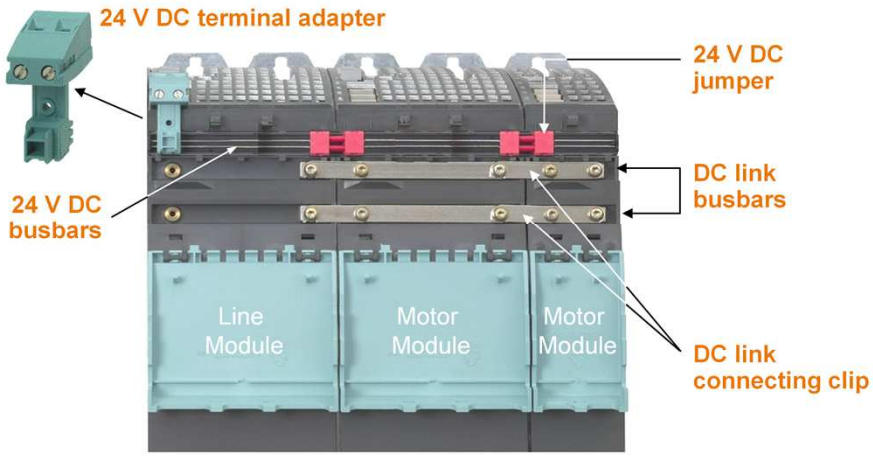
For high dynamics requirements also with long traverse distances.

Note The overall cable length of a Motion Connect connection must be calculated and is limited.
The maximum permissible cable lengths can be doubled if a DRIVE-CLIQ Hub Module (DMC20 or DME20) is used. After the hub, the same length conditions apply as in front of the hub.

It is possible to connect two DRIVE-CLiQ Hub Modules in series (cascade connection).

SINAMICS
Connection of "booksize" power components

SIEMENS



Course NC-84SL-SIP Page 2 - 20 SITRAIN © Siemens AG 2017 01/07/08

Protective barrier

When the protective barriers are folded down, the DC link and the 24 V electronic power supply can be seen behind them.

24 V

An external 24 V power supply is fed in via a terminal adapter. The red jumpers conduct the 24 V power supply to the next module.

Torque

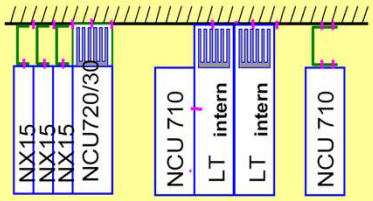
The DC link screws are tightened with a torque of 1.8 Nm using a T20 Torx screwdriver.

The 24 V terminal adapter is fastened with a Torx screw T10 with a torque of 0.5 Nm.

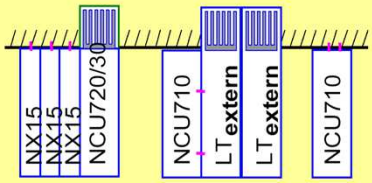
SINAMICS
Cooling types

SIEMENS

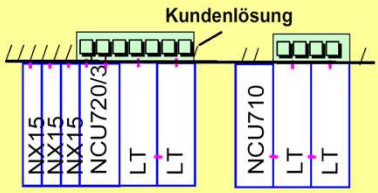
1. Internal air cooling



2. External air cooling



3. Cold plate



Cold plate

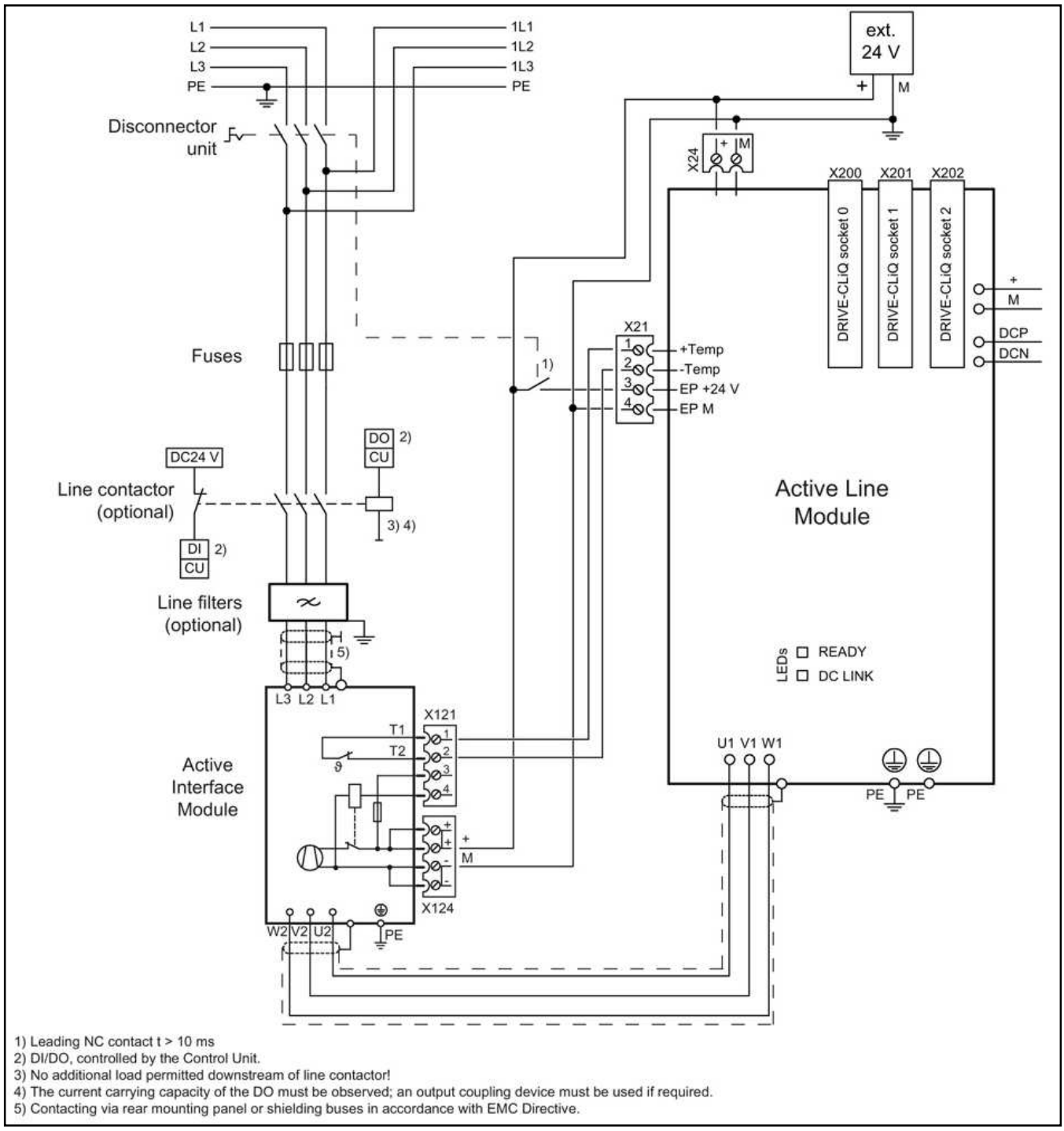
- Fanless module with flat rear
- Cooling via thermal jumper to external cooling plate
- Application support by CoC Chemnitz

Course NC-84SL-SIP Page 2 - 21 SITRAIN © Siemens AG 2017

Cold plate

A cold plate is a metal plate outside the control cabinet. It can either feature cooling fins for air cooling or even have fluid flowing through it. The heat is then dissipated no longer through air but through the cooling fluid. This is a very efficient method of cooling.

SINAMICS: Line connection



ALM / AIM The ALM (Active Line Module) infeed gets its line supply phases via the AIM (Active Interface Module).

Temp The temperature sensor contacts X21.1/2 on the ALM must be wired to X121.1/2 on the AIM.

EP The EP terminal must be disconnected with a lead of 10 ms with a contact at the main switch.

Line contactor control

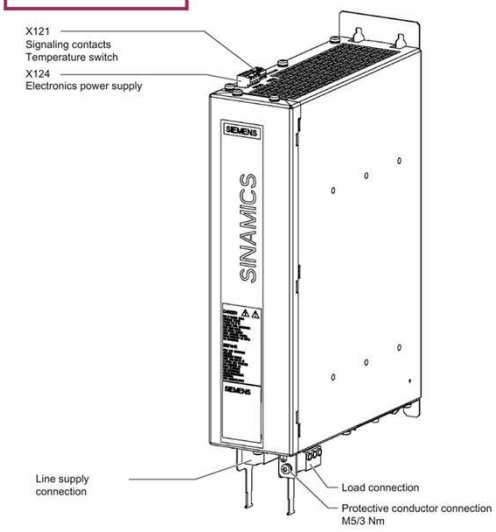
The use of line contactor control is optional.

Source Booksize Power Units Manual

SINAMICS
Active Interface Module (AIM): Terminals

SIEMENS

AIM 16 kW



Remove basic interference-suppression bridge:

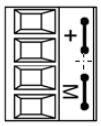
- **Connection to IT system**
- **Connection to ground-fault circuit interrupter**



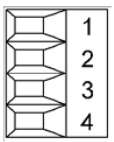
1. Release the screw of the connection bracket to the interference suppression capacitor using a Tx25 screwdriver.	2. Withdraw the connection bracket.	3. Connection bracket to the interference suppression capacitor

 SITRAIN © Siemens AG 2017

Terminal X124 Electronics power supply

	Terminal	Function	Technical data
	+	Electronics power supply	Voltage: 24 VDC (20.4 ... 28.8 V)
	+	Electronics power supply	Current consumption: max. 1.6 A
	M	Electronics ground	Max. current via jumper in connector: 20 A at 55° C
	M	Electronics ground	
Type: Screw terminal 2 (Page 750) Max. connectable cross-section: 2.5 mm²			

Terminal X121 Temperature sensor and fan control

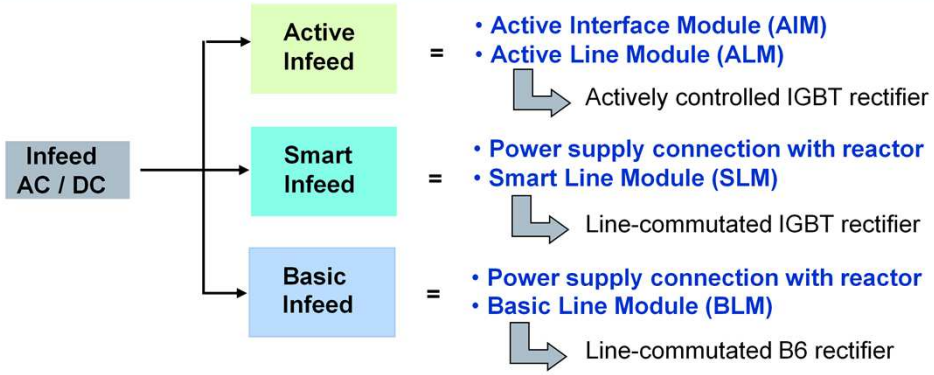
	Terminal	Designation	Technical data
	1	+Temp	Output temperature switch Must be connected to interface X21 of the Active Line Module.
	2	-Temp	Temperature switch output
	3	+24 V power supply for digital inputs	Current carrying capacity: 500 mA
	4	Disable Fan	**The fan can be disabled. The fan may be disabled only while the Active Line Module is disabled.**
Type: Screw terminal 1 (Page 750) Max. connectable cross-section: 1.5 mm²			

Power supply line L1, L2, L3 / load line U2, V2, W2

Klemmen	Bezeichnungen
Netzanschluss (Netz)	L1, L2, L3
Lastanschluss (Last)	U2, V2, W2

SINAMICS
Infeeds: Line Module

SIEMENS



Properties	Basic Infeed	Smart Infeed	Active Infeed
Operating mode	1Q operation/unregulated	2Q operation/unregulated	2Q operation/regulated
Power electronics	Thyristors/diodes	IGBTs	IGBTs
Power fluctuations	Affect VDC link	Affect VDC link	Are smoothed out
Energy recovery	No	Yes	Yes
Harmonics	Must be taken into account	Must be taken into account	Negligible
Reactive power compensation	No	No	Yes
DC link voltage	565 V	565 V	600 V (Active mode)
			565 V (Smart mode)

Course NC-84SL-SIP Page 2 - 24 SITRAIN © Siemens AG 2017

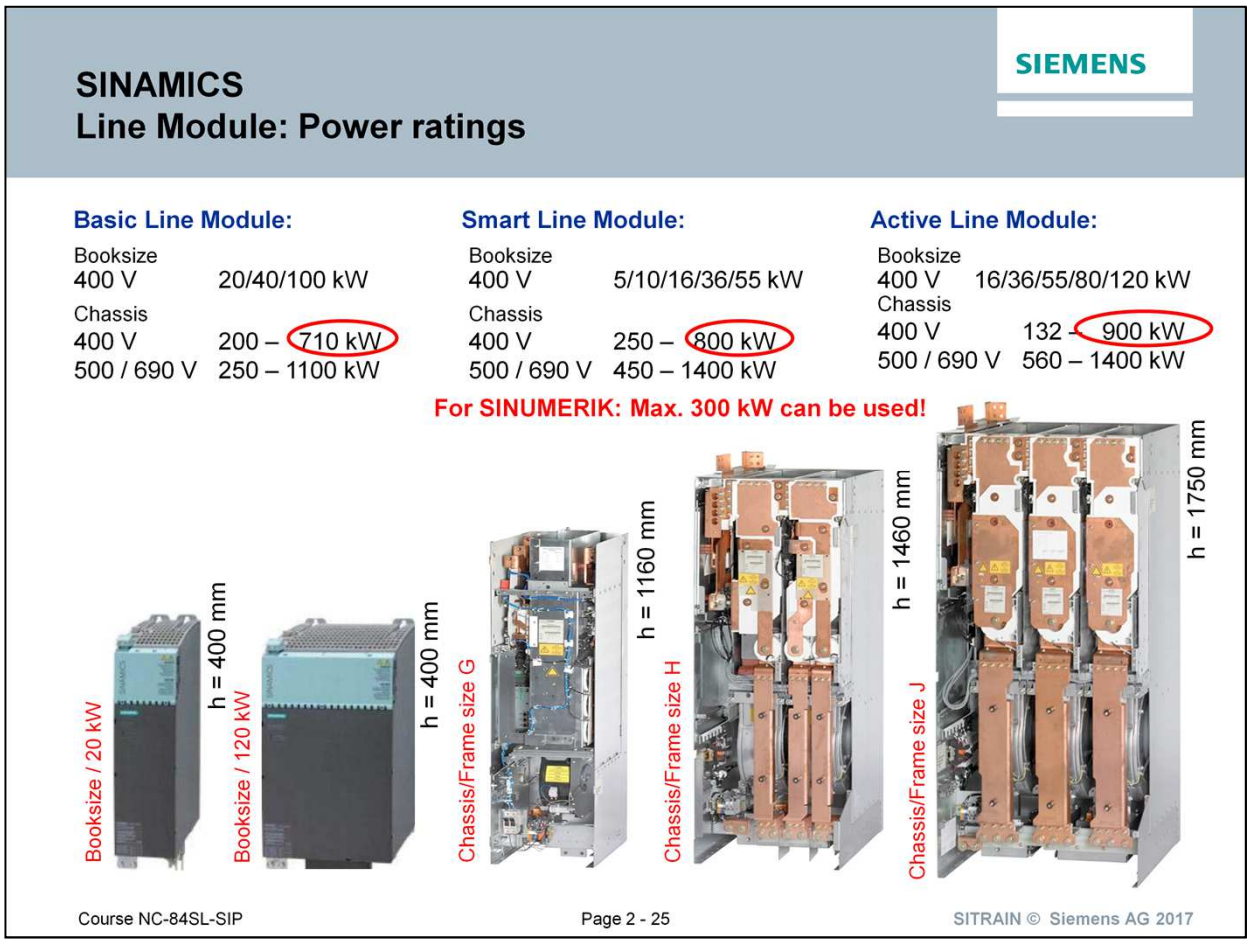
Line Modules Three different infeeds (Line Modules) are available for the SINAMICS drives.

ALM
Active Line Module
Rectifier with IGBTs
Regenerative feedback, stepping up and smoothing of DC-link voltage
Power factor correction

SLM
Smart Line Module
Rectifier with diodes and pre-charging with resistors
Regenerative feedback with IGBTs
No smoothing of the DC-link voltage

BLM
Basic Line Module
Rectifier with diodes and pre-charging with thyristors or resistors.
No regenerative feedback and no smoothing of DC-link voltage.

Note
The following combinations of Line Modules are not permissible (IH1):
Active Line Module (ALM) with Basic Line Module (BLM)
Active Line Module (ALM) with Smart Line Module (SLM)



Line Reactor for Line Modules

A line reactor is always required.

- On Active Line Modules, it is absolutely essential for operation of the step-up converter.
- With Smart Line Modules, it must be used to reduce line reactions and to provide protection against high current overshoots.

AIM Active Interface Module; featuring line reactor and basic interference suppression.

Systems All Line Modules are compatible for use in TN, IT, and TT systems.

SLM 5/10 kW These modules do not have a DRIVE-CLiQ connection.

BLM The 100 kW module does not feature integrated brake resistor control.

SINAMICS
ALM control modes

SIEMENS

Mode	DC link voltage	Regenerative feedback
Active Mode	regulated	possible
Smart Mode	rectified line voltage	possible

Supply voltage p0210 [V]	380...400	401...415	416...440	460	480
Control type p3400.0	"0" = Active Mode		"1" = Smart Mode		x 1.35
Vdc_set p3510 [V]	600	625	562-594[1)]	621[1)]	648[1)]

SITRAIN © Siemens AG 2017

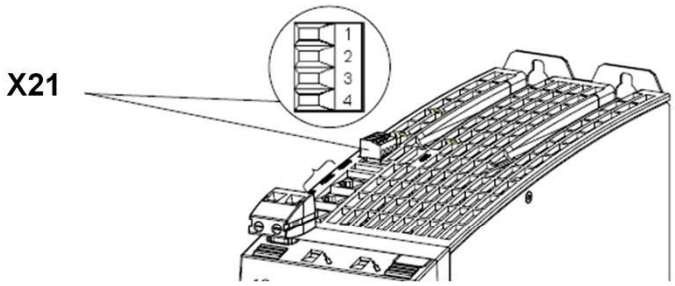
Control modes The **Active Line Module** can be operated in two different control modes depending on the parameterized line supply voltage (p0210):

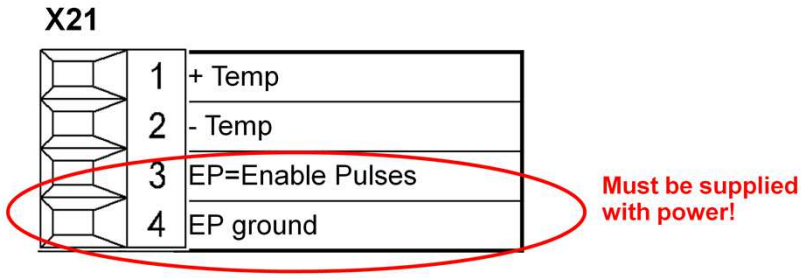
Active mode DC-link voltage is regulated to a settable value.
Step-up converter with regulated DC-link voltage and sinusoidal supply voltage.

Smart Mode DC-link voltage not regulated, but results from the rectified supply voltage.

SINAMICS
SLM > 10 kW or ALM: Terminals

SIEMENS





SITRAIN © Siemens AG 2017

Terminal X21

Terminal	Function	Technical data
1	+ Temp	Temperature sensor connection KTY84–1C130/PTC
2	- Temp	
3	EP +24 V (Enable Pulses)	Operating voltage: 24 V DC (20.4 V - 28.8 V) Current consumption: 10 mA
4	EP M1 (Enable Pulses)	

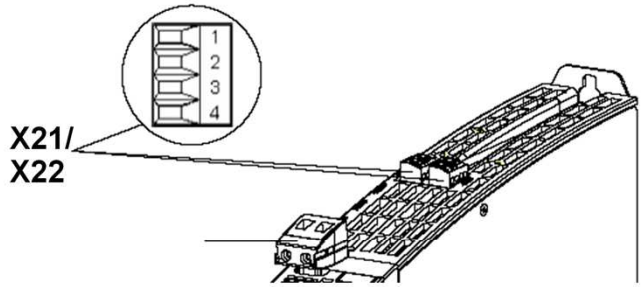
Temp contacts

For connection via an Active Interface Module (AIM), the Temp +/- contacts at X21 of the ALM should be wired to the corresponding contacts at X121 of the AIM, i.e.:

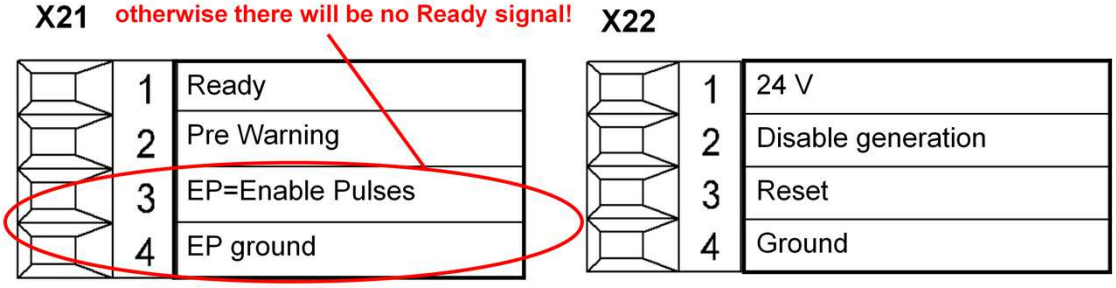
AIM		ALM
X121.1	———	X21.1
X121.2	———	X21.2

SINAMICS
SLM 5/10 kW: Terminals

SIEMENS



Must be supplied with power,
otherwise there will be no Ready signal!

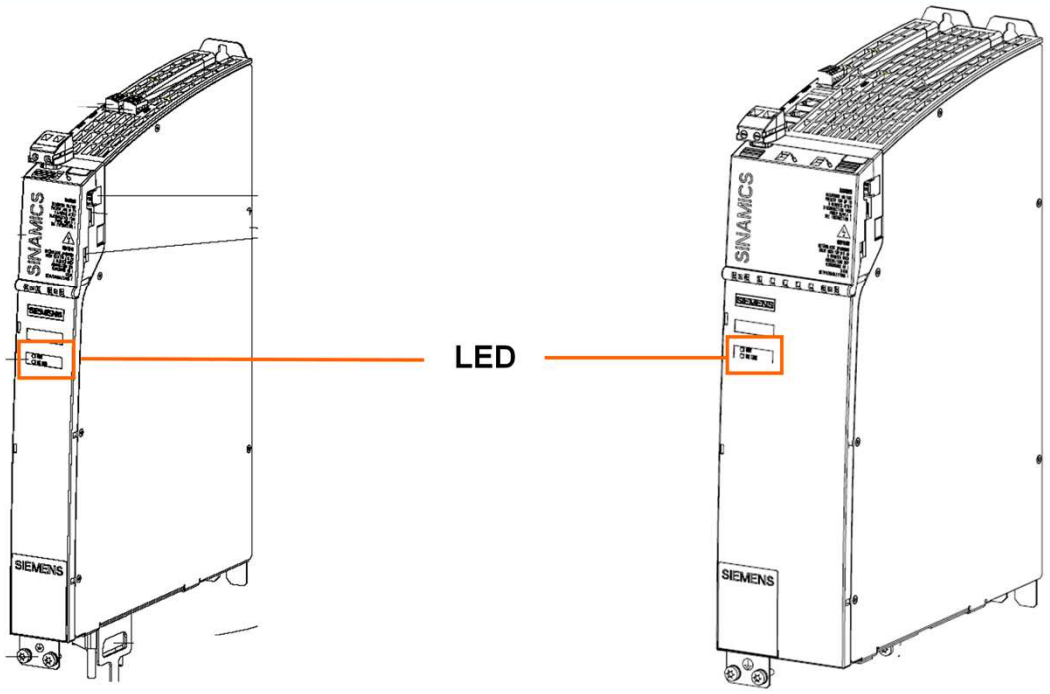


Course NC-84SL-SIP Page 2 - 28 SITRAIN © Siemens AG 2017

Terminal X21	Designation	Technical data
1	DO: Ready	Feedback signal Smart Line Module ready The signal switches to high level when the following conditions have been met: • Electronics power supply (X24) ok • DC link is pre-charged • Pulse enable (X21.3/.4) is present • No overtemperature • No overcurrent trip
2	DO: Pre Warning	Prewarning threshold overtemperature / I x t When 80 % of the maximum temperature of the Smart Line Module is exceeded, a low signal is output.
3	DI: Enable pulses	Voltage 24 V DC current consumption: 10 mA
4	DI: Enable pulses ground	Isolated input

Terminal X22	Designation	Technical data
1	24 V power supply	Electronics power supply for controlling digital inputs X22.2 and 3.
2	DI: Disable Regeneration	Deactivate regenerative feedback No power is fed back to the power system from the DC link. The regenerative energy of the motors may have to be reduced using a combination of the Braking Module and braking resistor.
3	DI: Reset	Reset faults (negative edge)
4	Ground	Electronics ground

SINAMICS
SLM / ALM: LED



Course NC-84SL-SIP Page 2 - 29 SITRAIN © Siemens AG 2017

	SLM 5/10 kW	Color	State	Description
LM without DRIVE-CLiQ	**READY**	Green	Continuous light	Ready
		Yellow	Continuous light	Pre-charging not yet complete; bypass relay dropped out; EP terminal without 24V
		Red	Continuous light	Overtemperature, overcurrent trip
	DC LINK		Off	The electronics power supply is missing or outside the permissible tolerance range
		Yellow	Continuous light	DC link voltage within permissible tolerance range
		Red	Continuous light	DC link voltage outside permissible tolerance range; line supply fault

	SLM / ALM	Color	State	Description
LM with DRIVE-CLiQ	**READY**	-	Off	The electronics power supply is missing or outside the permissible tolerance range
		Green	Continuous light	The component is ready for operation and cyclic DRIVE-CLiQ communication is taking place.
		Orange	Continuous light	DRIVE-CLiQ communication is being established.
		Red	Continuous light	At least one fault is present in this component.
		Green and red	Flashing light	Firmware download (2Hz) / completed (0.5Hz)
		Green/orange Red/orange	Flashing light 2 Hz	Component recognition via LED is activated.
	DC_LINK	-	Off	Electronics power supply outside the permissible tolerance range.
		Orange	Continuous light	DC link voltage within permissible tolerance range (only when ready for operation)
		Red	Continuous light	DC link voltage outside tolerance range

SINAMICS
DC-link components

SIEMENS

Braking Modules and braking resistors
Two braking resistors are available:
Booksize: 0.3 kW continuous braking power
(25 kW peak power)
1.5 kW continuous braking power
(100 kW peak power)



Capacitor Module
To increase the DC-link capacity to bridge short power failures.

Control Supply Module
Supplies 24 VDC power via the line supply or via the DC link.
This way it is possible, for example, to perform highly specific emergency retraction motions.



Course NC-84SL-SIP Page 2 - 30 SITRAIN © Siemens AG 2017

Braking Module/Braking resistor

Conversion of DC link energy into heat loss in the external braking resistor.

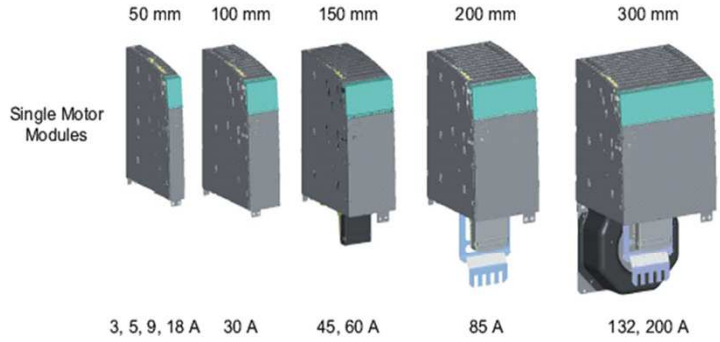
As the Smart and Active Line Modules are regenerative, Braking Modules and braking resistors are not required for normal operation.

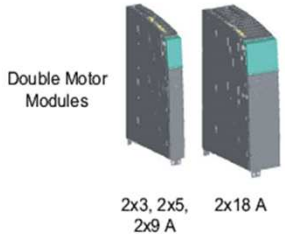
These components only have to be provided if a machine has to be stopped specifically in the event of a power failure (regenerative feedback no longer possible).

Note If a braking resistor is installed, it must be specified when setting up the infeed!

SINAMICS
Motor Modules, booksize

SIEMENS





Booksize format

- Compact design through high power density, particularly in the case of Double Motor Modules
- Integrated DC link and 24 VDC busbar
- Diagnostics LEDs
- Inherently short-circuit and ground fault proof
- Integrated safety functions “Safe motor braking control" and "Safe Torque Off"
- Integrated motor braking connection (up to and incl. 132 A) and braking by armature short-circuiting

Course NC-84SL-SIP Page 2 - 31 SITRAIN © Siemens AG 2017

Features of the booksize format

- Double Motor Modules integrate two autonomous Motor Modules in a single housing (saving up to 50 % in space)
- Connection via the DC link and 24 VDC busbars integrated in the device Integrated safety functions such as Safe Torque Off" (STO) and "Safe Brake Control" (SBC)
- Overload capacity of the Motor Modules similar to SIMODRIVE 611.
- The specified depth of 270mm only applies to devices with air cooling. Models with forced convection cooling and cold plate cooling have a reduced overall depth
- All components in the SINAMICS S system have cULus approval. The cULus mark is a test mark of Underwriters Laboratories (UL) and indicates for end products testing by UL according to UL and CSA (Canadian Standard Association) standard.

Properties of Motor Modules

- Supply voltage 510-720 V DC
- Degree of protection IP20
- Heat dissipation: Internal/external air cooling or cold plate cooling
- Ambient temperature: 0-40 °C without derating, up to 55 °C with derating
- Height: 380 mm
- Depth: 270mm

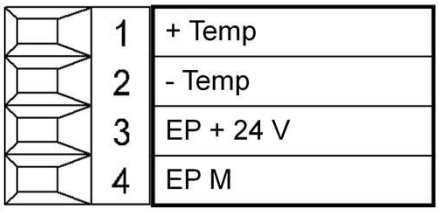
Single Motor Modules Rated output current: from 3 A – 200 A
Maximum output current: from 6 A to 282 A
Width: from 50 mm to 300 mm

Double Motor Modules Rated output current: from 3 A – 18 A
Maximum output current: from 6 A to 36 A
Width: from 50 mm to 100 mm

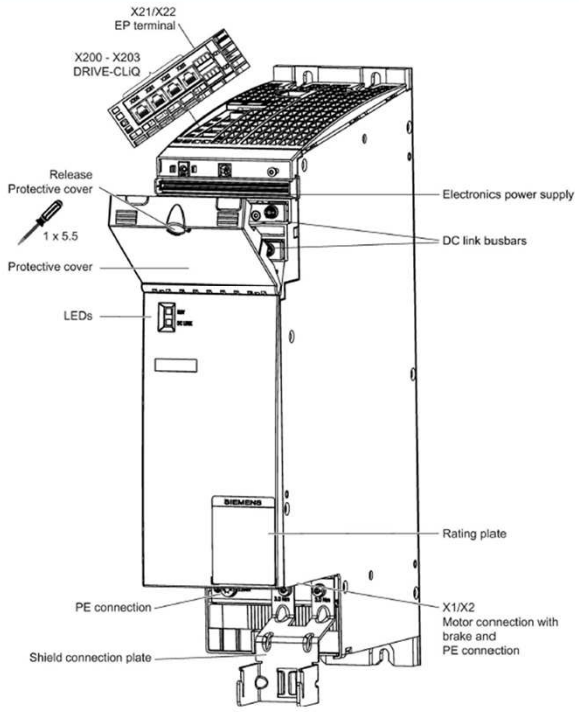
SINAMICS
Motor Module: Terminals

SIEMENS

X21/X22





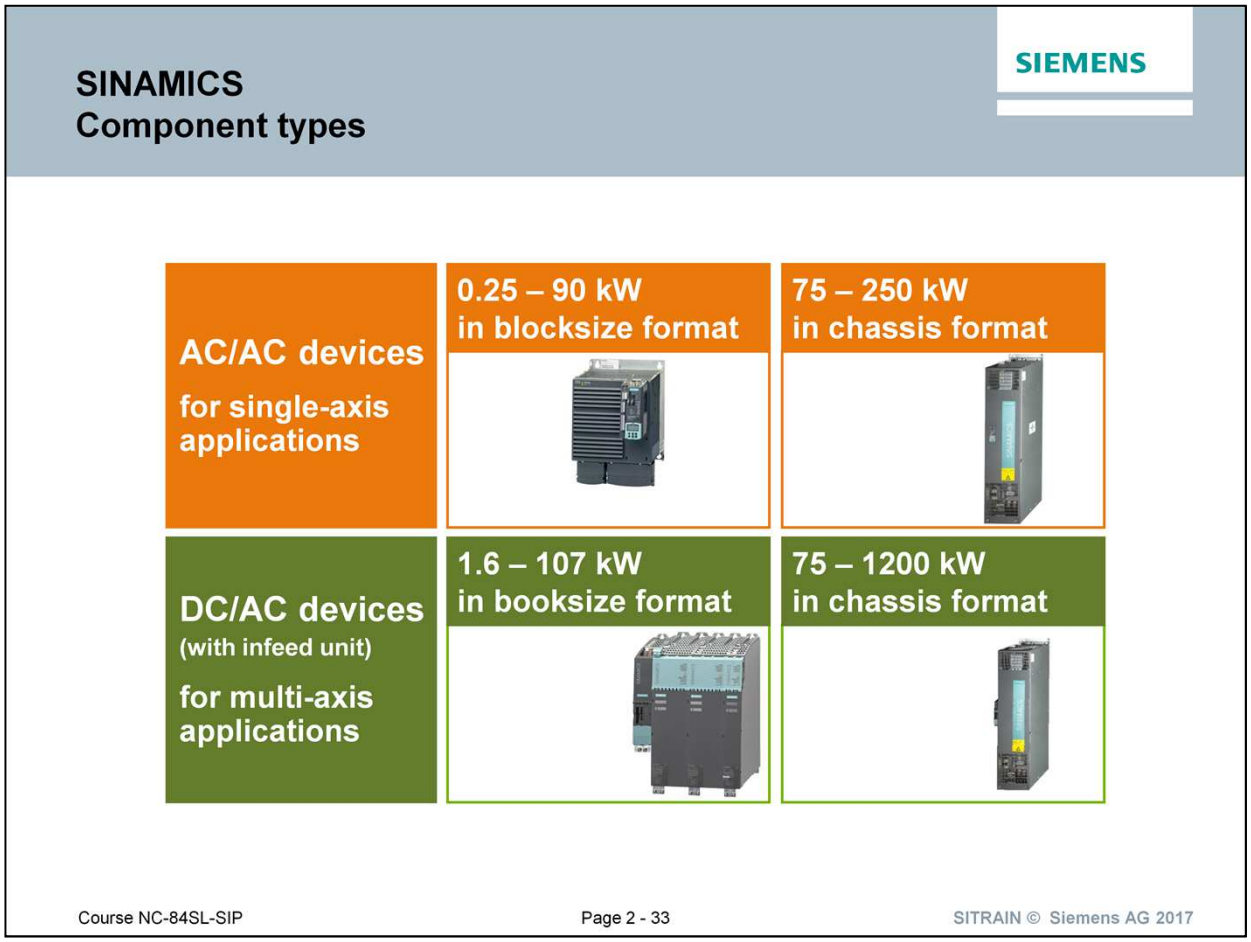


Course NC-84SL-SIP Page 2 - 32 SITRAIN © Siemens AG 2017

Terminals X21 / X22	Function	Technical data
1	+ Temp	Temperature sensor connection KTY84–1C130/PTC
2	- Temp	
3	EP +24 V (Enable Pulses)	Operating voltage: 24 V DC (20.4 V - 28.8 V) Current consumption: 10 mA
4	EP M1 (Enable Pulses)	

Note terminal EP + 24 V power must only be supplied to this terminal if you are using drive-integrated safety (safe stop deselected on Motor Module).

LEDs	Color	State	Description
READY	-	Off	The electronics power supply is missing or outside the permissible tolerance range.
	Green	Continuous light	The component is ready for operation and cyclic DRIVE-CLiQ communication is taking place.
	Orange	Continuous light	DRIVE-CLiQ communication is being established.
	Red	Continuous light	At least one fault is present in this component.
	Green/red	Flashing light	Firmware download active (2 Hz) / ended (0.5 Hz)
	Green/orange Red/orange	Flashing light 2 Hz	Component recognition via LED is activated Note: Both options depend on the LED status when module recognition is activated via p0124 = 1.
DC LINK	-	Off	Electronics power supply outside the permissible tolerance range.
	Orange	Continuous light	DC link voltage within permissible tolerance range (only when ready for operation)
	Red	Continuous light	DC link voltage outside permissible tolerance range (only when ready for operation)



"AC/AC devices" convert fixed-frequency alternating voltage (power supply grid) into variable-frequency alternating voltage (motor) in a single component; "DC/AC devices" convert fixed-frequency alternating voltage (power supply grid) into variable-frequency alternating voltage (motor) in multiple components.

AC/AC devices

AC/AC devices are drive modules that are connected directly to the fixed-frequency alternating voltage of the power supply grid and handle the entire process for conversion into variable-frequency alternating voltage internally.

Here too, conversion usually involves one DC link (in this case an internal DC link).

These devices do not need an infeed module.

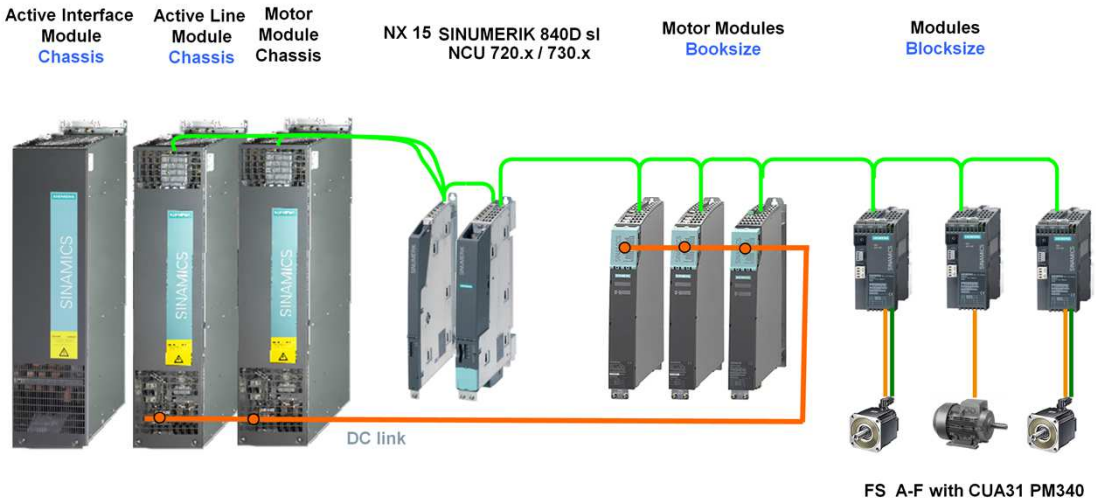
DC/AC devices

These are drive modules that are supplied with direct voltage by the infeed module via DC link busbars.

Therefore, in these modules, only the second step of the conversion process is performed (direct voltage into variable-frequency alternating voltage).

The actual infeed could then be referred to as the AC/DC device.

SINAMICS
Mixed operation chassis / booksize / blocksize



Course NC-84SL-SIP Page 2 - 34 SITRAIN © Siemens AG 2017

Note A maximum of 2 chassis axes are permitted as of SW 2.6.

Mixed operation In booksize/blocksize mixed operation, up to 6 axes are possible.

SINAMICS
Active Line Module Chassis

SIEMENS



	Active Line Modules			
Supply voltage	3AC 380-480V -15/+10%, 50/60Hz -6/+6%			
Infeed power (kW)	132	160	235	300
Width (mm)	326	326	326	326
Height (mm)	1400	1400	1530	1530
Depth (mm)	351	351	545	545
Degree of protection	IP20/IP00			
Cooling	Internal air cooling, Liquid cooling			
Ambient temperature	0-40°C, up to 55°C with Derating			

Course NC-84SL-SIP Page 2 - 35 SITRAIN © Siemens AG 2017

LED ALM	Color	State	Description
READY	-	Off	Electronics power supply outside the permissible tolerance range.
	Green	Continuous light	The component is ready for operation and cyclic DRIVE-CLiQ communication is taking place.
	Orange	Continuous light	DRIVE-CLiQ communication is being established.
	Red	Continuous light	At least one fault is present in this component.
	Green and red	Flashing light	Firmware download active (2 Hz) / ended (0.5 Hz)
	Green/orange Red/orange	Flashing light 2 Hz	Component recognition via LED is activated (p0124). Note: Both options depend on the LED status when module recognition is activated (p0124 = 1).
DC_LINK	-	Off	Electronics power supply outside the permissible tolerance range.
	Orange	Continuous light	DC link voltage within permissible tolerance range (only when ready for operation)
	Red	Continuous light	DC link voltage outside the permissible tolerance range (only when Active Line Module is ready for operation).

SINAMICS
Chassis Motor Module

SIEMENS



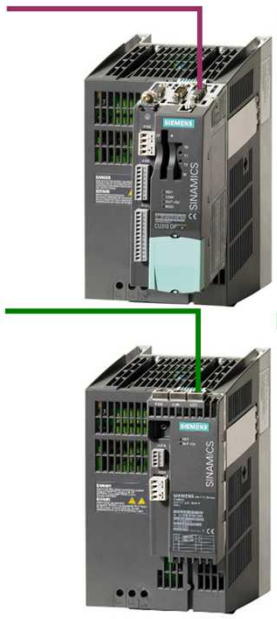
	Single Motor Modules
Supply voltage	DC 510-720V
Rated output current (A)	210 | 260
Width (mm)	326
Height (mm)	1400
Depth (mm)	351
Degree of protection	IP20
Cooling	Air cooling, Liquid cooling
Abient temperature	0-40°C, up to 55°C with Derating

Course NC-84SL-SIP Page 2 - 36 SITRAIN © Siemens AG 2017

	Color	State	Description
READY	-	Off	Electronics power supply outside the permissible tolerance range.
	Green	Continuous light	The component is ready for operation and cyclic DRIVE-CLiQ communication is taking place.
	Orange	Continuous light	DRIVE-CLiQ communication is being established.
	Red	Continuous light	At least one fault is present in this component.
	Green/red	Flashing light	Firmware download active (2 Hz) / ended (0.5 Hz)
	Green/orange or Red/orange	Flashing light 2 Hz	Identifying whether the component is activated using the LED (p0124). Note: Both options depend on the LED status when module recognition is activated via p0124 = 1.
DC LINK	-	Off	Electronics power supply outside the permissible tolerance range.
	Orange	Continuous light	DC link voltage within permissible tolerance range (only when ready for operation)
	Red	Continuous light	DC link voltage outside permissible tolerance range (only when ready for operation)

SINAMICS
Blocksize AC/AC Module

SIEMENS



AC drives for single-axis applications and additional axes

- **Applications**
  - Single-axis device, incl. closed-loop control (CU310: PROFIBUS)
  - Additional axis (CUA connection: DRIVE-CLiQ)
- **Power classes**
  - 0.250 – 1.1 kW with power supply connection 200 – 240 V 1 AC
  - 0.37 – 90 kW with power supply connection 380 – 480 V 3 AC

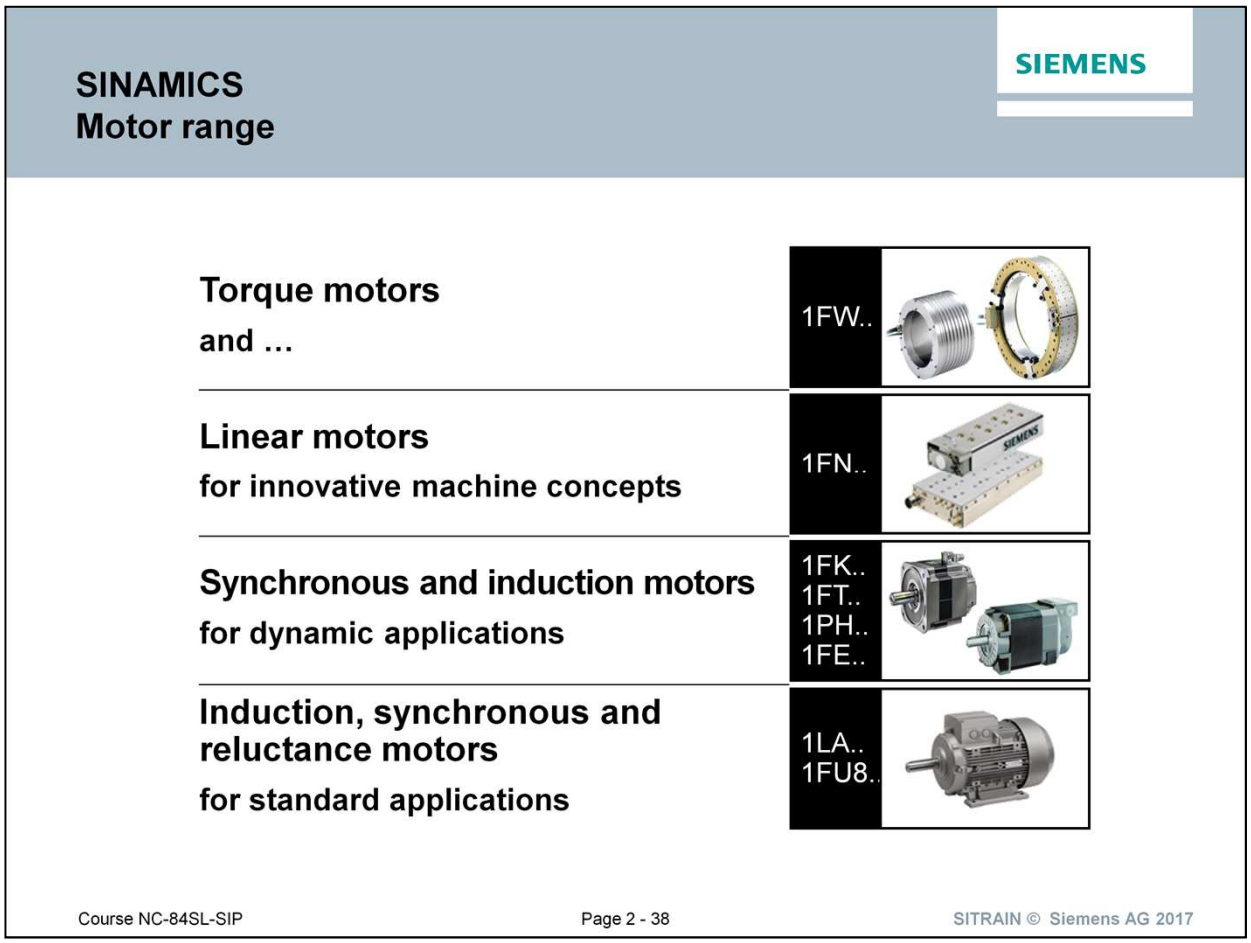
Course NC-84SL-SIP Page 2 - 37 SITRAIN © Siemens AG 2017

CU 310 You can use the plug-on CU310 to create a drive-autonomous solution or a PLC axis.

CUA The axis is integrated into the NC drive line-up via a plug-on adapter (CUA). The following variants are available:

CUA 31 Without encoder interface

CUA 32 With encoder interface (HTL/TTL/SSI)



Motors with DRIVE-CLiQ

The motor lines 1FK7; 1FT6 and 1PH7 are also available with DRIVE-CLiQ interface

- Dimension sheets for 1FK7 and 1FT6 are available on the INTRANET

1PH8

This new motor series, which was developed specifically for the SINAMICS S120, is available as:

- Synchronous motor
- Induction motor

Encoder systems for motors with DRIVE-CLiQ interface:

• 1FT6 / 1FK7 order code 14th position:

Incremental encoder sin/cos 1 Vpp 2048 S/R	**D**
Absolute encoder EnDat 2048 S/R	**F**
Absolute encoder EnDat 512 S/R	**L**
Multi-pole resolver	**U**
Resolver, 2-pole	**P**

•1PH7 order code 9th position:

Absolute encoder EnDat 2048 S/R	**F**
Incremental encoder sin/cos 1 Vpp 2048 S/R	
with C and D tracks	**D**
without C and D tracks	**Q**

SINAMICS
Sensor Module Cabinet

SIEMENS

Encoders and sensor modules:

- Incremental encoder
  - **sin/cos 1 Vpp**
  - **TTL/HTL**
- Absolute encoder
  - **EnDat**
- Resolver
  - **Multi-pole**
  - **2-pole**



Measuring systems	SMC 10	SMC 20	SMC 30
Resolver	Yes	-	-
Incremental encoder sin/cos (1Vpp)	-	Yes	-
Absolute encoder EnDat	-	Yes	-
Incremental encoder TTL/HTL	-	-	Yes
Temperature evaluation	Yes	Yes	Yes

SITRAIN © Siemens AG 2017

Notes

SINAMICS
SME 120, 125

SIEMENS



Sensor Module External with IP67 degree of protection for encoder signal conditioning on 1FN1, 1FN3, 1FW6, and 1FE1 built-in motors

Evaluation with safe electrical separation for 1 x KTY and 2 x PTC motor temperature encoders

 SITRAIN © Siemens AG 2017

Use

The component is always used when the temperature signals of the motors are not reliably separated electrically or where this separation is also not possible for certain reasons.

A Hall sensor box can be connected to determine the commutation position of a linear motor.

The SME120 / 125 should be used for linear and torque motors in particular. Neither the motor data nor the encoder data is saved.

SME120

Sensor Module SME 120 with signal conversion on DRIVE-CLiQ for incremental sin/cos measuring systems

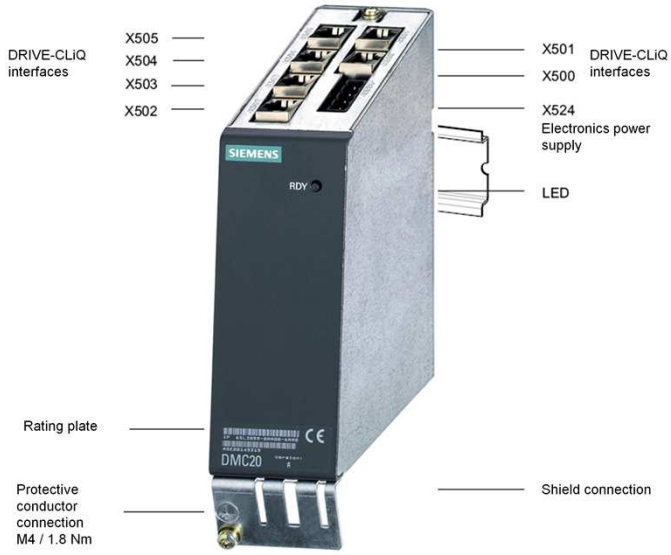
Evaluation with Hall sensor box connected to linear motors

SME 125

Sensor Module SME 125 with signal conversion on DRIVE-CLiQ for EnDat measuring systems

SINAMICS
DRIVE CLiQ Hub

SIEMENS



DMC20



DME20

Course NC-84SL-SIP Page 2 - 41 SITRAIN © Siemens AG 2017

DMC 20 DRIVE-CLiQ Hub Module Cabinet 20 / External 20

Use
- Extended DRIVE-CLiQ ports, e.g. direct measuring systems

Benefits
- Reduces the number of encoder cables in the cable trailing device by compressing the encoder cables from up to five measuring systems on a single DRIVE-CLiQ branch (5 to 1)

Note Each individual DMC20 or DME20 corresponds to two DRIVE-CLiQ nodes. This must be taken into account in the DRIVE-CLiQ node quantity structure.

SINAMICS
Terminal Module

SIEMENS



TM 15:
24 isolated DI/DO
Can be parameterized channel-by-channel as digital input, digital output, or measuring input or cam output.



TM 31:
8 digital inputs
4 bidirectional digital inputs/outputs
2 relay outputs with changeover contact
2 analog inputs
2 analog outputs
1 temperature sensor input
2 DRIVE-CLiQ sockets



TM 41:
4 floating digital inputs
4 digital inputs/outputs
1 analog input ± 10V (13 bits + sign)
1 TTL encoder output for encoder simulation
2 DRIVE-CLiQ interfaces

Course NC-84SL-SIP Page 2 - 42 SITRAIN © Siemens AG 2017

TM 15 With the Terminal Modules, digital inputs and outputs are implemented close to the drive. In addition, Terminal Modules provide IOs with short signal delay times.

TM31 The TM31 Terminal Module can be used to increase the number of available digital and analog inputs and outputs within a drive system.

TM41 An incremental encoder can be emulated using the encoder interface of the TM41. The TM41 can also be used to couple analog controls to SINAMICS.

SINAMICS
Terminal Module TM 120

SIEMENS



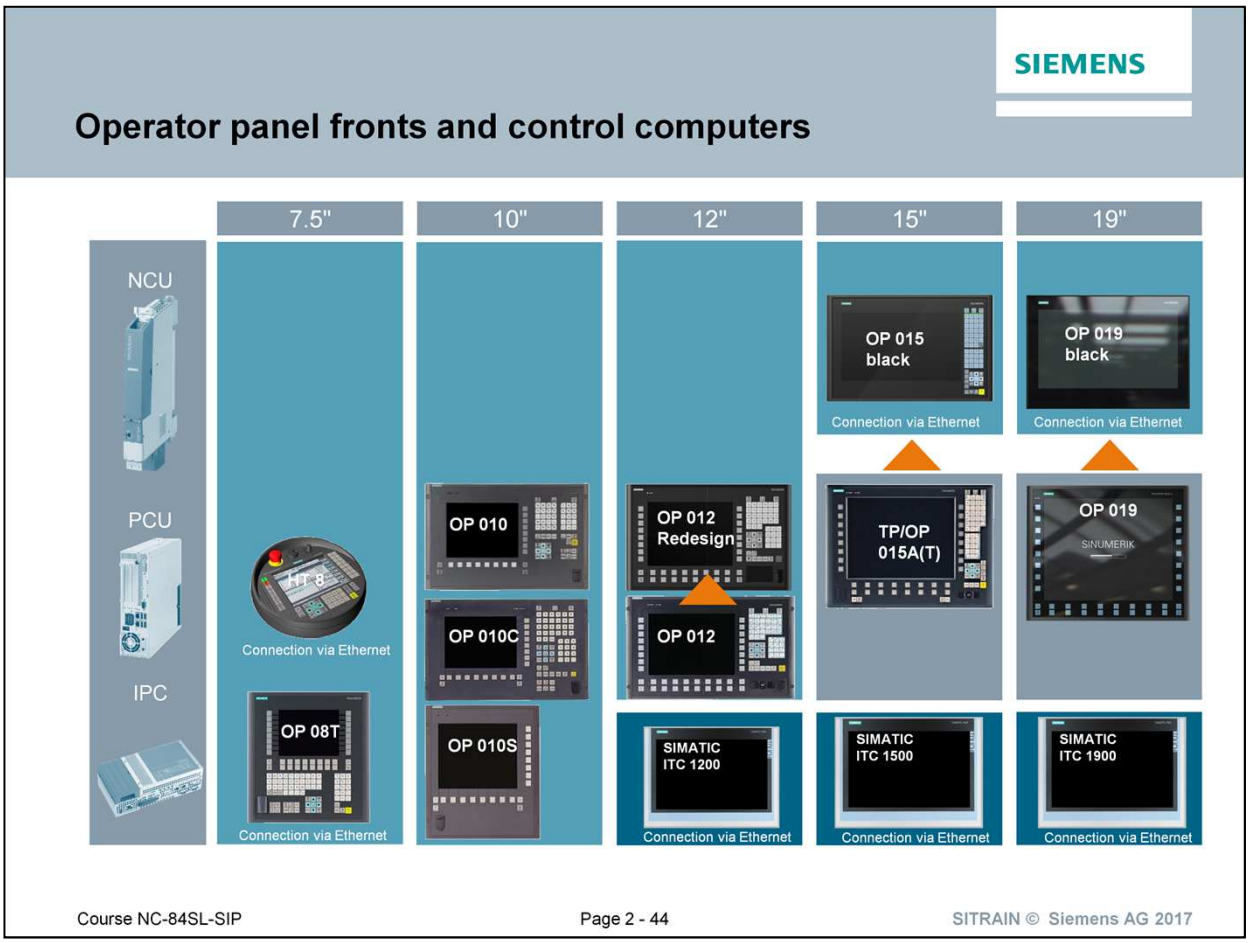
- Connection of up to 4 temperature sensors
- Sensors: KTY84-130, PTC 130, and PTC 150
- With protective separation
- Use BICO technology for assignment to axes

Course NC-84SL-SIP Page 2 - 43 SITRAIN © Siemens AG 2017

Description

The Terminal Module TM120 is a DRIVE-CLiQ component for safe electrically isolated temperature evaluation.

It can be used for 1FN, 1FW6, and third-party motors in which the temperature sensors cannot be installed with safe electrical separation.
The TM120 is mounted in the cabinet and is suitable for clipping onto a DIN rail (EN 60715).
If a TM120 is used, the function is distributed between temperature evaluation and encoder evaluation. The TM120 can acquire the motor temperature via 4 channels with different temperature sensors. Encoder evaluation functions are performed via Sensor Modules (e.g. SMCxx, SMExx). This means that, when connected to a Sensor Module SMCxx, the TM120 represents a control cabinet alternative to the SME120/SME125.



OP / TP

OP stands for "Operator Panel", TP stands for "Touch(-sensitive) Panel".

Screen size

The digit in the name of the OP indicates the screen size.
The "OP 012", for example, features a 12-inch screen.

Suffix A

The suffix "A" stands for Automotive. The Siemens HMI Pro is in widespread use in this industry. It requires a dual menu bar underneath the screen as well as buttons on the left and right of the screen.

Suffix T

The suffix "T" stands for TCU. The panel has a built-in TCU.

OP 10 and S/C

The difference between these two OPs is to be found in their control elements; one has a mechanical keypad (OP10S/C) and the other has a membrane keypad (OP10).

Further information

For further information about operator components such as OPs, PCUs, TCUs, machine control panels, etc., please refer to the 'Operator Components and Networking' manual.

Machine control panels, hand-held controllers



- SINUMERIK integrated key-operated switch
- Mounting slots for control devices
- Optional connector for direct keys/handwheel
- Acknowledgment button
- EMERGENCY OFF
- Ergonomic

SITRAIN © Siemens AG 2017

Up to 2 machine control panels and one handheld unit can be in use at the same time. There are various connection options (Ethernet/PROFINET, PROFIBUS) for the machine control panel (MCP) and handheld unit (HHU). It is possible to connect 2 MCPs to different bus systems (mixed operation is only possible on Ethernet and PROFIBUS).

Activation

Each component is activated either via the number of machine control panels ("MCPNum" parameter) or, in the case of the handheld unit, via the "HHU" parameter. The MCP and HHU connection settings are entered in FB1 parameters "MCPMPI", "MCPBusType" or "BHG", "BHGMPI".

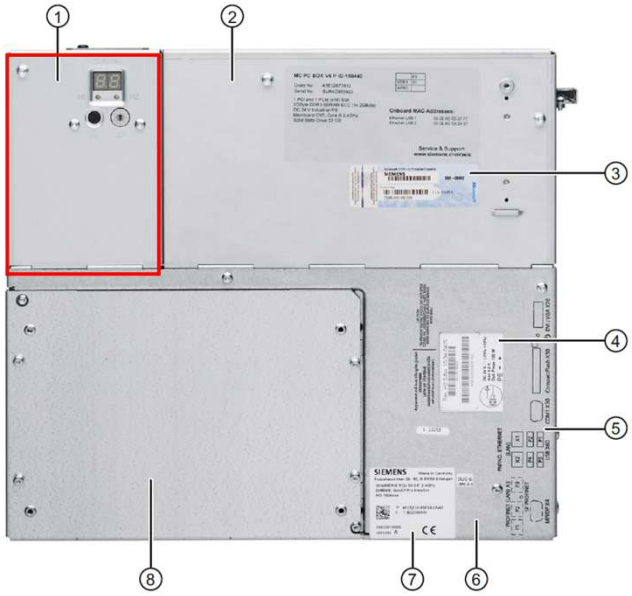
Handheld unit (HT2)

In the handheld unit the addressing is achieved via a parameter of the GD parameter set. This was necessary to ensure compatibility of parameter names.

PCU 50.5
Overview

SIEMENS

① Housing cover, service module
② Housing cover, motherboard
③ Windows license label
④ Rating plate for power supply
⑤ Overprint for the interfaces on the right-hand side of the housing
⑥ Housing cover, power supply
⑦ PCU rating plate
⑧ SSD housing cover



Course NC-84SL-SIP Page 2 - 46 SITRAIN © Siemens AG 2017

Description

The powerful SINUMERIK PCU 50.5 comes complete not only with an integrated 150 W power supply unit but also all of the interfaces necessary for communication via Ethernet and PROFIBUS (depending on the particular device) already on board.

Four USB ports (USB 2.0) provide points where a keyboard, mouse and other peripheral devices can be connected.

A covered slot is available for a CompactFlash card (CF card).

The PCU 50.5 can only be used in the SINUMERIK 840D sl system.

Versions

Operating system	WinXP ProEmb Sys (32 bits)		Win 7 ProEmb Sys (64 bits)	
Hardware	PCU 50.5-C	PCU 50.5-P	PCU 50.5-C	PCU 50.5-P
Processor	Celeron, 1.8 GHz	Core i5 2.4 GHz	Celeron, 1.8 GHz	Core i5 2.4 GHz
Random access memory	1024 MB	2048 MB	4096 MB	8192 MB
Mass storage	40 GB	40 GB	80 GB	80 GB
PROFIBUS DP	Yes	No	No	No
Slots	2 x PCI	PCI + PCIe x16	PCI + PCIe x16	PCI + PCIe x16

PCU 50.5
Service module

SIEMENS

① 2 x 7-segment display
② 2 x LEDs: H1 and H2
③ Service switch
④ Power button



Course NC-84SL-SIP Page 2 - 47 SITRAIN © Siemens AG 2017

Service module

The service module is located, depending on the device mounting, at two positions:

- Side panel, next to the equipment fan for vertical mounting in the control cabinet
- Cover above the equipment fan for standard installation in the operator panel

Displays

Standard display following successful run-up:

- 7-segment display: 00
  A display other than this may mean:
  - BIOS error message (refer to the manual for the operator components)
  - Applications code
- LED H1 | H2: both dark

Service switch

Main applications:

- Force USB boot (position E)
- Boot into the Windows system

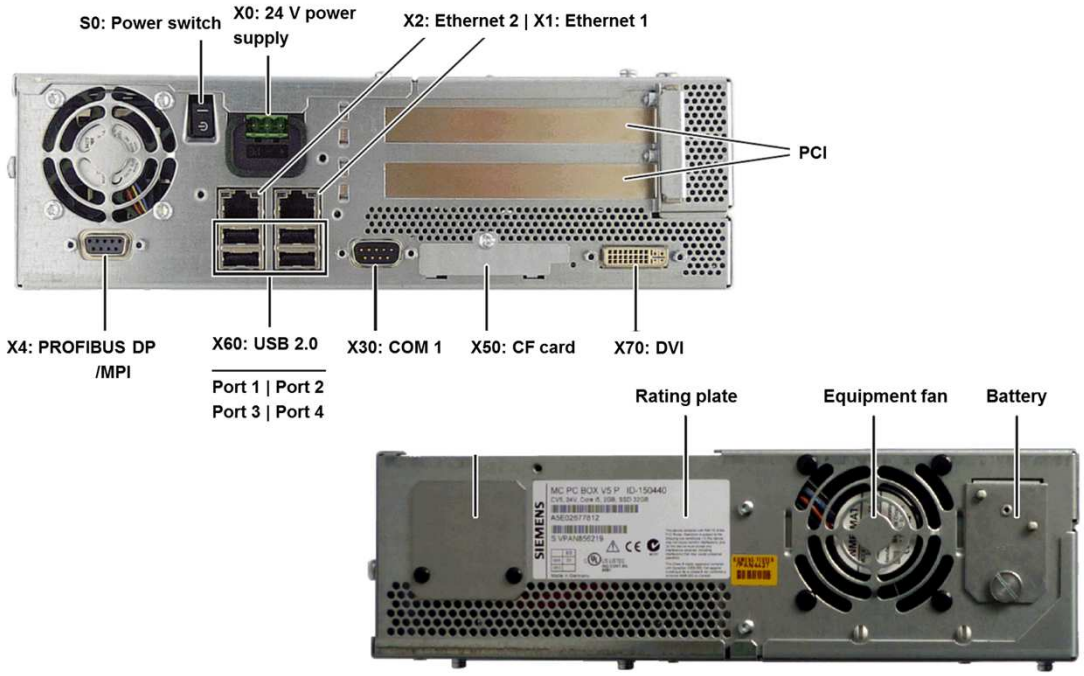
Switch position	Mode	Function
0		Normal operation (default setting)
3	-	Win7 PCU – calling the boot menu
6	-	Reserved for end users
B	Service	BIOS recovery after a failed BIOS update
E	Service	Restart is forced by USB drive

Power button

Alternative to the ON/OFF switch

PCU 50.5
Interfaces

SIEMENS

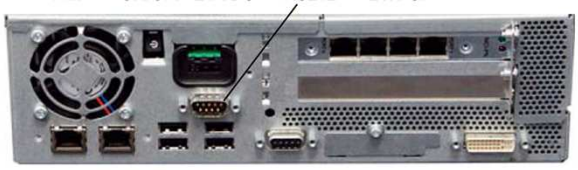


SITRAIN © Siemens AG 2017

Notes

PCU 50.3
Interfaces

SIEMENS



Connections

- 2x Ethernet 10/100 Mbps (RJ45)
- 4x USB 2.0 1)
- DVI (VGA possible via DVI-VGA adapter)
- 12 Mbps PROFIBUS DP/MPI
- SINUMERIK operator panel front
- RS 232C

Expansion slots

- 1 x PCI (max. length 265 mm)
- 1 x PCI (max. length 175 mm)
- 1 x CF card

1) Keyboard, mouse, external diskette drive via USB

Battery box **Fan**

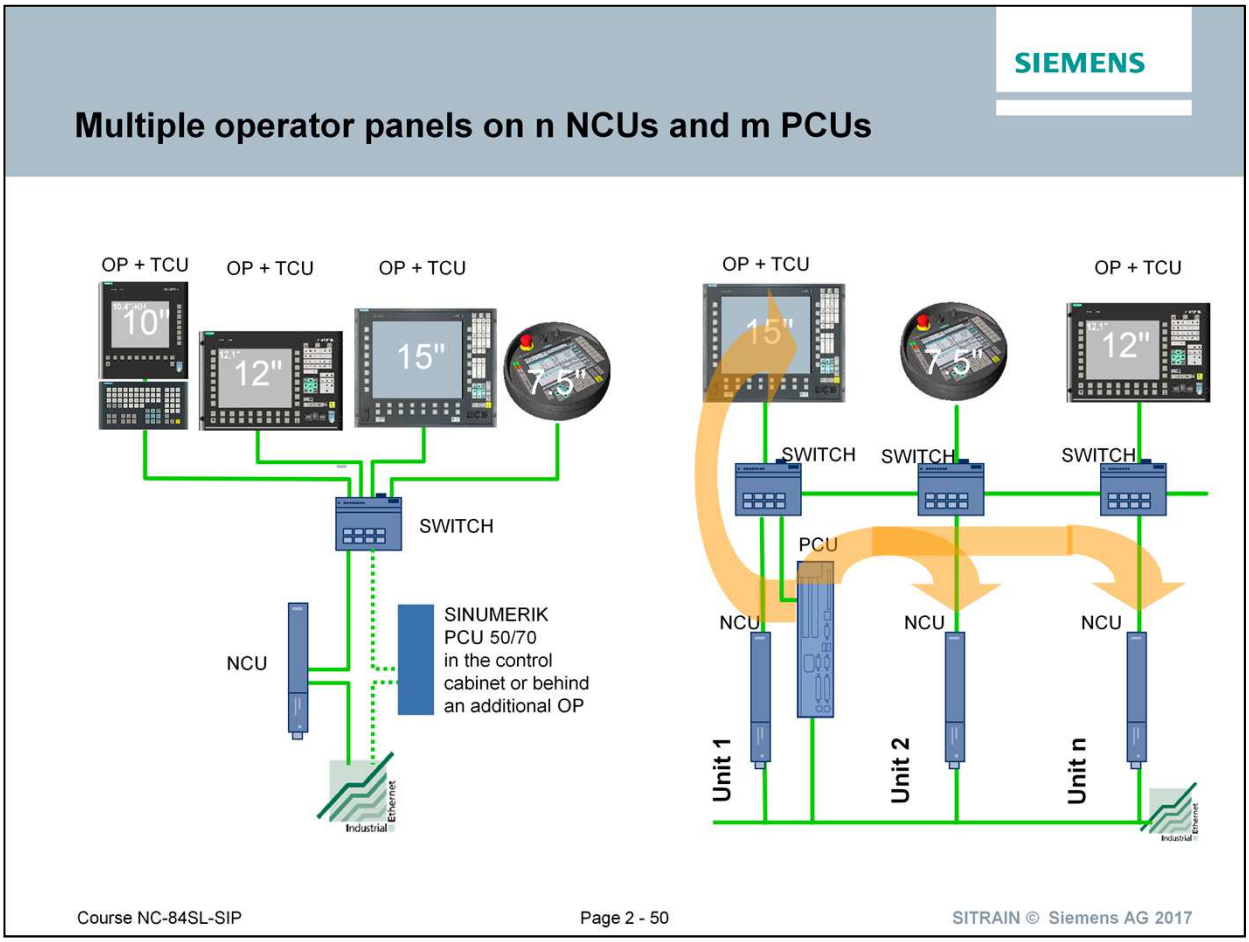




Easily accessible **battery**

SITRAIN © Siemens AG 2017

Notes



Multiple operator panels on one NCU / PCU

The TCU can be switched over via Ethernet address or channel menu (Netnames.ini).
Shadowing (=replication) of the operator interface on multiple OPs, each with a TCU.
Up to 4 different screen resolutions are possible.
The cable length between the switch and the device can be up to 100 m.
The PCU simultaneously receives data from n controls.

Several operator panels on several NCU / PCU

One NCU (to which the PCU is connected) and a second NCU (with integrated HMI to which the HT 8 is connected, for example) can be operated and monitored at the same time.
The TCUs are connected via a switch to NCUs / PCUs.

NCUs/PCUs connected directly or bundled via a separate switch to the company network.

Machine control panels
MCP 310/MCP 310 IE

SIEMENS



- **Connection: PROFIBUS DP, Industrial Ethernet**
- **SINUMERIK integrated key-operated switch**
- **6 slots for command devices (D16 mm)**
- **Direct keys/ handwheel connection**

Course NC-84SL-SIP Page 2 - 51 SITRAIN © Siemens AG 2017

Note This machine control panel is addressed in the PLC program via the FC 24.

Machine control panels
MCP 483 /MCP 483 IE

SIEMENS

- **Connection: PROFIBUS DP, Industrial Ethernet**
- **2 slots for command devices (D16 mm)**
- **EMERGENCY OFF button (D22 mm), contacts can be expanded**
- **Direct keys/handwheel connection**



Course NC-84SL-SIP Page 2 - 52 SITRAIN © Siemens AG 2017

Note This machine control panel is addressed in the PLC program via the FC 19/25.

MCP 483: DIP switch S3 On DIP switch S3 of the MCP 483, the type of communication (DP or MPI), baud rate and the MPI or PROFIBUS address are set.

Factory setting MCP 483

The S3 switches are set as follows on delivery:

Meaning	1	2	3	4	5	6	7	8	9	10
MCP address …	On	Off	On	Off	On	On	Off	Off	Off	Off

The set address must correspond to parameter MCP1BusAdr or MCP2BusAdr of the FB1 in the basic program block OB100
(Standard address: 6)

For that reason, the following readjustment must be made:

Meaning	1	2	3	4	5	6	7	8	9	10
PROFIBUS, MCP address 6	Off	On	On	Off	Off	Off	Off	Off	On	On

MCP 483 PN / MCP 483C PN:

DIP switch S2 On DIP switch S2 of the MCP 483 PN or MCP 483C PN, the address is set. This is an 'alias address' that must correspond to the parameter MCP1BusAdr or MCP2BusAdr of the FB1 in the basic program block OB100 (default address: 192).

As-delivered condition MCP 483 PN / MCP 483C PN:

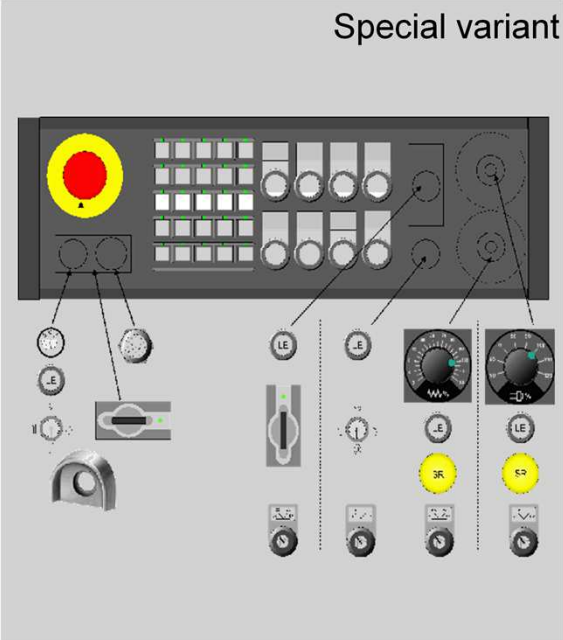
Meaning	1	2	3	4	5	6	7	8	9	10
IE, MCP address 192	Off	Off	Off	Off	Off	Off	On	On	Off	Off

Readjustment on PROFINET

That is possible with S2.9=on, S2.10=on. The device name is then mcp-pn192

Machine control panels
MPP 483IE S push button panel

SIEMENS



Flexible

Fixed elements

Connection: PROFIBUS/IE

EMERGENCY OFF button

25 membrane-type keys/LEDs with slide-in strips

Variable elements

8 custom keys/LEDs with slide-in strips (long-stroke key/dummy element)

Handwheel port + key-operated switch

Key-operated switch (II/0/I)

EKS identification system

2 overrides

Rapid retraction buttons (SR)

Authorization locking-type switch BSS

5 x custom keys/LEDs, switching elements can be variably assigned, slide-in name strip

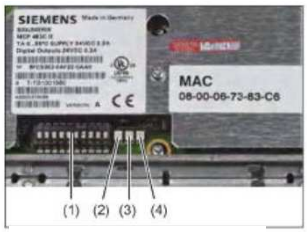
Mode selector switch

Protective shroud for EMERGENCY OFF

SITRAIN © Siemens AG 2017

Special feature Mechanical keys

Note On Ethernet MCPs, the alias address is set via a DIL switch block on the rear.
The MAC address is indicated on an adhesive label.



MCP 483C IE - LEDs
(1) Switch S2
(2) H1 – Power OK (green)
(3) H2 – Bussync
(4) H3 - Busfault

Meaning	1	2	3	4	5	6	7	8	9	10
IE, MCP address 192	Off	Off	Off	Off	Off	Off	On	On	Off	Off

Changes to addresses are activated following a restart (voltage recovery).
The address can be relocated in the PLC program (MCP1BusAdr parameter).

The default address is **192**

Other settings can be made as required.

HT8
Handheld terminal

SIEMENS

Passive components



integrated

Display of the NCU/PCU operator interface including the configured extensions

Safe operation

Large 7.5" display

Touch keys and number keys

Ergonomic acknowledgment button in the grip

Optional handwheel

Rugged design

Rugged membrane keyboard

Degree of protection IP65

Protective shroud for control elements

SITRAIN © Siemens AG 2017

Application

Machine-level operator control and monitoring, teach-in, and programming of:

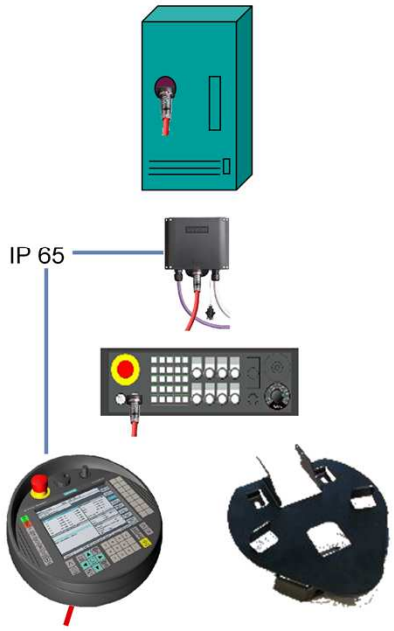
- Manipulators / robots
- Machine tools
- Production machines
- Can be supplied with pen on request

Note

The machine control panel of the HT8 is addressed in the PLC program via the FC 26.

HT8 – HT2
Connection, mount

SIEMENS



Connection options

At the control cabinet with
connection modules (optional)
Auto. EMERGENCY OFF bypass
Additional Ethernet connection

Via junction boxes (optional)
Auto. EMERGENCY OFF bypass
Additional Ethernet connection

Via MPP 483 IE

Mount

For stationary operator control and monitoring

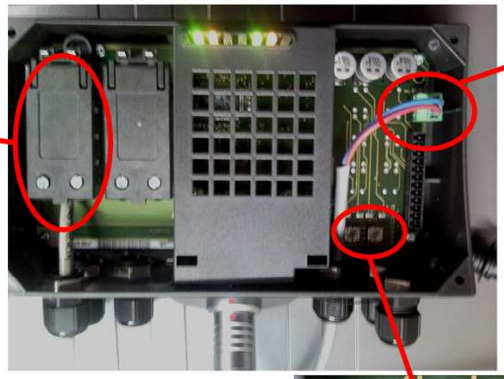
SITRAIN © Siemens AG 2017

Notes

HT8 – HT2
Terminal box PN Basic/Plus

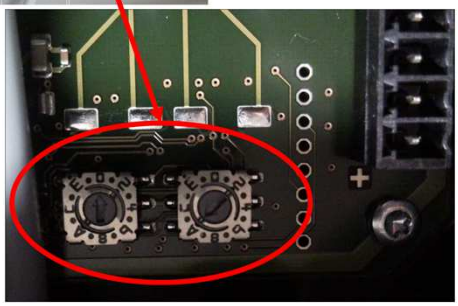
SIEMENS





Network connection

24 V supply



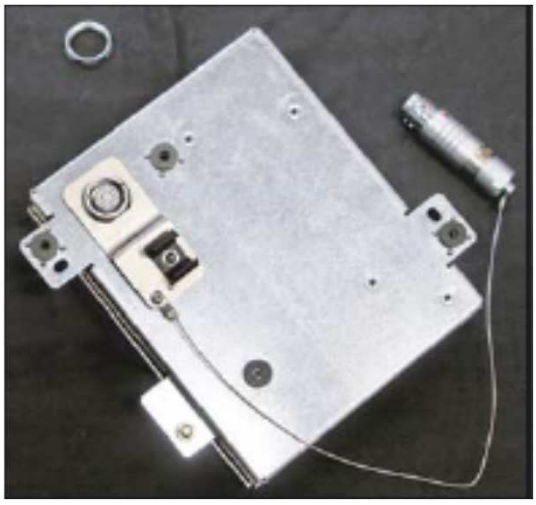
The value set at the rotary switches
Must be entered in decimal format in FB1 parameter
BHGRecGDNo (in this case, therefore, 10)

Course NC-84SL-SIP Page 2 - 56 SITRAIN © Siemens AG 2017

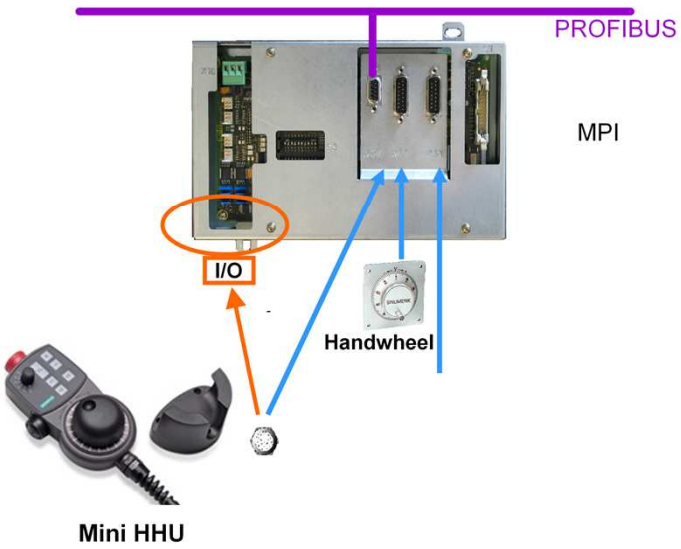
Note

A junction box is also available for mounting in the control cabinet with only the socket to which the HT2/8 is connected being routed externally.

However, the switch position for the rotary switches, which are also present, is identical.



Handwheel connection module



Connection options

for handwheels and components with handwheels without MCP or MPP

2 handwheels

6 inputs

6 outputs

Course NC-84SL-SIP Page 2 - 57 SITRAIN © Siemens AG 2017

The new redesigned mini-handheld unit is 100 % electrically compatible with its predecessor. This means that the current mini HHU can be removed from existing systems and replaced with the new successor product. The new design has a metal connector.

As the housing dimensions are smaller, there are bound to be no problems even in installation locations where space is at a premium.
Existing circuit diagrams can be left unchanged as the pin assignment has been retained. Due to the compatibility of the redesigned mini HHU with its predecessor, the existing accessories (connection kits, etc.) can continue to be used. The mini HHU can be used in conjunction with Sinumerik power line and Sinumerik solution line.

SIEMENS



Chapter 3.1

Operation of the machine control panel and operator panel

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 1
OP012 operator panel 2
Machine area, return, ETC key, area switchover 3
Shift key, channel switchover, acknowledge alarm, information key 4
Select window, cursor, page, select, end of line 5
Edit key, delete character, blank, input key 6
Additional keys OP012 7
MCP 483/MCP 483 IE machine control panels 8
MCP – Operating modes 9
Machine control panel machine functions 10
Machine functions INC 1 to 10000 11
Axis and direction keys, MCS/WCS switchover 12
Machine control panel program control 13
Machine control panel feed control 14
Machine control panel spindle control 15

Learning objectives

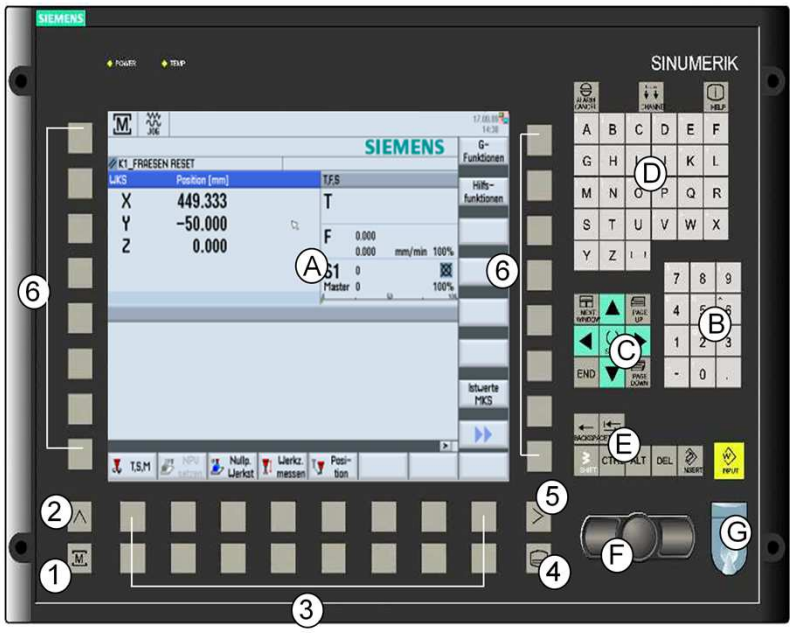
- You will know how to operate machine control panels
- You will know how to operate operator panels



Course NC-84SL-SIP Page 3.1 - 2 SITRAIN © Siemens AG 2017

Notes

OP012 operator panel



 SITRAIN © Siemens AG 2017

Description

The OP012 slimline operator panel has a membrane keyboard with 59 keys, and two horizontal and two vertical softkey bars, each consisting of eight keys.

The standard user interface Sinumerik Operate supports only two softkey bars.

An external keyboard or mouse can be connected via the USB interface.

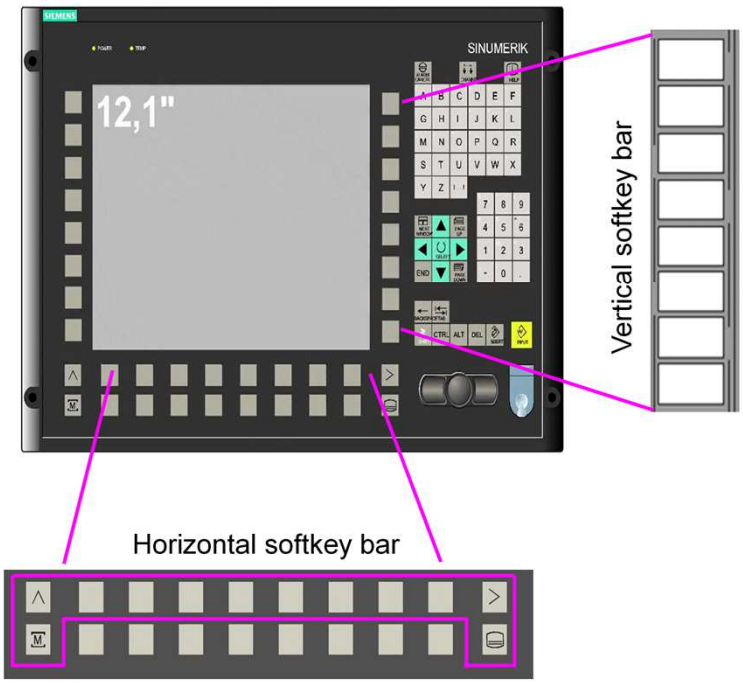
The separate alpha, numeric, and cursor pads on the membrane keyboard provide enhanced ease of use for operating and programming the control.

Keyboard areas

- **A** Display
- **B** Numeric pad
- **C** Cursor pad
- **D** Alpha pad
- **E** Offset and control keys
- **F** Integrated mouse
- **G** USB interface for keyboard or mouse

- **1** Machine area key
- **2** Return or Recall key
- **3** Horizontal softkey bar (up to 3 bars)
- **4** Area switchover key
- **5** Menu expansion or ETC key
- **6** Vertical softkey bar (two bars)

Machine area, return, ETC key, area switchover



Course NC-84SL-SIP Page 3.1 - 4 SITRAIN © Siemens AG 2017

Machine area key



Direct branch to the *"Machine"* operating area. This key will take you back to the main "Machine" screen from any menu or operating area. The machine area key is **always** functional.

Return or Recall key



Return to the higher-level menu. Recall is used to exit a window. The Recall key is only functional if it is displayed on the *lower left-hand side of the screen*.

ETC key



Expansion of the horizontal softkey bar in the same menu. The ETC key is only functional if it is displayed on the *lower right-hand side of the screen*.

Area switchover



You can call the main menu from any operating area by pressing this key. *Press the key twice* to switch from the current to the previous operating area and back again.

The Standard main menu branches to the following operating areas:

1. Machine
2. Parameters
3. Program
4. Program Manager
5. Diagnostics
6. Setup

Shift key, channel switchover, acknowledge alarm, information key

SIEMENS

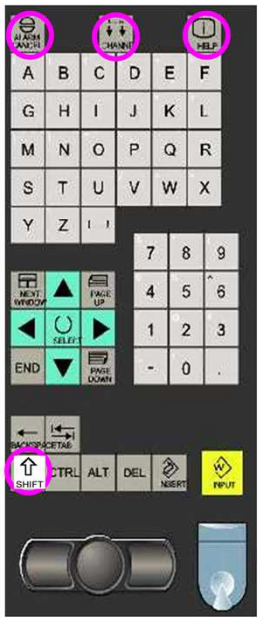
OP010/OP010C keypad



KB310C keypad on the OP010S or OP012T



OP012 keypad



Course NC-84SL-SIP Page 3.1 - 5 SITRAIN © Siemens AG 2017

Shift



Switches between functions accessed via keys with double assignment. With the OP012, this key can be configured as *"Single Shift"* and *"Permanent Shift"* (press twice); i.e. the characters that are displayed on the top row of keys are output when the keys are pressed.

Switch channel



It is possible to switch between channels when more than one is in use.

When a "Channel menu" is configured, all existing communication links to other NCUs plus the associated channels are displayed on softkeys.

Acknowledge alarm



Press this key to acknowledge the alarm marked by this Cancel symbol.

Information or help key

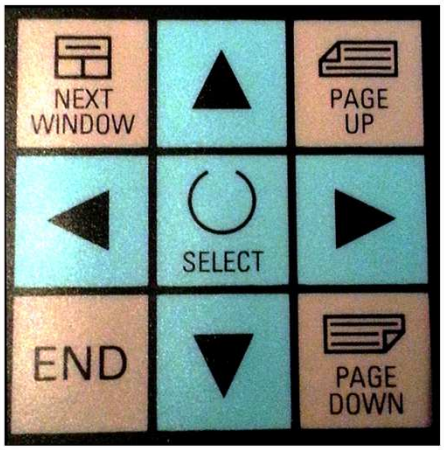


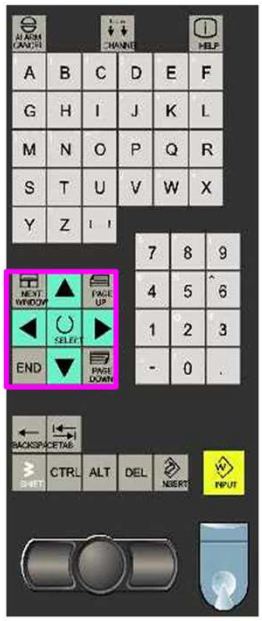
This key displays explanatory information about the current operating status (e.g. dialog programming, diagnostics, PLC, alarms).

Select window, cursor, page, select, end of line

SIEMENS

OP012 keypad





 SITRAIN © Siemens AG 2017

Select window



If several windows are displayed on the screen, it is possible to make the next window the active window by using the window selection key. (The active window has a title bar.)

Keyboard input, e.g. the page keys, is possible only in the active window.

Cursor



Use the "cursor" to move from the current position on by one line or entry field.

Page



Use the *"Page"* key to page down or up one page at a time. In a part program you can *"page"* the display down (toward end of program).

With the page keys, you scroll the visible/displayed area of the window that is active. The scroll bar indicates which part of the program/document/... is selected.

Select key



Selection key for values set in input fields and selection lists labeled with this key symbol.

Activate/deactivate a field:

☒ Active	◉ Active
☐ Not active	○ Not active

End of line

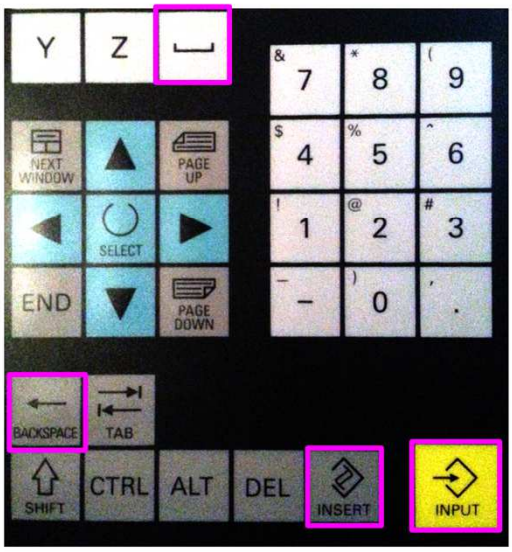


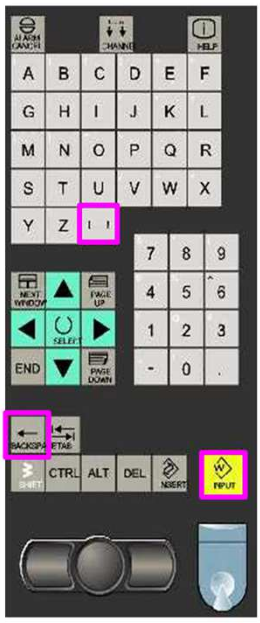
The End of line key has several functions:

- With this key the cursor is moved to the end of the highlighted line in the input fields or in the editor.

Edit key, delete character, blank, input key

OP012 keypad





SITRAIN © Siemens AG 2017

Edit key/Undo key



Switch over to edit mode in tables and entry fields (in this case, the input field is in insert mode) or

UNDO function on table elements and entry fields. (The value is not validated when you exit a field using the edit key; instead it is reset to the previous value.)

UNDO

Backspace



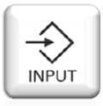
Delete characters from right to left.

Blank, space



This allows you to insert a blank in the base keyboard layout or an underscore with the dual key assignment.

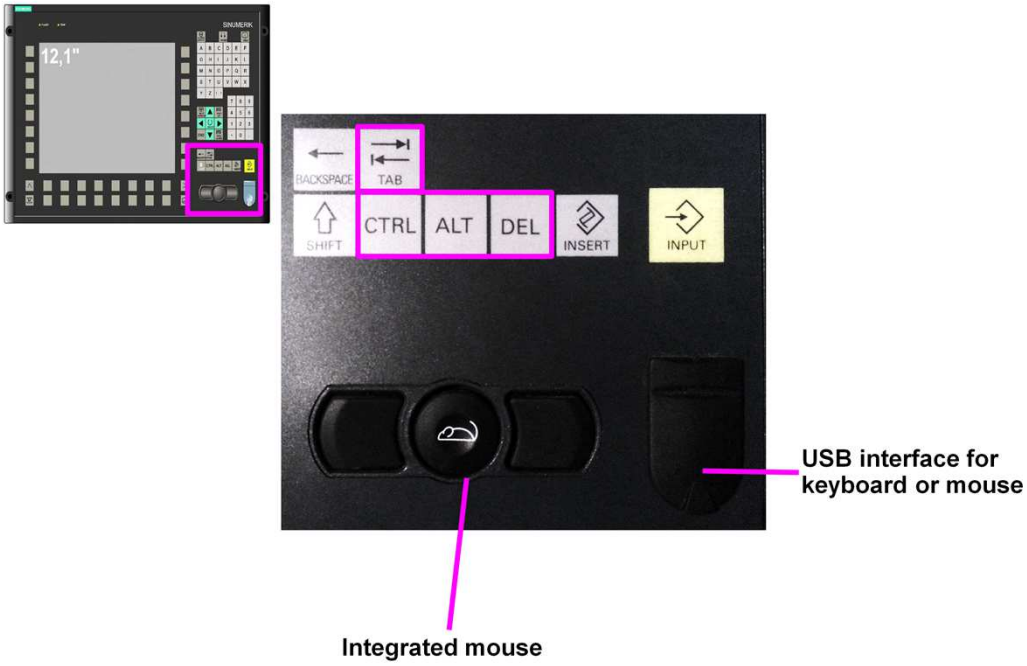
Input key



The Input key has several functions:

- Accepting an edited value
- Opening/closing a directory
- Opening a file

Additional keys OP012



Course NC-84SL-SIP Page 3.1 - 8 SITRAIN © Siemens AG 2017

Tabulator



The Tab key allows quick positioning on entry fields.

Control



The Control key allows access to special functions via keyboard combinations (copy, paste, etc.).

ALT



The ALT key allows access to various input modes in conjunction with function keys.

DEL



The Delete key allows you to delete a parameter field; the parameter field remains blank. In the text editor, you can also use it to delete the character the cursor is currently positioned on.

MCP 483/MCP 483 IE machine control panels



Course NC-84SL-SIP Page 3.1 - 9 SITRAIN © Siemens AG 2017

General

Actions on the machine tool (traversing the axes or program start, for example) can only be initiated via a machine control panel.

The machine tool can be equipped either with a standard machine control panel from SIEMENS or with a specific machine control panel from the machine tool manufacturer.

The following section provides a description of the SIEMENS machine control panels. If you are using a different machine control panel, please refer to the machine tool manufacturer's operating guide.

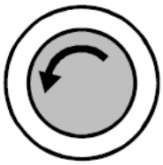
MCP – Operating modes





Course NC-84SL-SIP Page 3.1 - 10 SITRAIN © Siemens AG 2017

EMERGENCY OFF



Press the red button in emergency situations:

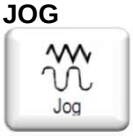
1. If life is at risk,
2. If there is a risk of damage to the machine or the workpiece.

As a rule, when the EMERGENCY OFF is activated, all drives are brought to a controlled standstill with the greatest possible braking torque.

For other reactions to the EMERGENCY OFF signal, please refer to the respective machine tool manufacturer's specifications.

Operating modes

JOG



Axis travel in jog mode via:

- Continuous motion of the axes using the direction keys
- Incremental movement of the axes by activating the direction keys or the handwheel.

MDA



Manual Data Automatic

Control of machine through execution of a block or a sequence of blocks. The blocks are input via the operator panel.

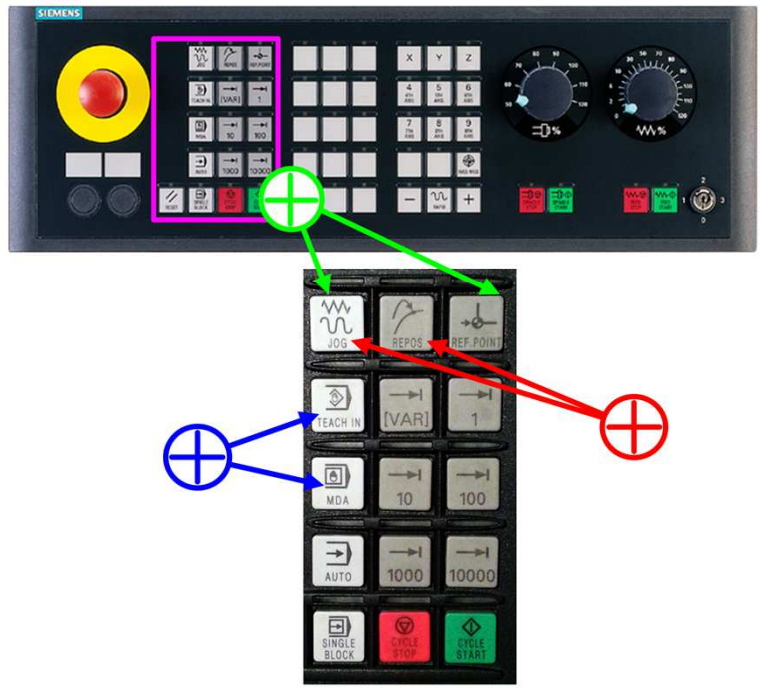
AUTO



Automatic

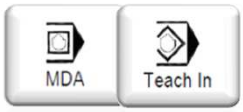
Control of machine through automatic execution of programs.

Machine control panel machine functions



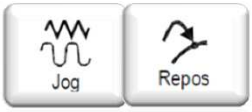
Course NC-84SL-SIP Page 3.1 - 11 SITRAIN © Siemens AG 2017

TEACH IN



Creation of programs in interactive mode with the machine in "MDI" mode. In other words, the geometries of a workpiece can be determined either partially or in full by approaching the required position with the tool or a probe. After saving the coordinates, you can then enter the required technological functions, e.g. by inputting data manually.

REPOS

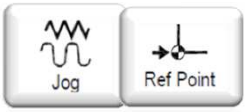


Reposition

Return to contour in "JOG" mode. This function provides a means of repositioning to the abort position directly in "JOG" operating mode following program abort in "AUTO" operating mode.

The control will allow manual traverse in all axes until the point of interruption only. You cannot traverse beyond this point with REPOS.

REF



Reference point approach

Reference point approach in "JOG" mode. This machine function allows you to reference your machine tool, i.e. synchronize the actual value display and axis position.

Please refer to your machine manufacturer's instructions for more information on how the reference point approach is conducted on your specific machine (automatic referencing of axes, axis sequence, etc.).

Machine functions INC 1 to 10000





INC keys

Course NC-84SL-SIP Page 3.1 - 12 SITRAIN © Siemens AG 2017

INC function

You can activate the INC functions in conjunction with the following modes:

- *"Jog"* mode
- *"MDI / Teach In"* mode

INC VAR



*Inc*remental Feed *var*iable

Incremental mode with variable incremental size that can be defined in the input screen form in the Machine operating area.

INC

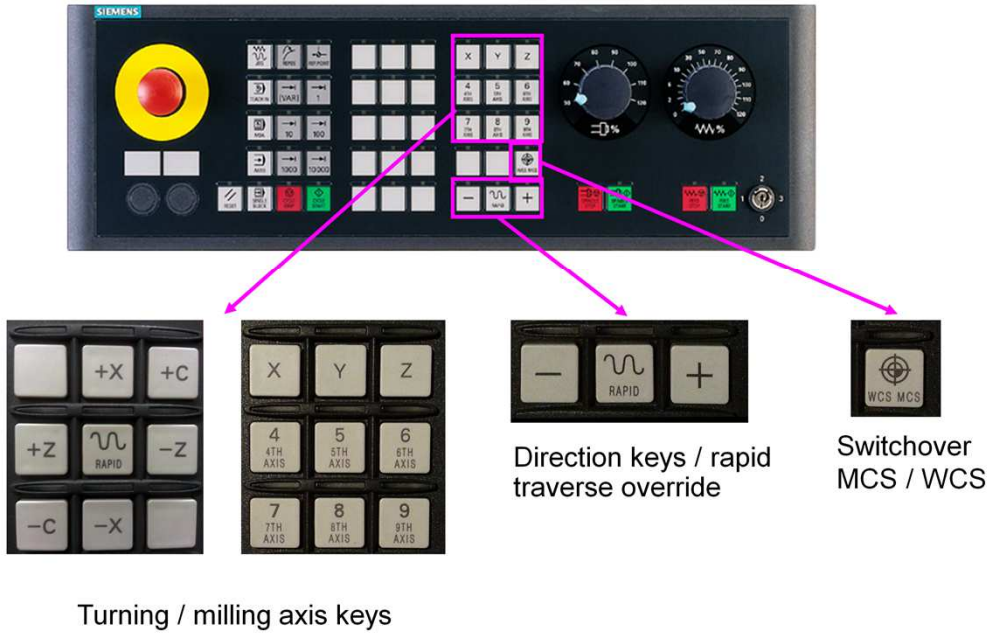


*Inc*remental Feed

Incremental mode with fixed incremental size of 1, 10, 100, 1,000, or 10,000 increments.

The increments are set in µm by default. You can change this setting to mm in the machine data.

Axis and direction keys, MCS/WCS switchover



Course NC-84SL-SIP Page 3.1 - 13 SITRAIN © Siemens AG 2017

Turning machine axis keys



In the case of a turning machine, the axis keys are usually arranged in a cross representing the corresponding directions of movement.
You traverse the selected axis (X ... Z) in the positive direction.

Milling machine axis keys



…

Unlike with turning machines, the axis and direction keys are separated in the case of the milling configuration. Therefore, you first need to select the required axis via its axis key before traversing the axis with the direction keys "**+**"/"**-**".

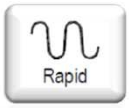
You can change the default axis identifiers 4...9 to customized axis names by attaching new names to the keys.

Direction keys



Direction keys are for specifying the travel direction of the selected axis. The speed at which this axis is traversed depends on the entries in the machine data and setting data.

Rapid traverse override



Press this key together with the "+" or "-" key to move the selected axis in rapid traverse in jog mode. The speed at which this axis is traversed depends on the entries in the machine data and setting data, if applicable.

MCS/WCS switchover



This key on the MCP is for switching between using the machine coordinate system and the tool coordinate system as the reference system. You can also switch the actual value display on the screen to whichever coordinate system is active by setting the appropriate parameters on the PLC. You can also change the actual value display by pressing the "MCS/WCS" softkey on the OP.

Machine control panel program control



 SITRAIN © Siemens AG 2017

Program control

The program control keys allow the operator to start and stop execution of a part program as well as to execute the program block by block or reset program execution.

NC Start



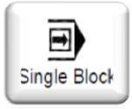
If you press the "NC Start" key, the selected part program (the name of the part program is displayed in the OP header) is started at the current block and the associated LED lights up.

NC Stop



If you press the "NC Stop" key, processing of the active part program is halted and the associated LED lights up. You can resume execution with NC Start.

Single block



This function allows you to execute a part program block by block. You can activate the "single block" function in "Automatic" and "MDI" modes. If single block is activated, the associated LED on the MCP lights up.

Reset



When you press the "Reset" key, execution of the current part program is interrupted. Messages from the monitoring function are deleted (except for POWER ON alarms).

The channel state changes to "Reset", i.e.:

- The NC control remains synchronized with the machine,
- The control is in its initial state and ready for a new program run.

Machine control panel feed control

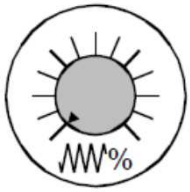


Feed control

Course NC-84SL-SIP Page 3.1 - 15 SITRAIN © Siemens AG 2017

The specified increment and the control range apply to standard machine data (MD). These MD can be edited by the machine-tool manufacturer to suit the specific application. The max. axis velocity and the values for the feedrate override position are specified in the machine data or setting data.

Feedrate override



The rotary selector switch with detent positions allows the user to reduce or increase the programmed feedrate value "F" (corresponds to 100 %).

The set feedrate "F" is displayed as an absolute value and as a percentage value in the "Feed" screen
(vertical softkey in "Machine" operating area).

Control range: 0 % to 120 % of the active feedrate setpoint

Increment: 1/2/5/10 % depending on the detent position

Feed Stop



When you press the "Feed Stop" key:

- Execution of the current program is stopped,
- The axis drives are brought to a controlled standstill,
- The associated LED lights up as soon as feed stop is accepted.
- The header section (channel status display) will display FST (= Feed STop)

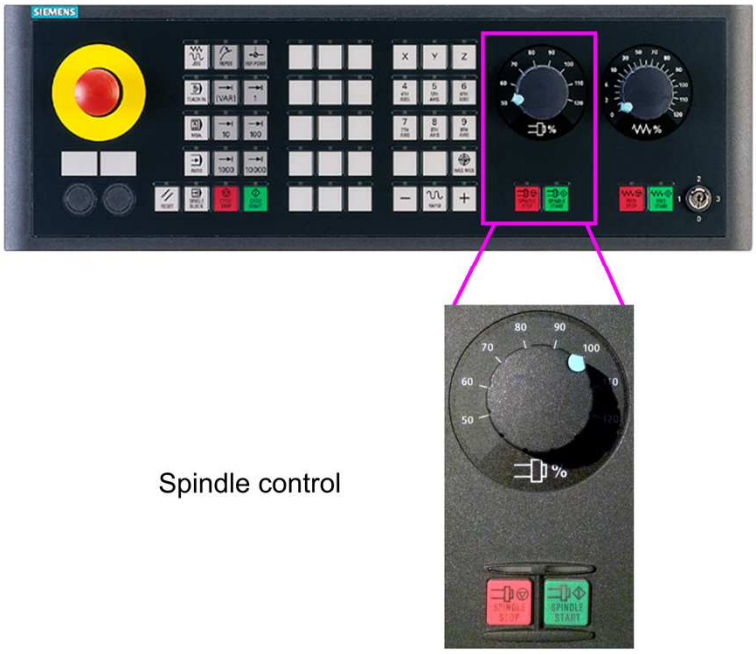
Feed Start



When you press the "Feed Start" key:

- Execution of the part program is continued in the current block
- The feedrate is accelerated to the value specified by the program
- The associated LED lights up as soon as feed start is accepted

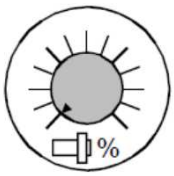
Machine control panel spindle control



 SITRAIN © Siemens AG 2017

The specified increment and the control range apply to standard machine data (MD). These MD can be edited by the machine-tool manufacturer to suit the specific application. The maximum spindle speed and the values for the spindle speed override position are defined in the machine data and setting data (see information supplied by the machine-tool manufacturer).

Spindle speed override



The rotary switch with detent positions allows you to increase or decrease the programmed spindle speed "S" (corresponds to 100 %).

The set spindle speed value "S" is output as an absolute value and a percentage in the "Spindles" screen (vertical softkey in "Machine" operating area).

Control range: 50 % to 120 % of the active spindle setpoint speed

Increment: 5 % between detent positions

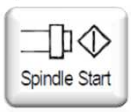
Spindle Stop



When you press the "Spindle Stop" key:

- The spindle is decelerated to zero speed
- The associated LED lights up as soon as "spindle stop" is accepted

Spindle Start



When you press the "Spindle Start" key:

- The spindle speed is accelerated to the value defined in the program
- The associated LED lights up as soon as "spindle start" is accepted by the control

SIEMENS



Chapter 3.2

Operating Sinumerik Operate

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Overview of operating areas (OA) 3
Screen structure 4
Machine operating area 5
Machine OA 11
Parameters operating area – Tool offsets 12
Parameters operating area – User variables 13
Program and Program Manager operating areas 14
Diagnostics operating area – Alarm list 15
Operating area Setup 16
Programming with programGUIDE 17
Programming with ShopMill/ShopTurn 19
Operation with PG/PC 20
Switching on the training rack 21
HMI selection on the TCU 22
Reference point approach 23
Exercise 1: 24
Exercise 2: 25
Exercise 3: 26
Additional exercise: 34

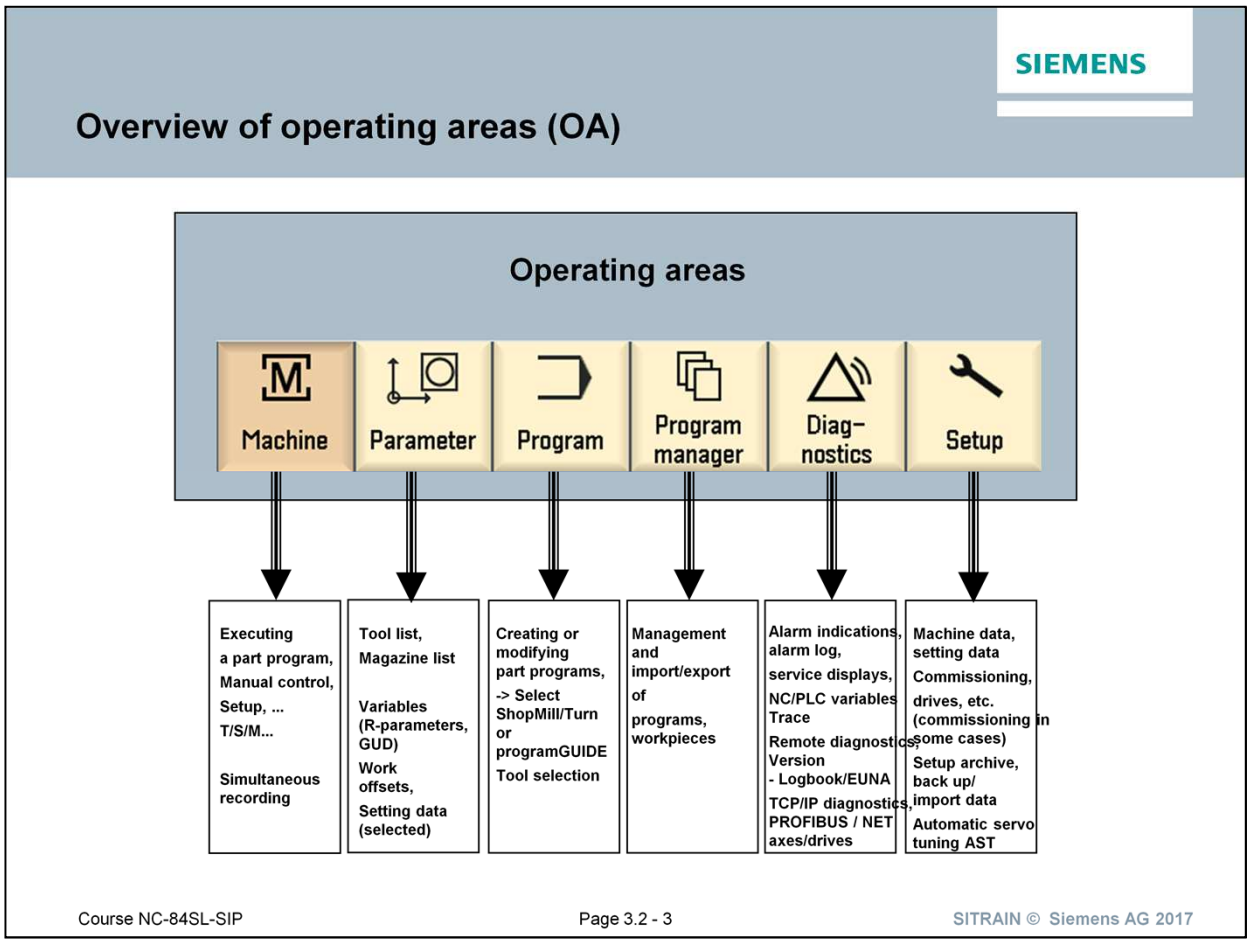
Learning objectives

- You are familiar with using Sinumerik Operate



SITRAIN © Siemens AG 2017

Notes



To facilitate operation of SINUMERIK, important data has been grouped in various operating areas (**OA**) for users. The user can call up all the functions via the control system user interface.

The SINUMERIK 840Dsl provides the following operating areas:

Machine

In the "Machine" operating area, the operator can perform all the necessary operations from setting up the machine to automatic execution of part programs. The "Machine" operating area will display different softkeys (**SK**) with different functionalities depending on the operating mode

Parameters

The operator or machine setter can enter data for tools, work offsets, R-parameters and variables in the Parameter operating area. Here, the machine can be adapted to the particular workpiece using setting data.

Program

The "Program" operating area provides all the necessary help and functions to easily modify and create programs.

Program Manager

Programs can be managed, backed up or reimported via the "Program Manager" operating area.

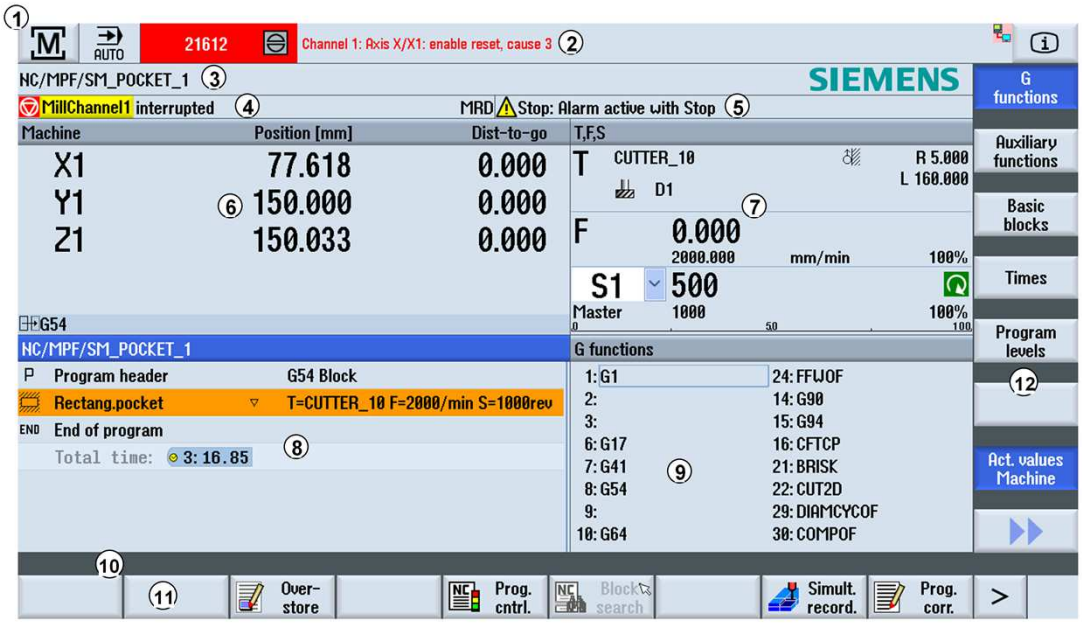
Diagnostics

In case of an error, you can display help for the current message or alarm in the "Diagnostics" operating area. The alarm log is also located in this area.

Setup

The "Setup" operating area is mainly used by commissioning and service engineers/specialists to access all data areas of the NC.

Screen structure



SITRAIN © Siemens AG 2017

The display is structured into the areas shown in the diagram above:

1 Active operating area and mode
2 Alarm/message line
3 Program name
4 Channel state and program control
5 Channel operational messages
6 Axis position display in the actual value window
7 Window "T,F,S": Display for
- active tool T
- present feedrate F
- active spindle with present status (S)

8 Operating window with program block display
9 Display of active G functions, all G functions, H functions and input window for different functions (for example, skip blocks, program control)
10 Dialog line to provide additional user notes
11 Horizontal softkey bar
12 Vertical softkey bar

Machine operating area



SITRAIN © Siemens AG 2017

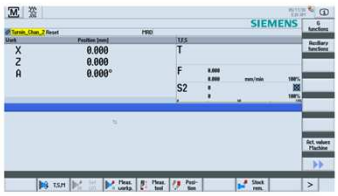
The "Machine" operating area includes all functions and influencing variables that initiate actions at the machine tool or detect its status.

The "JOG" mode is used for manual operation and for setting up the machine. For setting up, the functions reference point approach, repositioning, handwheel or traversing in specified increments and setting new absolute (machine) zero are available.

Machine operating area
JOG mode (user interfaces)



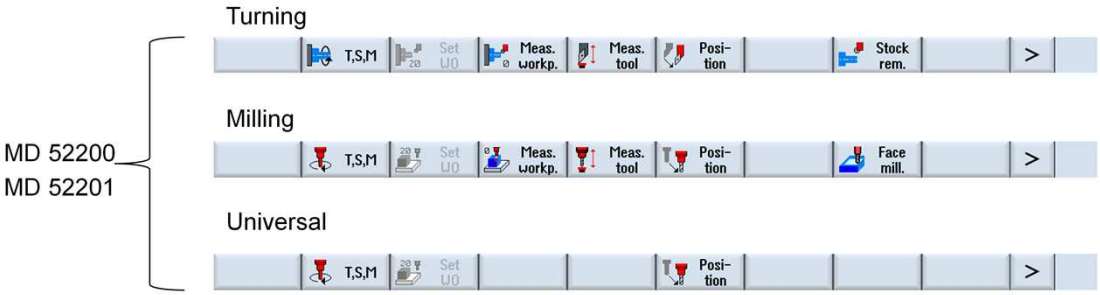
Universal



Turning



Milling

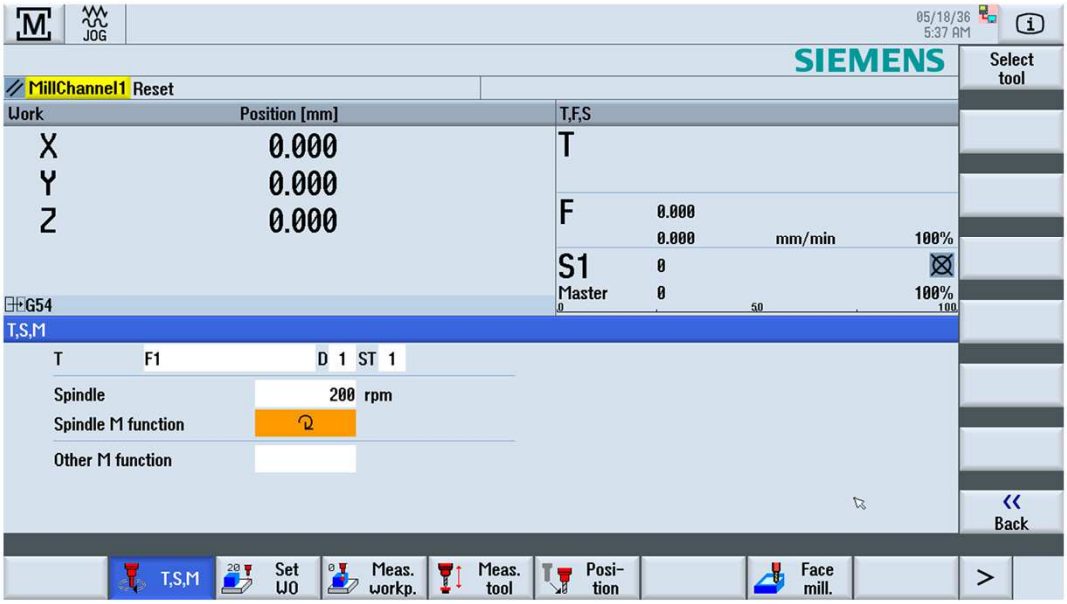


SITRAIN © Siemens AG 2017

JOG

"JOG" mode is used for manual operation and for setting up the machine. For setting up, the functions reference point approach, repositioning, handwheel or traversing in specified increments and setting new absolute (machine) zero are available.

Machine operating area
T,S,M function



 SITRAIN © Siemens AG 2017

T, S, M function

Allows the loading of a tool and the switching of

- Spindle speed and direction
- M functions (auxiliary functions)
- Selection of a work offset for machining
- Selecting the measurement unit (mm/inch)

Procedure: JOG operating mode > T, S, M > Select tool > In Manual (the tool is now loaded into the spindle)

- Enter the speed, e.g. 2000 rpm
- Select the direction of rotation using the SELECT key
- Other M functions (auxiliary functions), e.g. enter 7 for M7
  Coolant on
- If required, select a work offset
- Procedure JOG operating mode > T, S, M > Select WO > e.g. check G54 and switch on with In Manual
- Select the measurement units using the Select key if required
  The switch-on state is mm

The set values can now be started using the NC Start button (cycle start – green button) for manual operation.

Machine operating area
MDI mode

SIEMENS

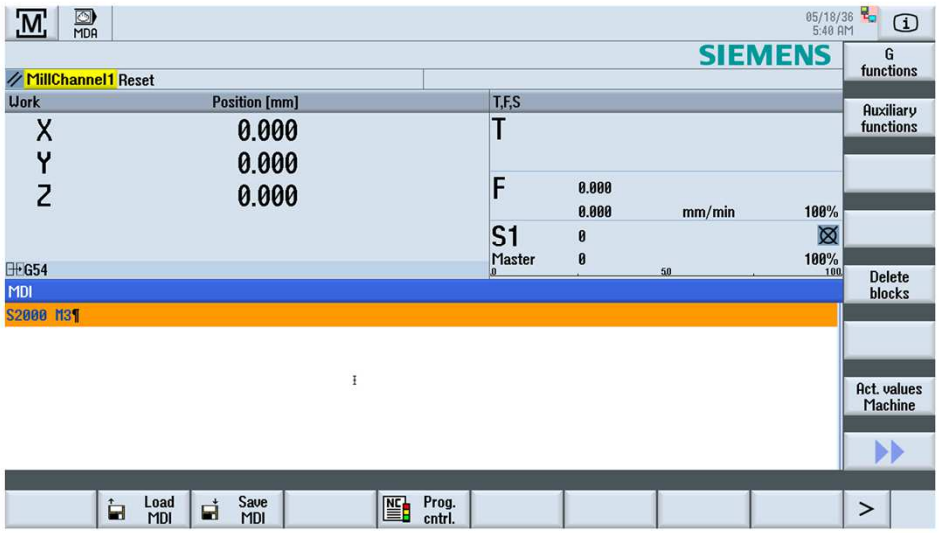
e. g.
G0 G90 X100 Y100
M30

Or

S2000 M3

Or

T1 M6
S2500 M3
G0 G90 Y250 Y150
X150 Y100
X0
Y0
X150
Y100
G0 X250 Y150
M30



 SITRAIN © Siemens AG 2017

In "MDI" (**M**anual **D**ata **I**nput) mode, you can write part programs block by block and execute them. To do this, you can use the keyboard to selectively enter the required motion as individual part program blocks.

The control processes the entered program blocks when you press the "NC Start" key.

Note You need the same safety interlocks and prerequisites as for fully automatic operation. In "MDI" mode, all automatic functions are active.

Notes The Program Control horizontal softkey is explained in detail in the section "Machine" operating area "AUTO" mode.

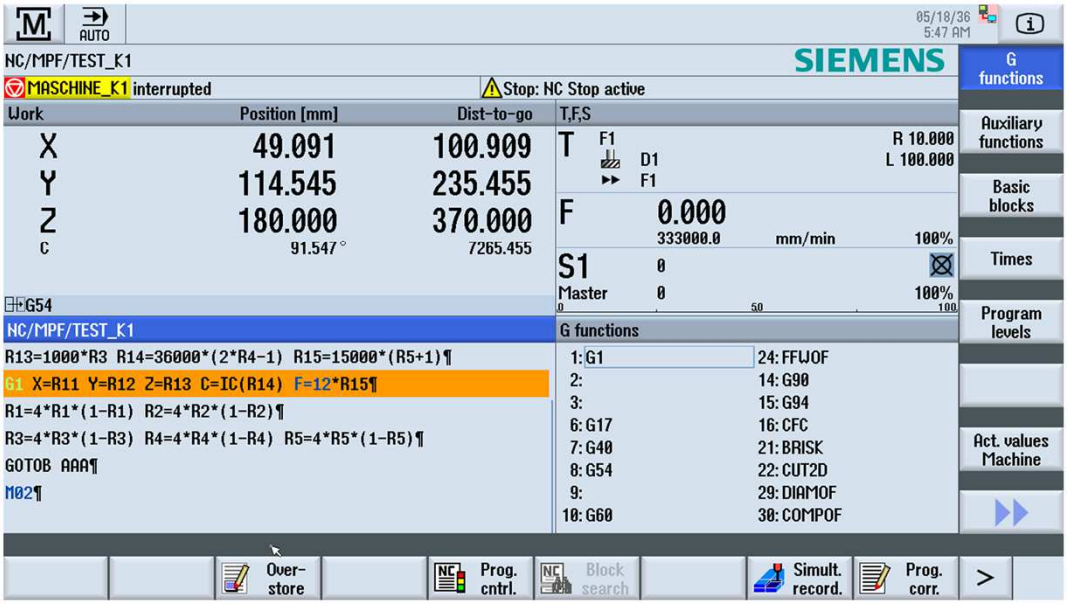
MDI buffer Programs generated in MDI mode can be saved in the program memory using the "Save MDI" softkey.

With "Load MDI", part programs can be loaded from the program memory into the MDI memory.

Tip If MDI fails with the message
'Program has already been opened by another application',
this is due to the HMI active on the PCU which is causing this blockage.

Remedy: Switch HMI on PCU to operating area OA

Machine operating area
AUTO mode



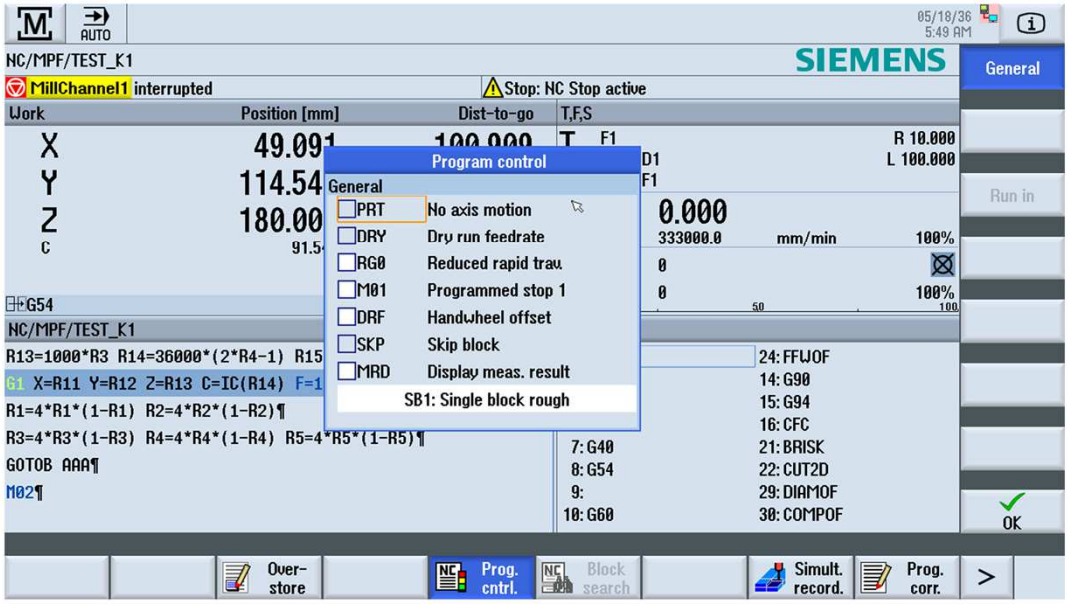
 SITRAIN © Siemens AG 2017

In "AUTO" mode, the part programs are executed fully automatically.

This is the operating mode for normal operation of part machining.

The following conditions must be met before you can execute part programs:

- You have already synchronized the measuring systems of the axes (reference point approached).
- You have already selected the associated part program in the control.
- You have already checked and entered the necessary offset values, tool offsets, ...
- The required safety interlocks are activated.



 SITRAIN © Siemens AG 2017

Program control

In "AUTO" and "MDI" modes, this function can be used to change the execution of a program. The following program controls can be activated or deactivated

- SKP

Skip block; up to max. 10 skip levels can be activated. With more than one skip level, you obtain an additional selection screen

- PRT Program testing
- DRY Dry run feedrate
- RG0 Reduced rapid traverse
  If this is selected, a reduction factor for the rapid traverse override can be entered under softkey: Settings. (SD 42122)
- M01 Programmed stop
- DRF DRF selection (handwheel override)

Single block selection

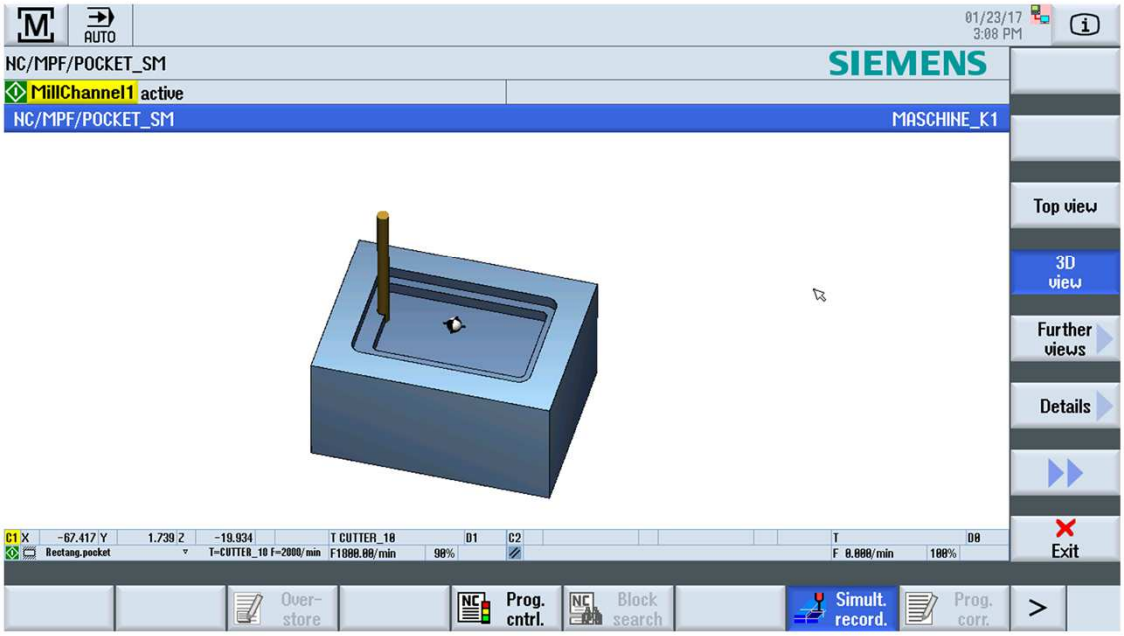
- SB1 Single block, coarse
- SB2 Calculation block
- SB3 Single block, fine

Display

All blocks, e.g. of comments are displayed in

operating area Program > Editor opened > Double expansion key > Settings

Machine OA
Simultaneous recording

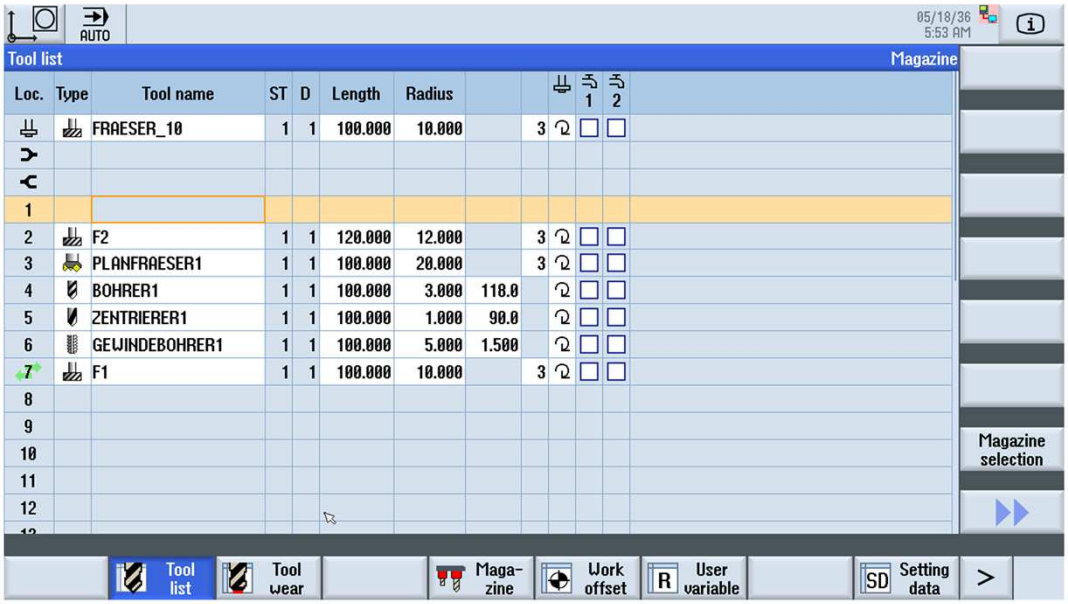


Course NC-84SL-SIP Page 3.2 - 11 SITRAIN © Siemens AG 2017

Simultaneous recording during machining of the workpiece

If the view of the working area is blocked by coolant, for example, while the workpiece is being machined, you can also track the program execution.

Parameters operating area – Tool offsets



 SITRAIN © Siemens AG 2017

Tool list

This is the list of tools with their geometry data.

The spindle and gripper locations are at the top of the list.
They are followed by the tool magazine locations and finally by the tools outside the machine (loading magazine).

The following functions are available:

- Generate / Delete / Load / Unload

Tool wear

The tool wear data can be accessed here.

OEM tool data

This softkey (horizontal softkey no. 3 = HSK 3) is only available if it has been configured by the machine manufacturer.

Magazine

The magazine list contains additional properties such as:

- Location locked
- Oversize tool
- Fixed-location-coded tool

Tools can also be moved to different locations

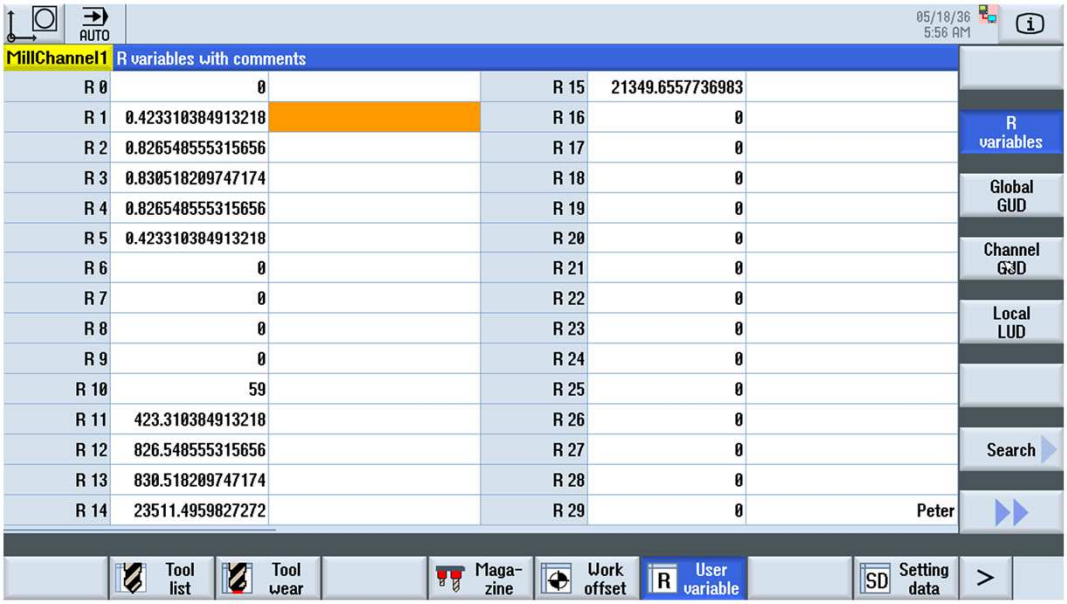
Other data in the Parameters operating area

Zero offsets
Settable work offset, angle of rotation, etc.

User variables
R-parameters and user data (GUD, LUD)

Setting data
Working area limitation, spindle settings
Other setting data can be accessed under "Machine data".

Parameters operating area – User variables



 SITRAIN © Siemens AG 2017

User variables

That includes the **R parameters** and the user-defined variables **GUD, LUD.**
The list is opened with softkey: **User variables**.

R parameters

R parameters are used as "placeholders" in NC programs. You can use them to rapidly adapt e.g. drilling depths, lengths and feed values. The name **R** parameter comes from "**R**echenparameter" (arithmetic parameter).

R parameters can be modified manually in this input screen.

The number of available R parameters is defined in a machine data item.
A maximum of R0 ... R32534 can be created; no gaps occur in the numbering.

Search

Using "Search" softkey, an input window appears for a parameter.

Enter the R parameter number you wish to find via the numeric keypad. After pressing the "Input" key, the system automatically positions to this R parameter number.

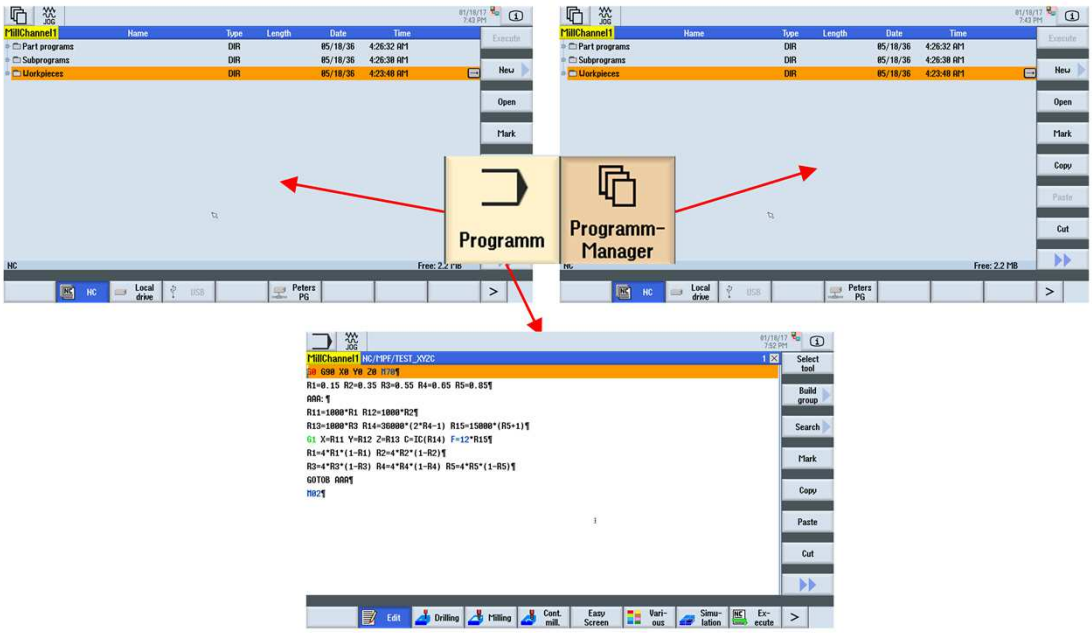
Further notes

R parameters are channel-specific, i.e. a two-channel machine has parameter R1 twice. A key-operated switch can be used to prevent parameters being entered and deleted.

GUD, LUD

Can be viewed here
These are variables (-> softkey: **Global GUD / Channel GUD**) that are declared in special definition files (SGUD, MGUD, UGUD, GUD4 to GUD9)
or local variables (->softkey: **Local LUD**) declared in the currently active part program.

Program and Program Manager operating areas



Course NC-84SL-SIP Page 3.2 - 14 SITRAIN © Siemens AG 2017

Operation Program and Program Manager operating areas

Explanation As their names suggest, part programs are managed in these two operating areas.

- OA: Program Program editor (can only be accessed if a program is being edited)

- OA: Program Manager

Program management

The "Program" softkey takes you to the same menu as the "Program Manager" softkey (OA: Program Manager) as long as a part program is not open in the editor.

If a part program is open, the "Program" softkey will also take you to the Program operating area (see also the symbol in the top left-hand corner).

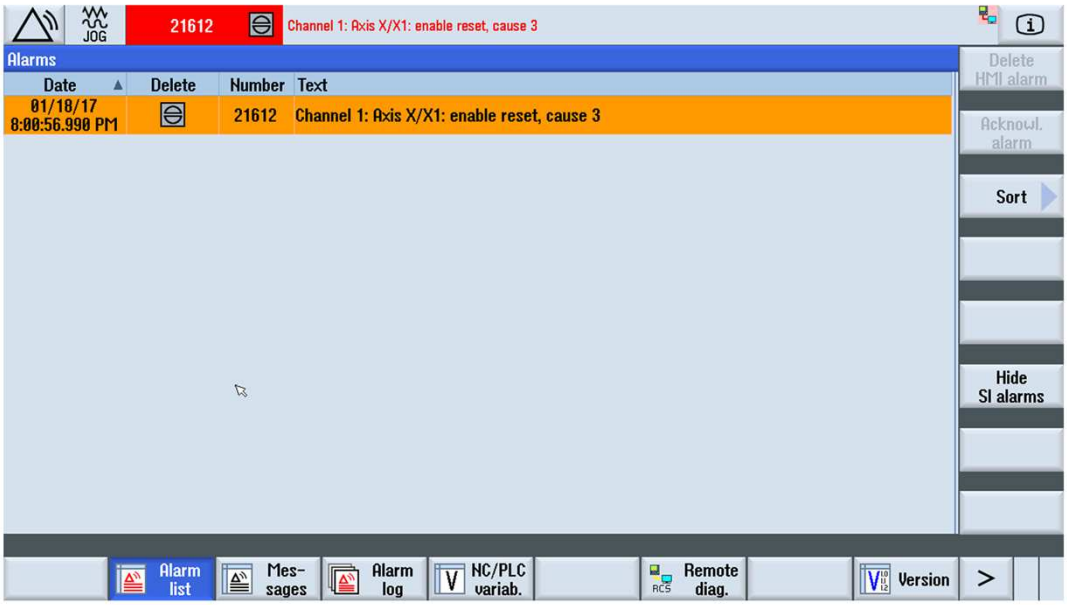
The program memory is divided into subdirectories:

Part programs Displays an overview of all main programs.

Subprograms Displays an overview of all subprograms

Workpieces You obtain a current workpiece overview. A workpiece directory contains all files that are necessary to machine a workpiece. The files can include main program files, subprogram files, or any initialization program. It can also contain comment files for a technological description of the workpiece.

Diagnostics operating area – Alarm list



 SITRAIN © Siemens AG 2017

Alarms are displayed in the alarm and message line with alarm number and alarm text. If several errors occur at the same time, all of these alarms are listed in the "Diagnostics" operating area.

Notes on resolving the error can be called using the Info key.

The time for switching to the next alarm in the alarm line can be entered in ms in the display MD 9056.

Alarm list

All alarms present are listed.
(After the cause has been removed, alarms must be acknowledged using the displayed icon.)


Acknowledged using the "NC Start" key on the MCP

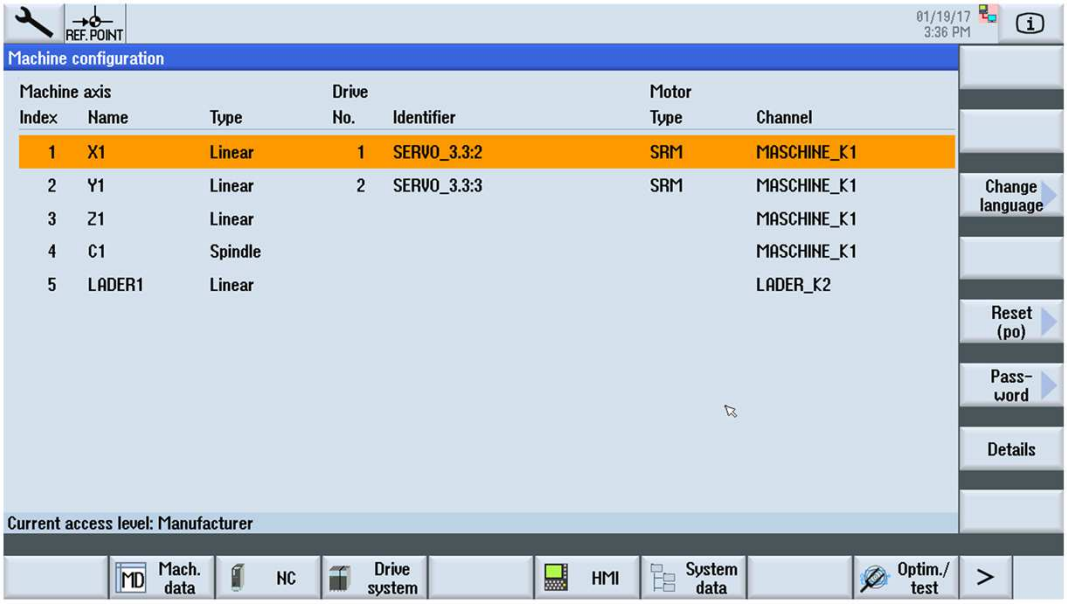


Acknowledged using the "Acknowledge alarm" key


Acknowledged using the "Reset" key on the MCP


Acknowledged using "POWER ON", i.e. the control is switched off and on again

Operating area Setup

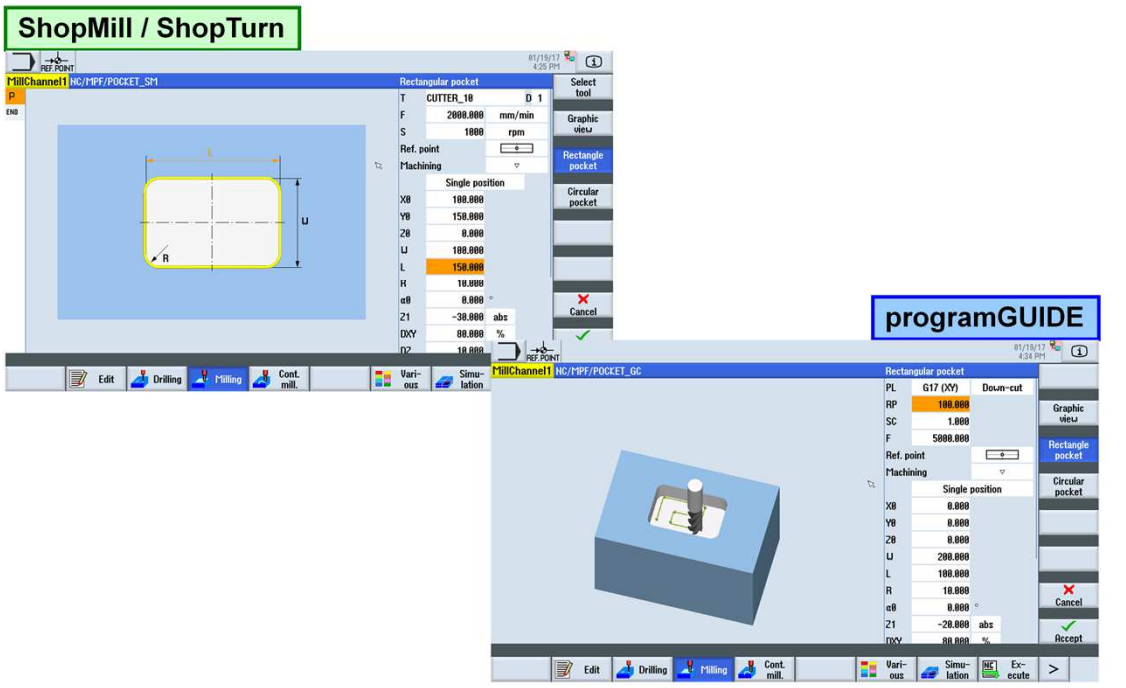


 SITRAIN © Siemens AG 2017

Setup operating area

As its name suggests, this operating area is used to commission the machine.

This operating area is described in detail in the various chapters on commissioning.



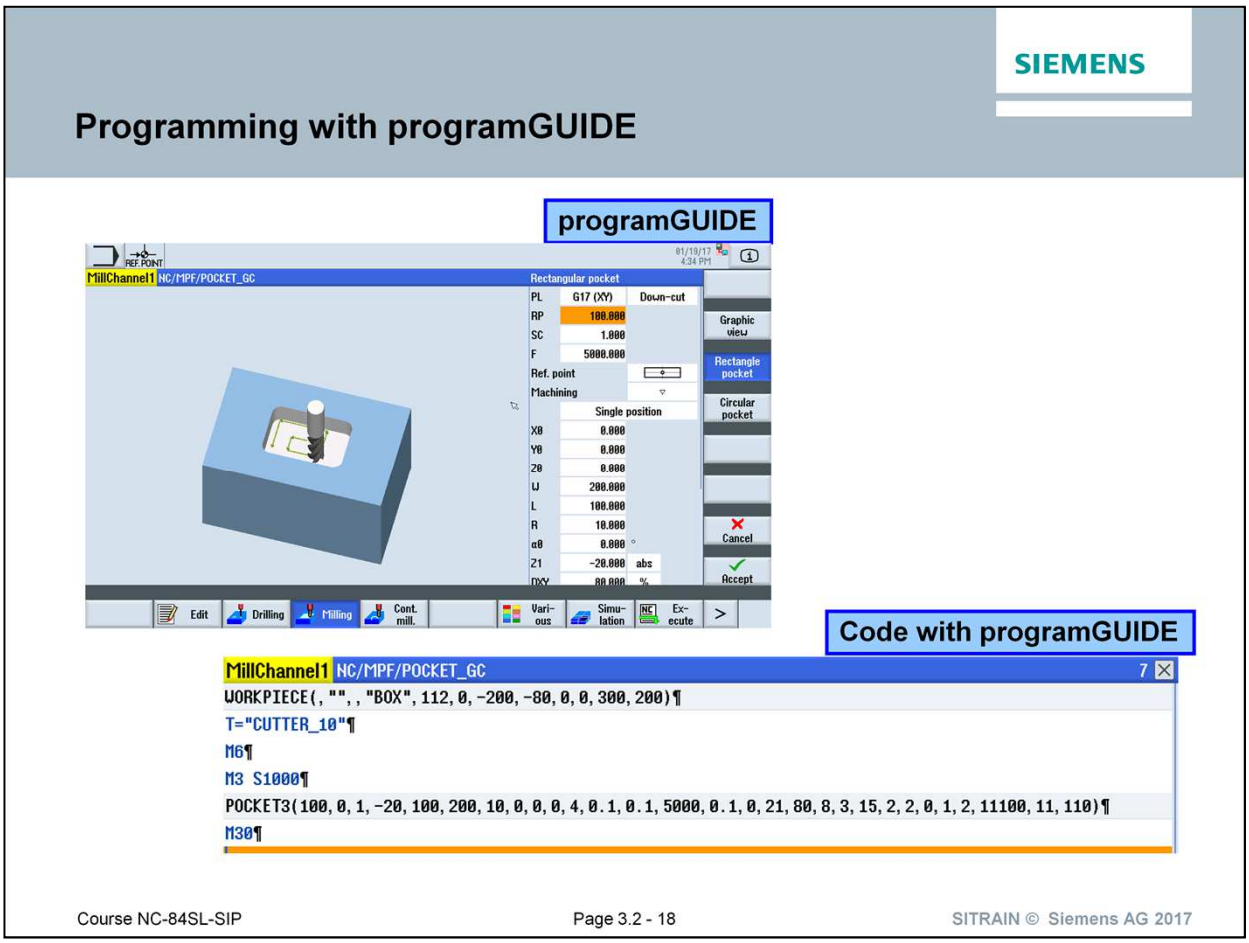
 SITRAIN © Siemens AG 2017

Programming tools

The programGUIDE and ShopMill or ShopTurn programming tools make standard machining operations such as milling rectangular pockets and drilling holes easier, etc.
The programming settings are made with graphics-based support.

Notable features of this support are:

- **Geometry** — Geometry data such as the length and width of a pocket is specified in the input screens.
- **Contour calculator** — The powerful integrated contour calculator supports the programming and graphical display of complex workpiece contours
- **Technology** — Technology data such as feedrates and infeeds can also be entered.
- **Help** — Each entry field is supported by a screen with animated elements describing the selected parameter.
- **Machining steps** — The program generated presents the corresponding machining operations (drilling, centering, grooving, or pocket milling, for example) in a simple and transparent format based on machining steps.
- **Free G code** — All programming tools also support free G code entry.
- **Test** — The programs generated can be tested using 2D/3D simulation.
  The supplementary function 'Simultaneous recording' enables the current machining status to be depicted in graphical format concurrently with program execution.



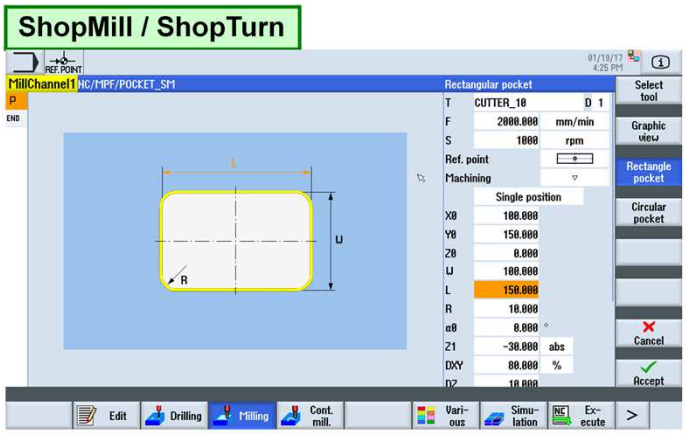
programGUIDE

Programming with programGUIDE involves the following steps:

- **Blank** Definition of the blank as a separate "cycle call" WORKPIECE(...)
- **Tool** Selection of the tool prior to machining the pocket / hole, etc.
- **Spindle** Programming of the spindle also prior to machining.
- **Machining cycle** The machining cycle (the milling of a rectangular pocket in the example above) defines the geometry and remaining technology.

Summary

ProgramGUIDE makes for flexible programming with cycle support. Of course, this might require more in-depth programming knowledge.





ShopMill Programming with ShopMill or ShopTurn involves the following steps:

- **Blank** Definition of the blank in the program header (generated immediately when the program is created)
- **Machining** Each machining operation (drilling, centering, grooving, or pocket milling, etc.) is defined by the entries in a parameterization display.

  All geometrical and technological data must be entered in **one** screen. These are:

  - **Tool** The selection of the tool is a parameterization step in the input display for machining.
  - **Geometry** The geometry data is entered in a very similar way as when working with programGUIDE.
  - **Technology** Technology data such as feedrates, infeeds, and spindle speed are additional machining parameters.

    The machining cycle (the milling of a rectangular pocket in the example above) defines the geometry and remaining technology.

Difference compared with programGUIDE

- **Encapsulation** Remember: With ShopMill or ShopTurn, all of the data entries required for a machining step are requested in one input display. This means that the steps are better "encapsulated" than with programGUIDE and fewer manual programming entries are required.

  The resulting program is more transparent.

- **Tool call** The tool function is part of the cycle.

Operation with PG/PC

Volltasta-tur	F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12
mit SHIFT	**ver-tik** Soft. **1**	**vertik** Soft. **2**	**vertik** Soft. **3**	**vertik** Soft. **4**	**vertik** Soft. **5**	**vertik** Soft. **6**	**vertik** Soft. **7**	**vertik** Soft. **8**	>	M		
ohne SHIFT	**horiz** Soft. **1**	**horiz** Soft. **2**	**horiz** Soft. **3**	**horiz** Soft. **4**	**horiz** Soft. **5**	**horiz** Soft. **6**	**horiz** Soft. **7**	**horiz** Soft. **8**	^		W	Y
Volltasta-tur:	5	Esc	Insert	Home	Page Up	Page Down	Enter					
ohne SHIFT	:	X		?	'	%						

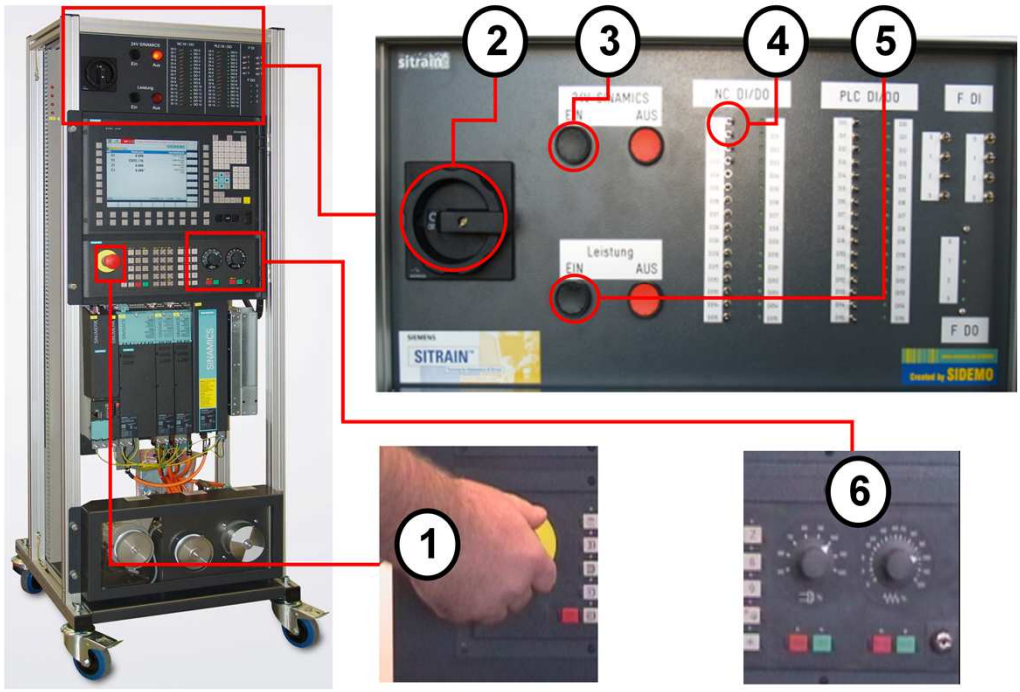
Course NC-84SL-SIP Page 3.2 - 20 SITRAIN © Siemens AG 2017

Operation PG/PC

The HMI software is also available as a PC version. System requirements for the PC version are an Ethernet interface on the PC and STEP 7 installed, at least V5.5 SP3.

Operation is via mouse or using the keyboard shortcuts represented above.

Switching on the training rack



Course NC-84SL-SIP Page 3.2 - 21 SITRAIN © Siemens AG 2017

Switching on training rack

This page describes how to switch on the training rack.

1. Release the "**EMERGENCY OFF**" switch.
2. Close the **main switch**.
3. Press the '**24V SINAMICS ON**' key to supply the NCU and drives with 24 VDC voltage.
4. Activate the first two NC inputs '**NC DI/DO**, **DI0** + **DI1**'.
5. Press the '**Power ON**' key to supply the power units with 400 V AC voltage.
6. Press the keys **Spindle Start** and **Feedrate Start** to enable the spindle and feedrate. Turn both override switches to positions other than 0 %.

Now you can traverse the axes and spindle.

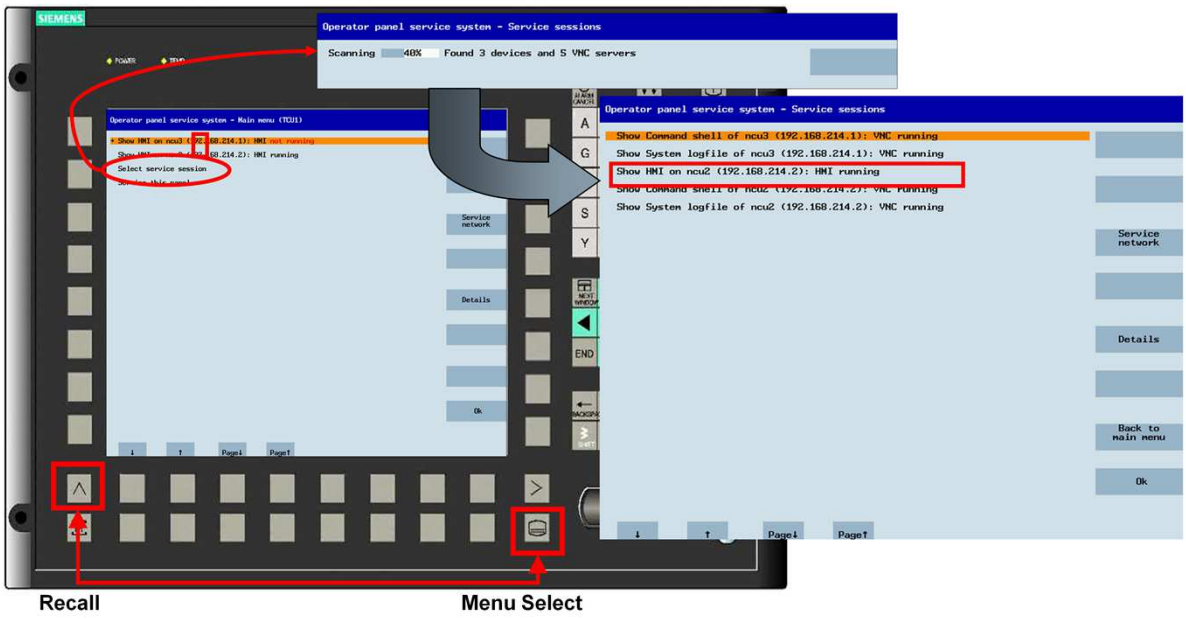
Switching off training rack

To switch off the training rack, press the "**EMERGENCY OFF**" switch.

Additional notes

It is advisable to shut down the PCU before switching it off.

HMI selection on the TCU



 SITRAIN © Siemens AG 2017

Steps

1. Press RECALL ('^' key) and MENU SELECT ("=" key) simultaneously.

2. If the HMI you require is already listed, move the cursor to select the corresponding line and click the "OK" softkey.

3. If no IP address with an HMI extension is displayed, select the "Select service session" menu. A scan for available sessions is performed. Move the cursor to select a line containing "HMI Running" at the end and confirm your selection with "OK".

Note

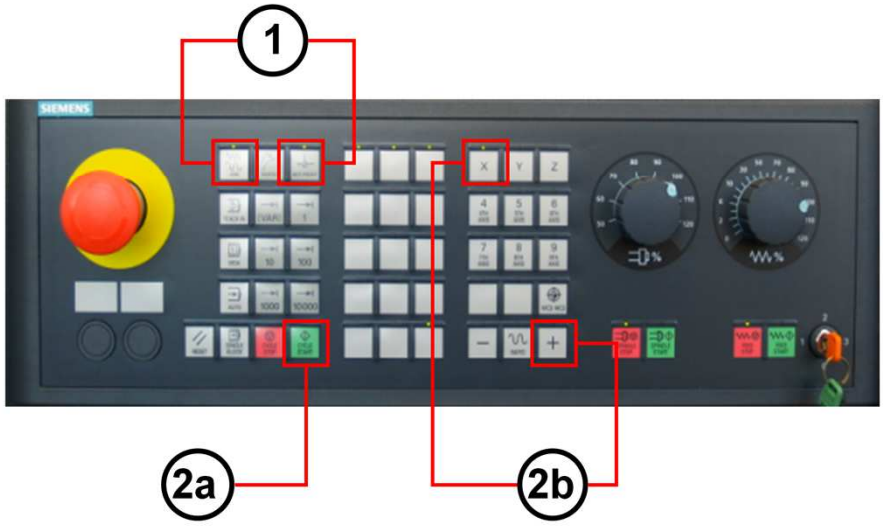
Meaning of displays:

192.168.214.241:0: IP address = **192.168.214.241**
Port = **0** (Standard Display)

The IP addresses of the HMI versions are as follows:

IP = 192.168.214.241 => SINUMERIK Operate of the PCU
IP = 192.168.214.1 => SINUMERIK Operate of the NCU

Reference point approach



Course NC-84SL-SIP Page 3.2 - 23 SITRAIN © Siemens AG 2017

Referencing the training rack

This section describes how to proceed to reference the training rack. Referencing is necessary to match the current axis position with the screen display.

1 Switch the training rack on. Activate "JOG" operating mode and the "REF" machine function (referencing).

The homing cam for the axes is simulated because, unlike on the machine tool, it does not exist as hardware on the training rack.

2a Now you can reference all axes with **NC Start** (channel-specific referencing)

or

2b **select** a single axis (X or Z) and only reference this **one axis** using the **direction key +** (single axis referencing).

Referenced

For each referenced axis, the machine screen displays the following symbol:

Note

Monitoring functions (e.g. working area limitation, protection areas, software limit switches, etc.) are only recognized by the control and taken into account after the axes have been referenced.

Exercise 1: Operator panel keyboard and screen display

Operator panel keys

Use the **area switchover key** 

select the different operating areas.

The **machine key** 

always selects the main machine screen.

If you press the **area switchover key** twice, you switch immediately from the main machine screen to the screen for the previous operating area.

Press the **channel switchover key** and check the selected channel in the screen display. 

Use the "Act. val. MCS / Act. val. WCS" softkey to switch between **machine coordinate system** and **workpiece coordinate system**.

Exercise 2: Machine operating area

Referencing axes

Approach the reference point with the X axis. The axis has no reference cam and is configured accordingly. Use the axis selector switch to select Axis 1.

Traversing axes/spindles (continuously) in JOG mode

JOG MODE 

Axis key 



...

Traversing direction 
or 
Rapid traverse override 

Traversing axes/spindles with fixed travel paths (incremental)





and traversing of 1000 µm

Axis key 



Traversing direction 


In the menu extension (">" key), you switch to the next menu area and select the "Settings" menu. There, you enter any value for the variable increment (e.g. 2540 -> 0.1 inch). Confirm with Enter and switch back to the main menu.

Select the user-specific increments and travel with the axis keys. 

Starting individual part program blocks with MDI mode



MDI mode

Input: G1 X100 Y100 F1000



Program start



Reset and "Delete blocks" softkey

Input: S1000 M3

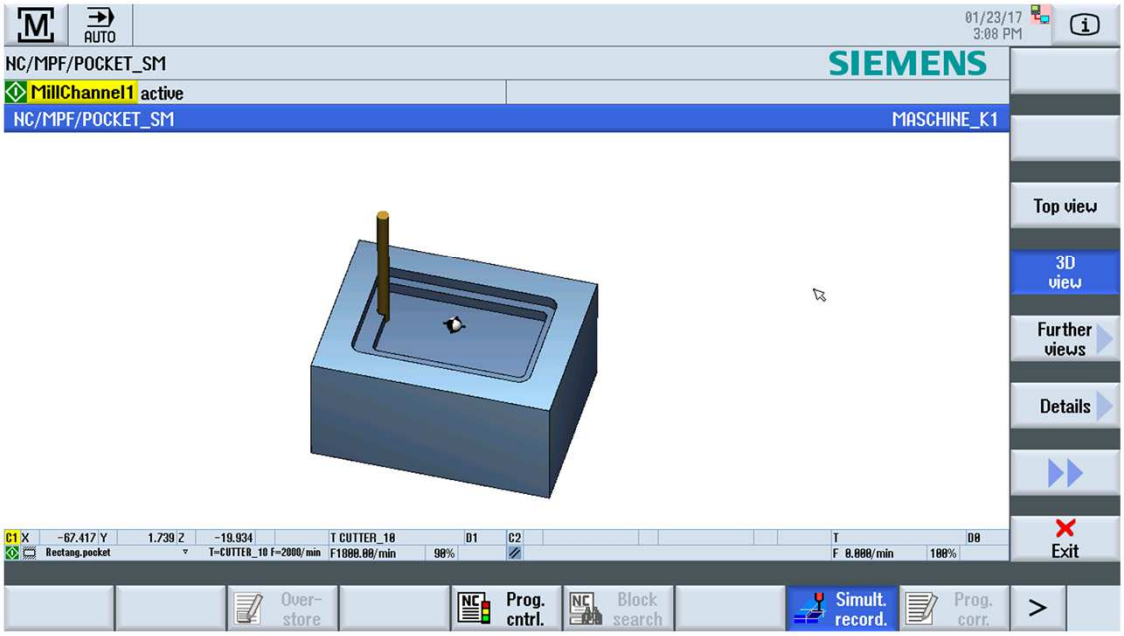


Program start



The spindle can be stopped by pressing Reset.

Exercise 3:
Rectangular pocket with ShopMill



 SITRAIN © Siemens AG 2017

Objective

We are going to use ShopMill to create a simple program (milling a rectangular pocket).

Data

The following data is known:

• Tool

The tool is to be an end mill with the following tool data:
length = 100 mm, radius = 10 mm, number of teeth: 3.

All of the following dimensions are in mm.

• Blank

The blank is a block, defined by the two corner points:

(X0,Y0) = (0,0) and (X1,Y1) = (200,150)

The following are defined in Z: Initial dimension ZA = 0 mm, finishing dimension ZI = -50 mm

• Spindle

The tool spindle is to rotate at 1,000 rpm (clockwise).

• Pocket

The rectangular pocket data is as follows:

- Reference point (X0,Y0) = (100, 75)
- Extent in (X,Y): (length, width) = (L, W) = (150, 100)
- Reference point Z0 = 0 mm
- Depth Z1 = 30 mm
- Corner radius: R = 20 mm
- Infeed in the X-Y plane: DXY = 16 mm, DZ = 10 mm
- Feedrate in the X-Y plane: F = 5,000 mm/min;
  in Z: FZ = 0.3 mm/tooth

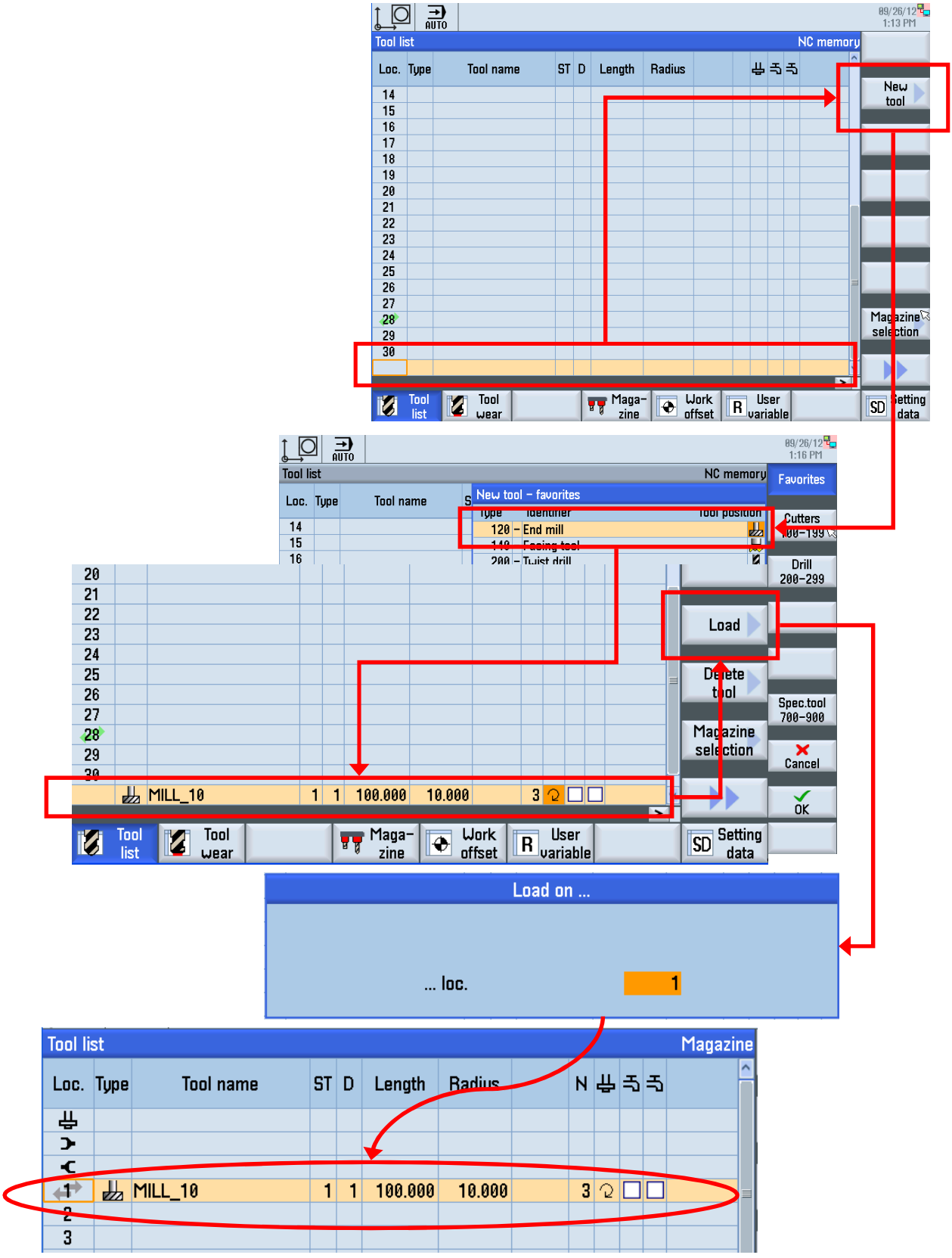
Steps

1. Create tool
2. Create program with ShopMill
3. Define blank in program header
4. Create rectangular pocket
5. Tool selection
6. Parameterized rectangular pocket
7. Test

Sequence

1. Create and load tool

In the Parameters operating area, create an end mill with the specified dimensions and load it in the tool magazine.

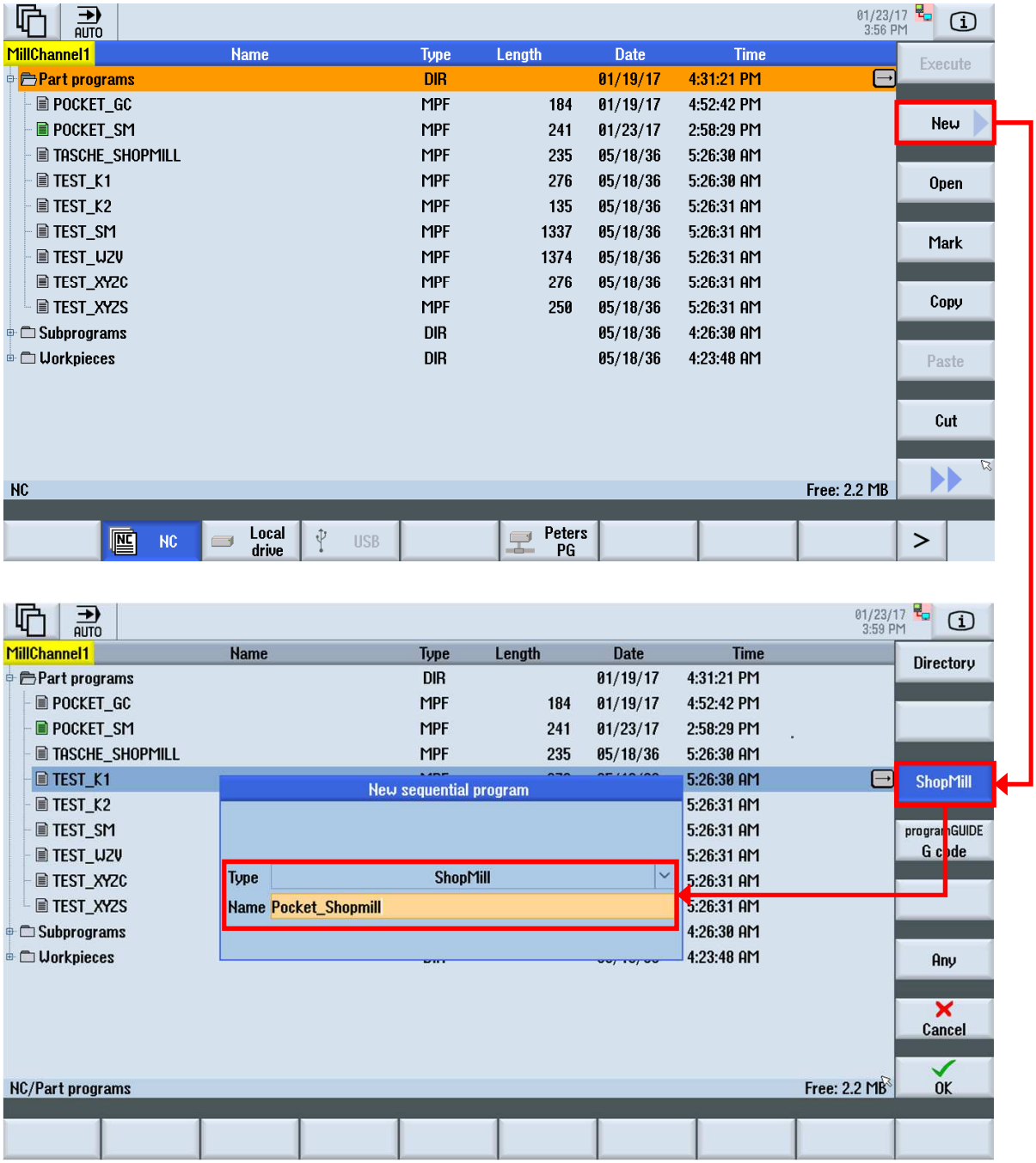


2. Create program with ShopMill

Create a new program with ShopMill:

-> OA: Program -> softkey: New -> softkey: ShopMill

-> Give the program a name, e.g.: **POCKET_ShopMill**



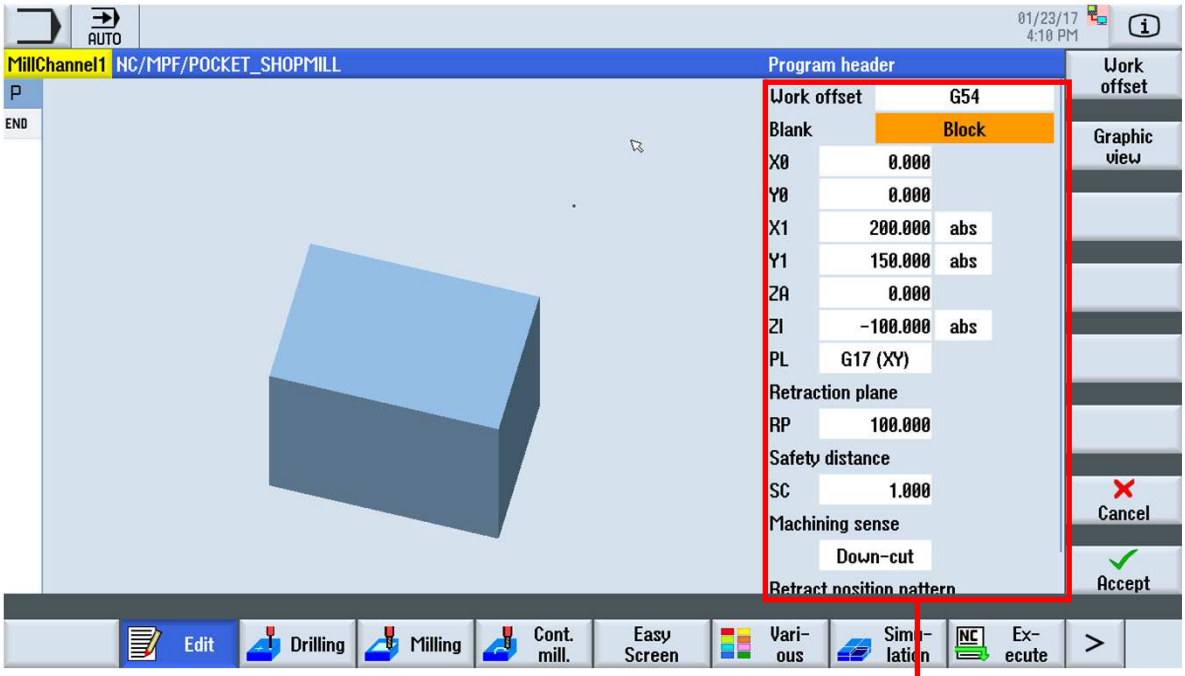
3. Define blank in program header

Start by defining the blank with the specified dimensions:

Enter the dimensions (see figure)

-> softkey: Accept

The blank data is written to the program header.

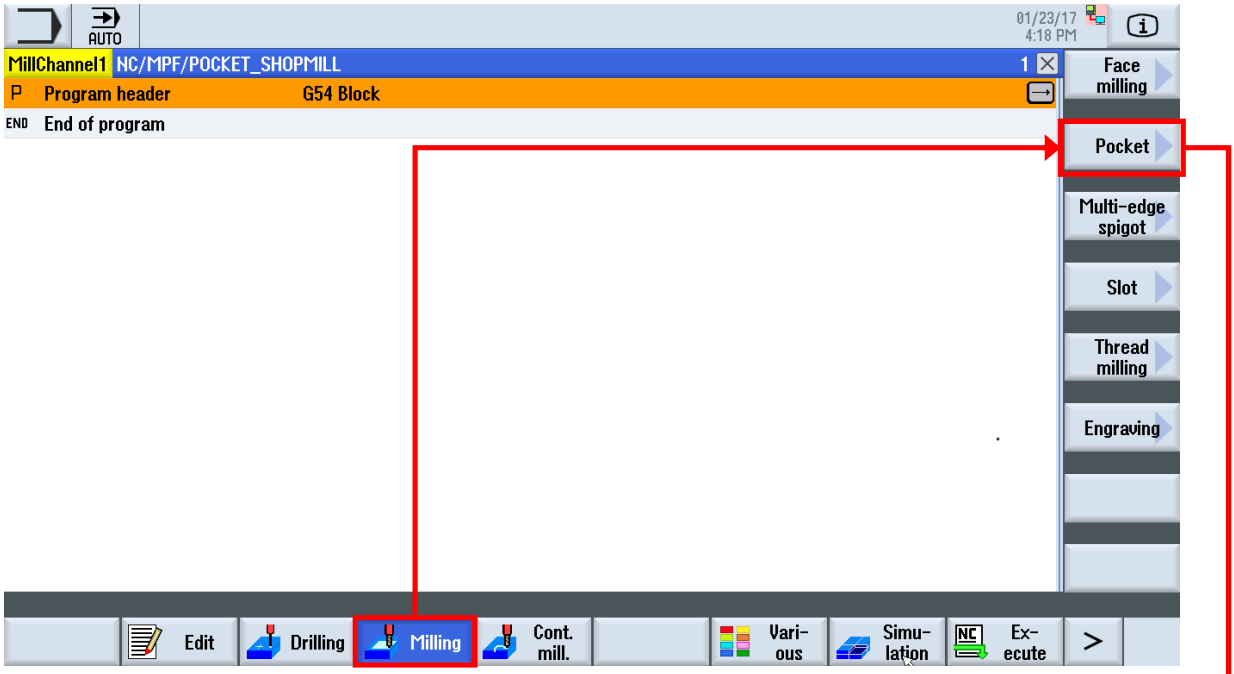


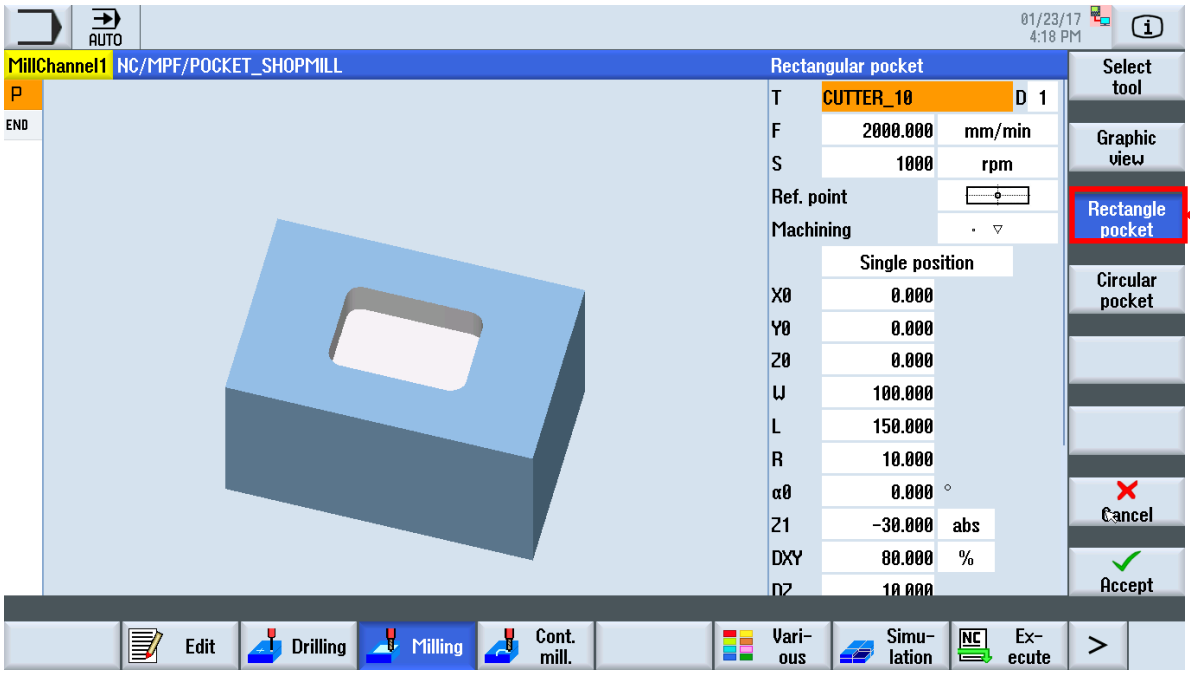


4. Create rectangular pocket

Create a rectangular pocket:

Softkey: Milling -> softkey: Pocket -> softkey: Rectangular pocket



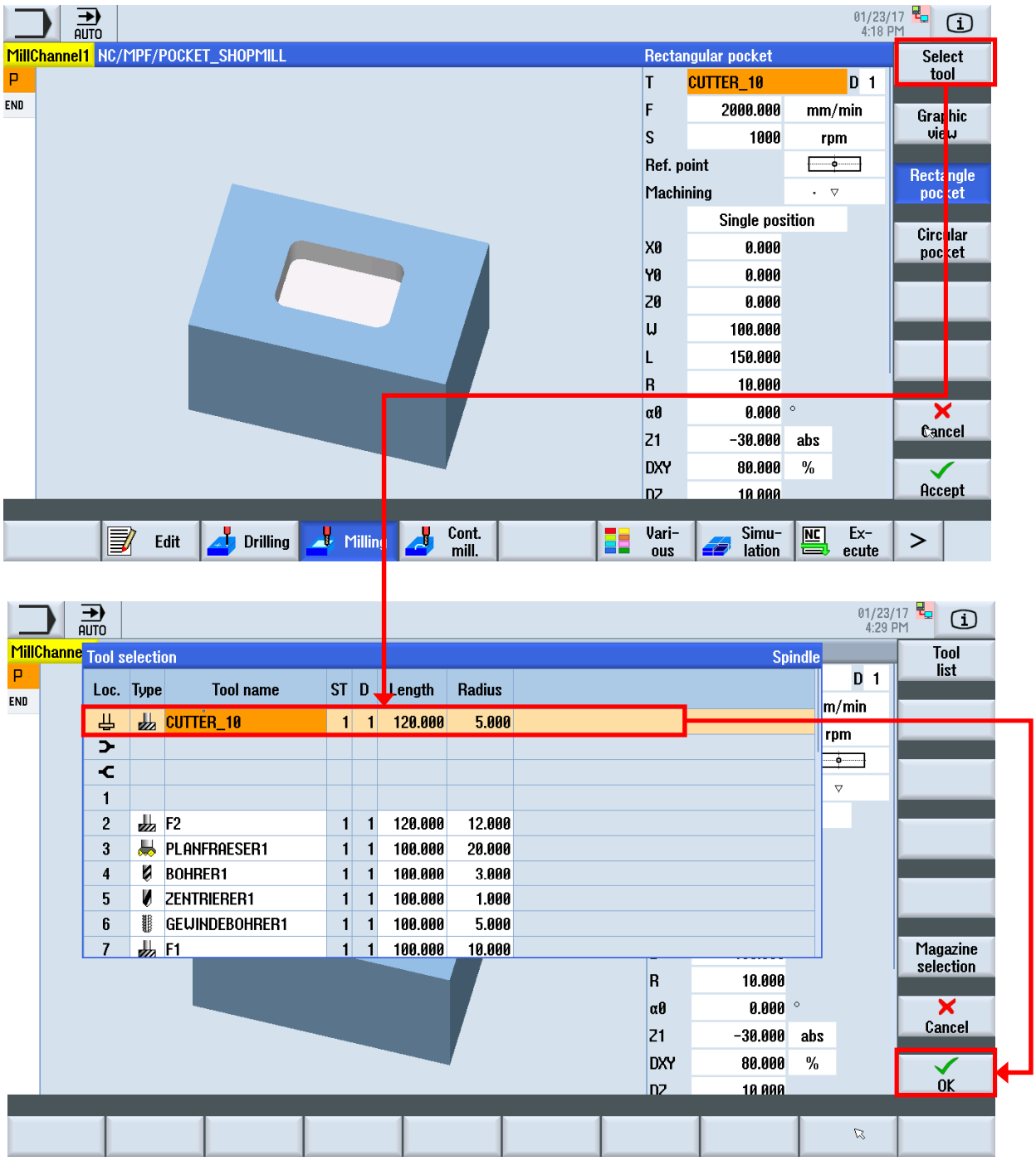


5. Tool selection

In ShopMill, the tool is selected in the parameterization display:

-> softkey: Select tool

-> Select the tool from the list **-> softkey: OK**



6. Parameterize the rectangular pocket

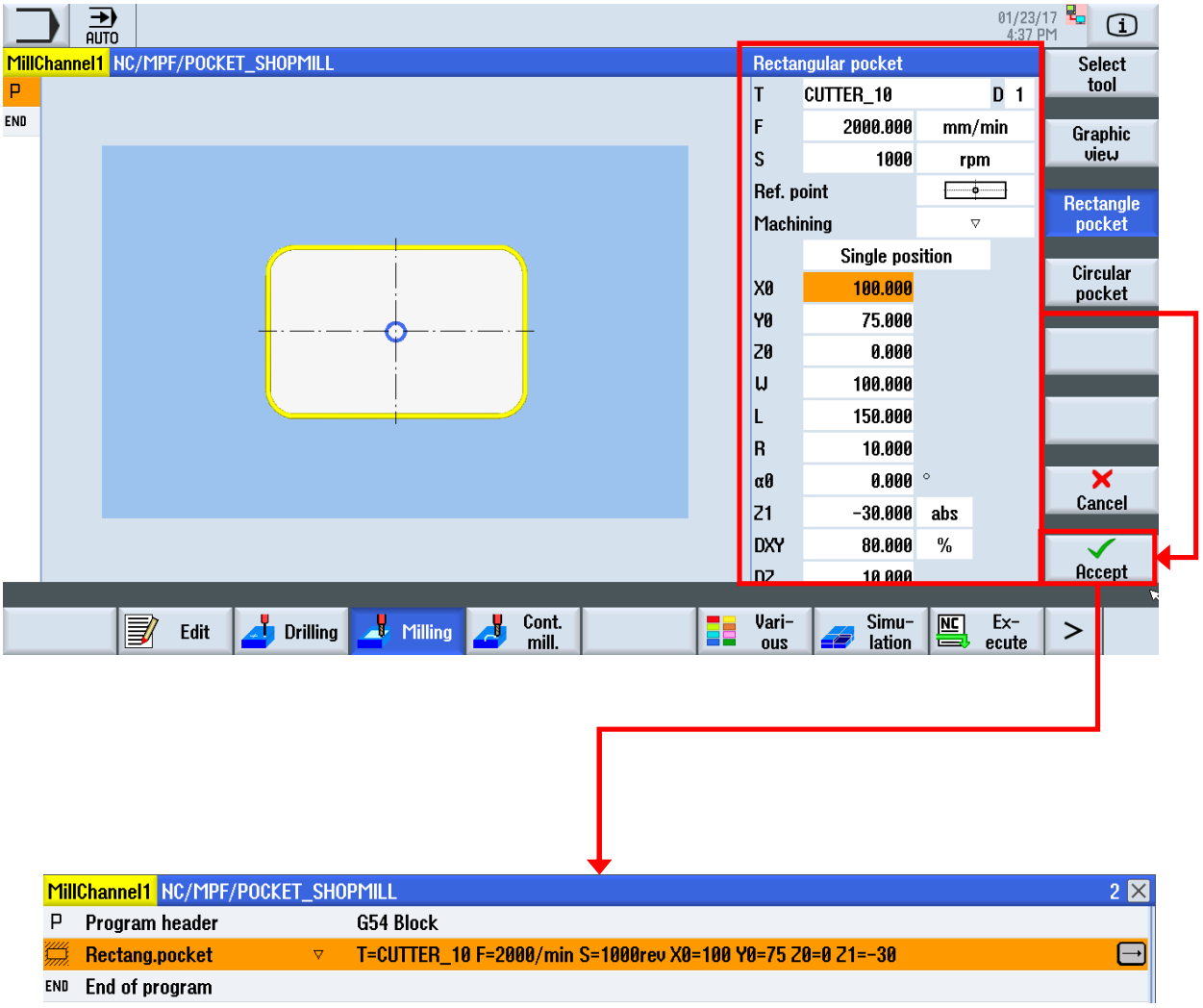
Now enter the remaining values in the parameterization display as shown below.

Confirm the settings with **-> softkey: Accept**

The resulting program code in the ShopMill display can be seen below.

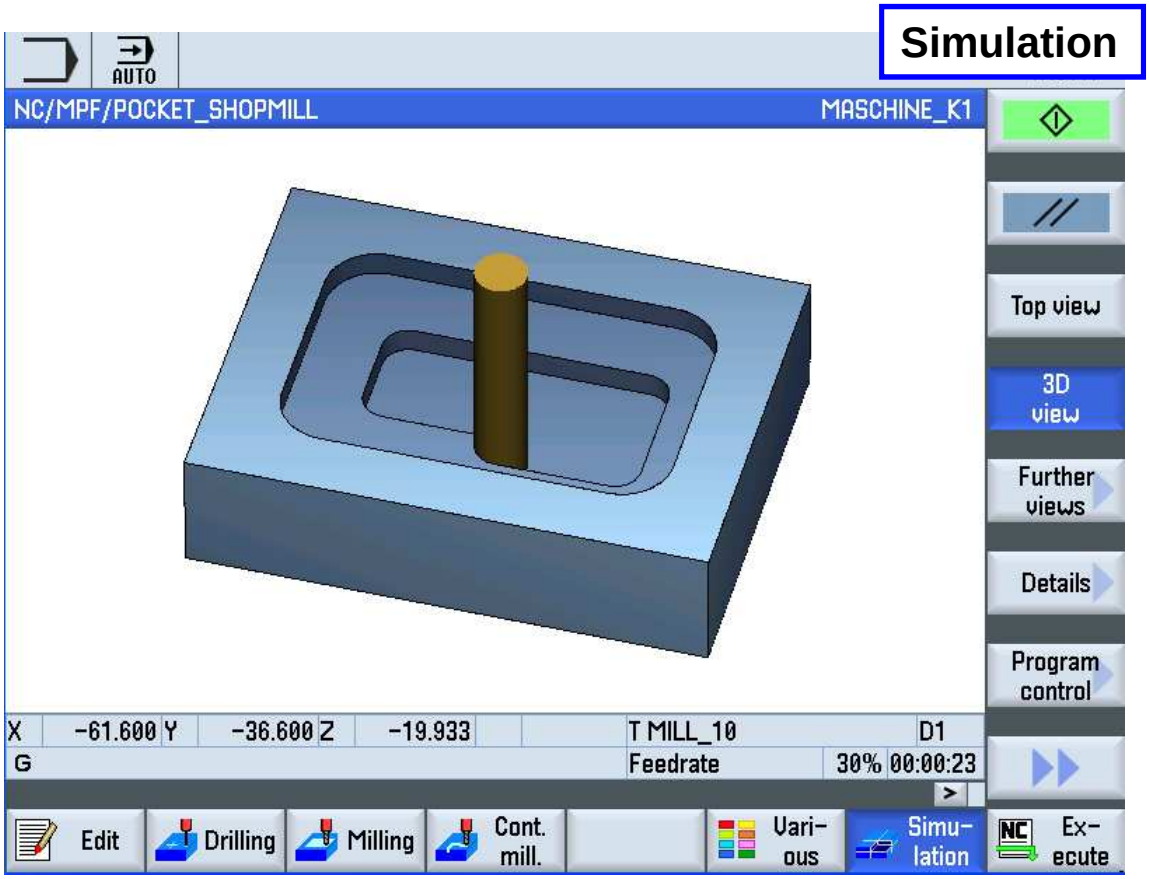
• **Note**

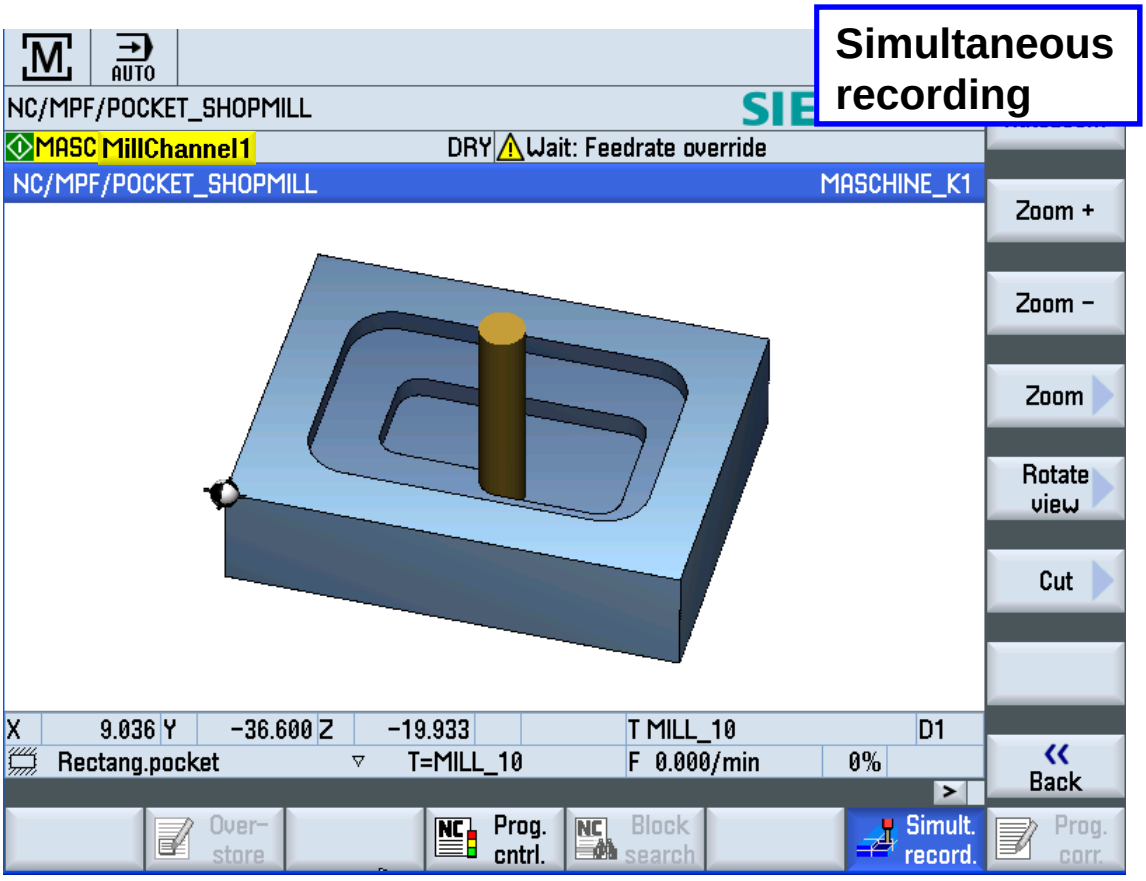
In ShopMill (unlike in programGUIDE), additional technological data such as the tool selection and spindle speed must be entered during cycle parameterization.



7. Test

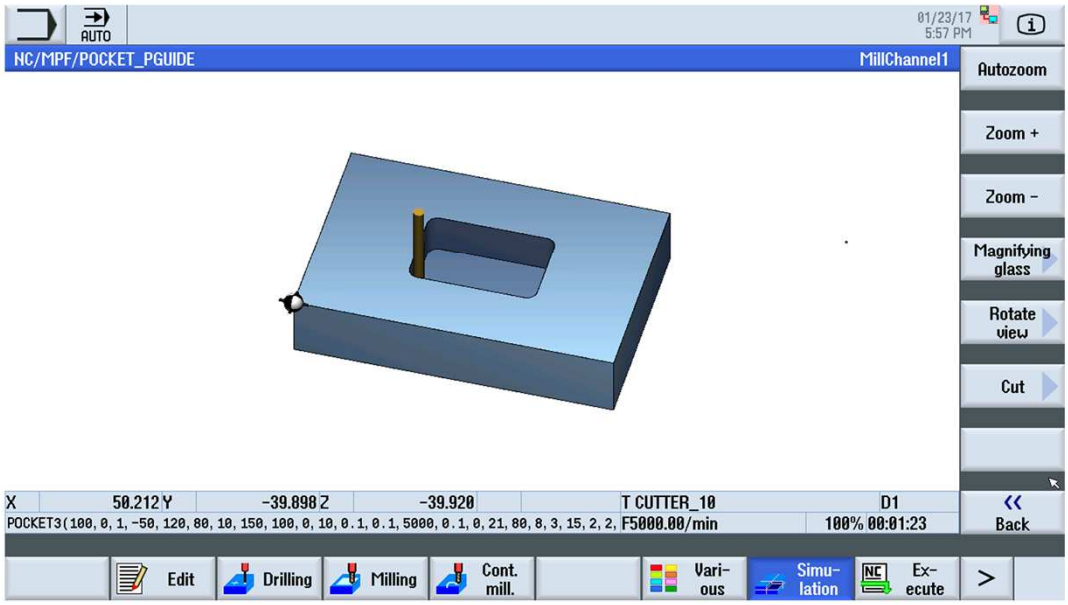
Simulation or **simultaneous recording** can be used to test the program.
Results:





Additional exercise:
Rectangular pocket with programGUIDE

SIEMENS



 SITRAIN © Siemens AG 2017

Target

We are going to use programGUIDE to create a simple program (milling a rectangular pocket).

Data

The following data is known:

• Tool

The tool is the end mill we have already created with the following tool data: length = 100 mm, radius = 10 mm, number of teeth: 3.

All of the following dimensions are in mm.

• Blank

The blank is a block, defined by the two corner points:

(X0,Y0) = (0,0) and (X1,Y1) = (200,150)

The following are defined in Z: Initial dimension ZA = 0 mm, finishing dimension ZI = -50 mm

• Spindle

The tool spindle is to rotate at 1,000 rpm (clockwise).

• Pocket

The rectangular pocket data is as follows:

- Reference point (X0,Y0) = (100, 75)
- Extent in (X,Y): (length, width) = (L, W) = (150, 100)
- Reference point Z0 = 0 mm
- Depth Z1 = 30 mm
- Corner radius: R = 20 mm
- Infeed in the X-Y plane: 16 mm, in Z0 = 10 mm
- Feedrate in the X-Y plane: 5,000 mm/min; in Z: 1,000 mm/min (~0.3 mm/tooth)

Steps

1. Create program with programGUIDE
2. Define blank
3. Tool selection; M6 for tool change
4. Program spindle speed
5. Create and parameterize the rectangular pocket
6. End of program and test

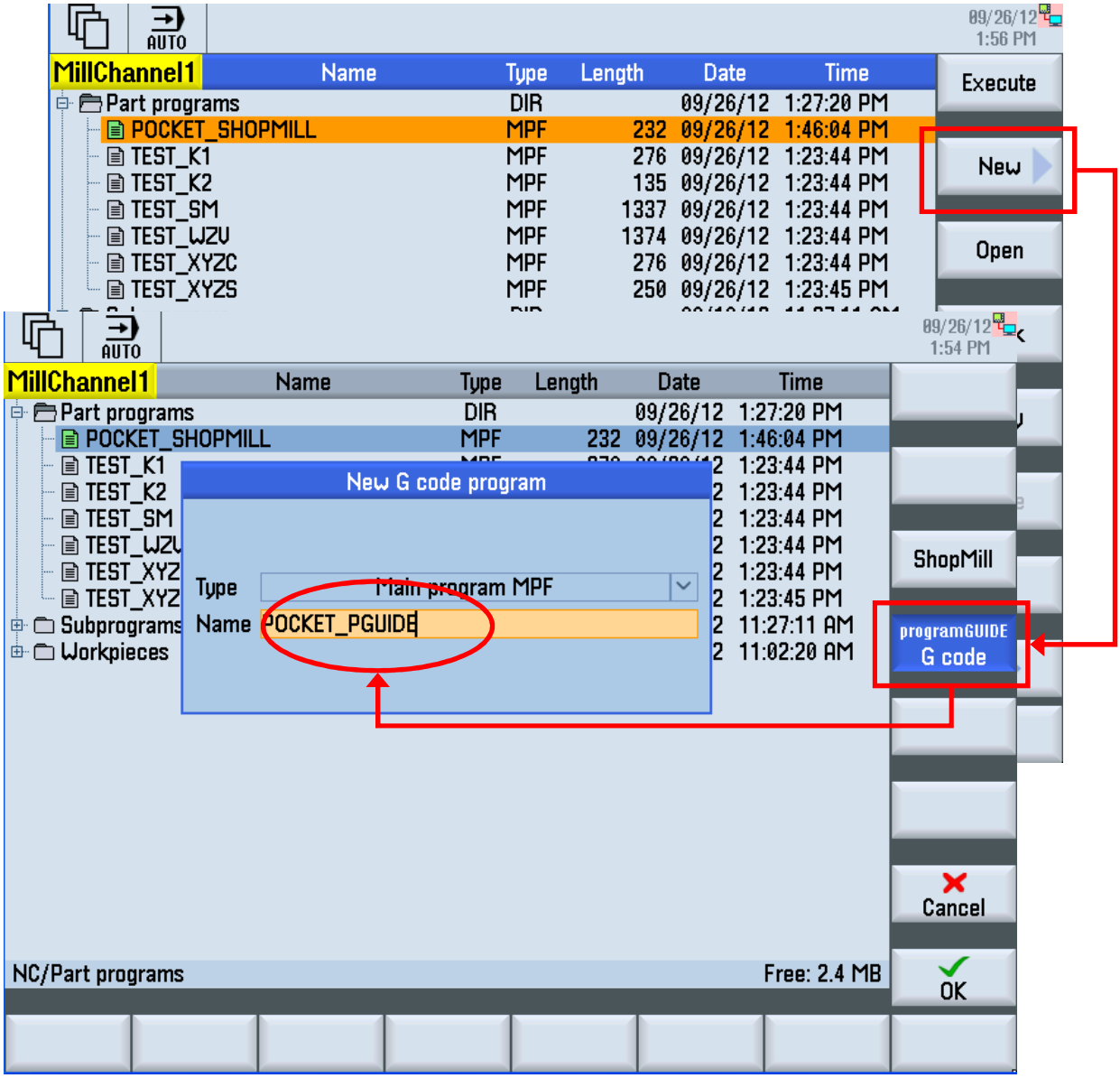
Sequence

1. Create program with programGUIDE

Create a new program with programGUIDE:

-> OA: Program -> softkey: New -> softkey: programGUIDE G code

-> Give the program a name, e.g.: **POCKET_PGUIDE**



2. Define blank

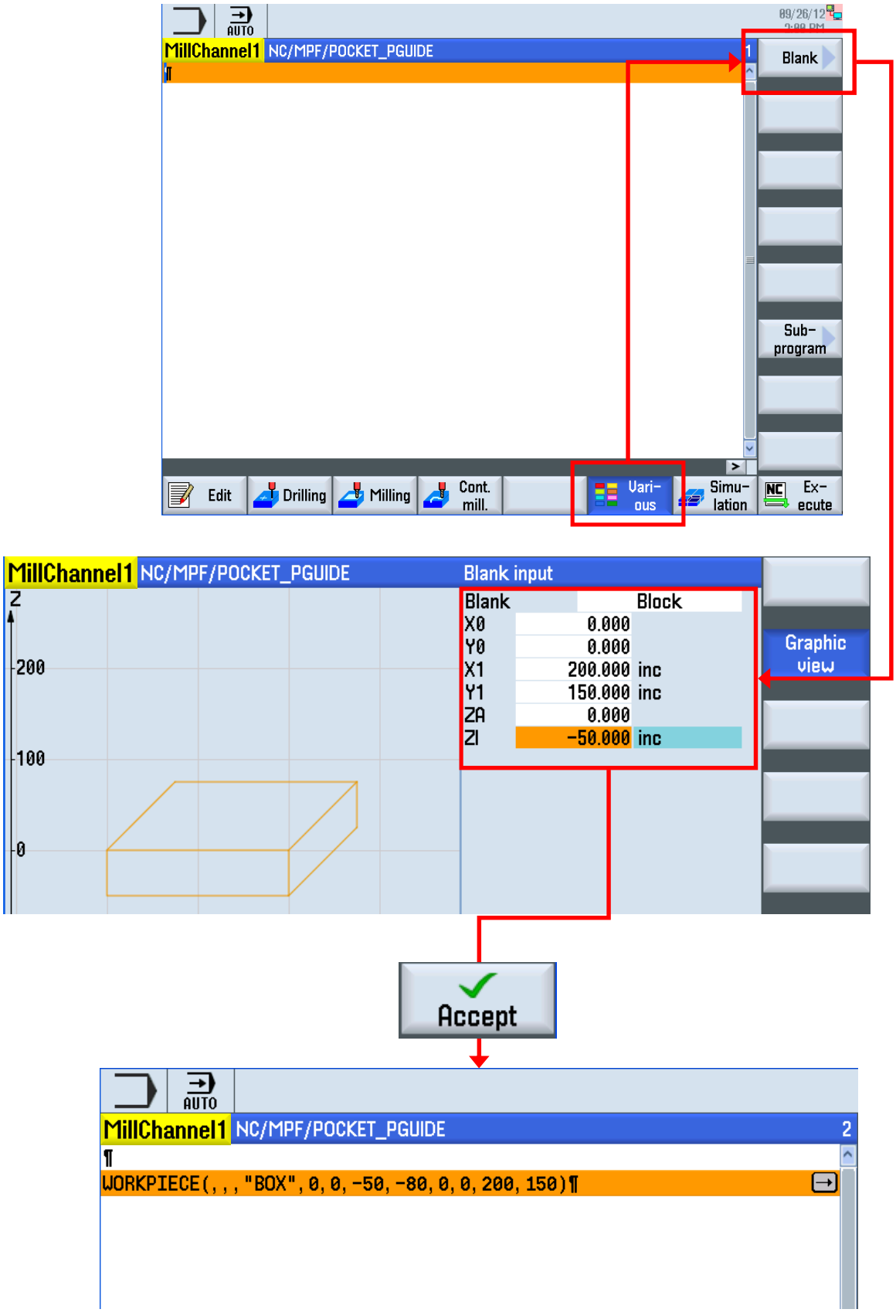
Define a blank with the specified dimensions:

-> softkey: Various -> softkey: Blank

Enter the dimensions (see figure)

-> softkey: Accept

The result is a cycle call "WORKPIECE(....)" as shown below.



3. Tool selection; M6 for tool change

To switch back to edit mode at this point, press:

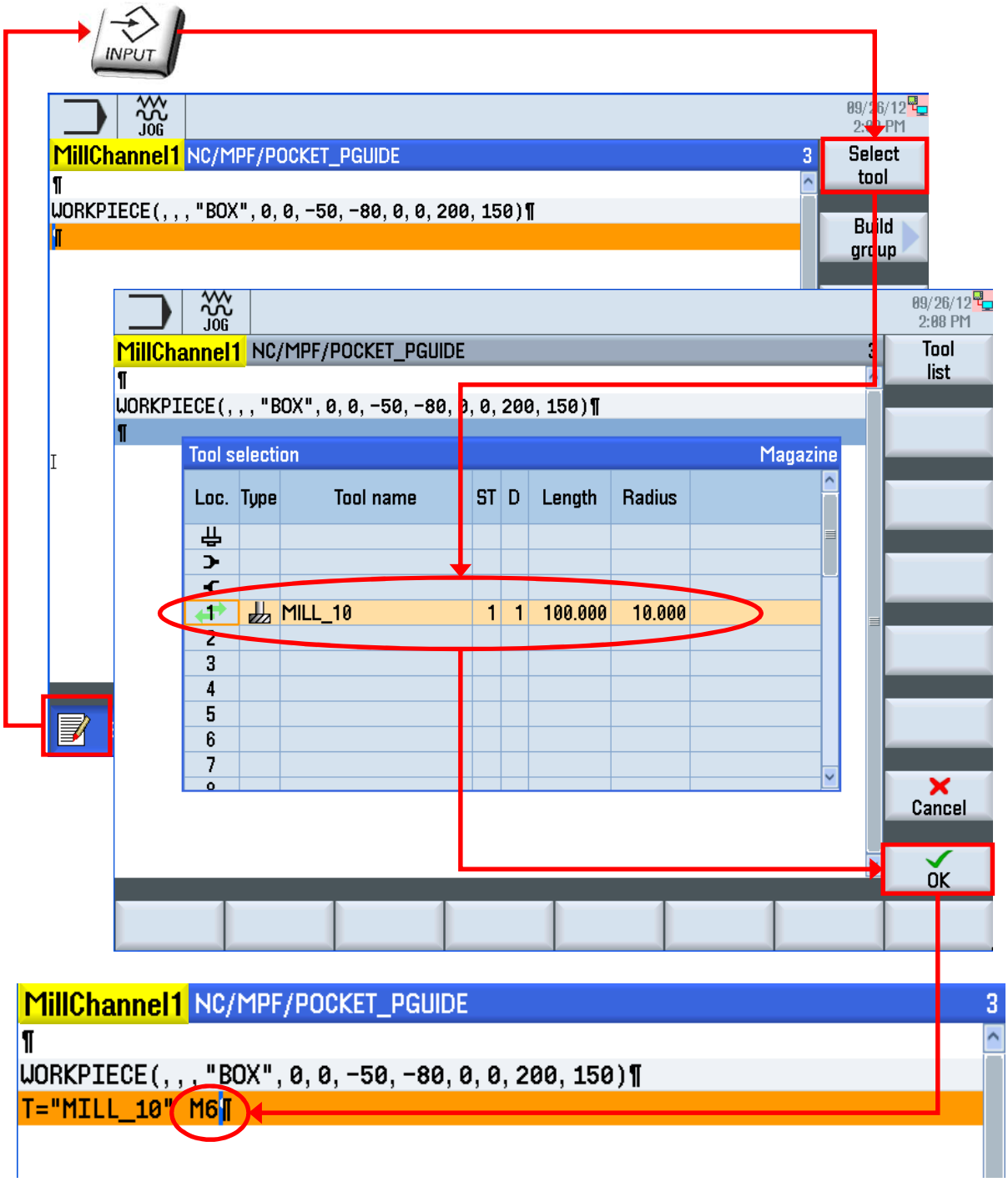
-> softkey: Edit -> INPUT key 

Finally, select the milling cutter created previously with:

-> softkey: Select tool -> softkey: OK

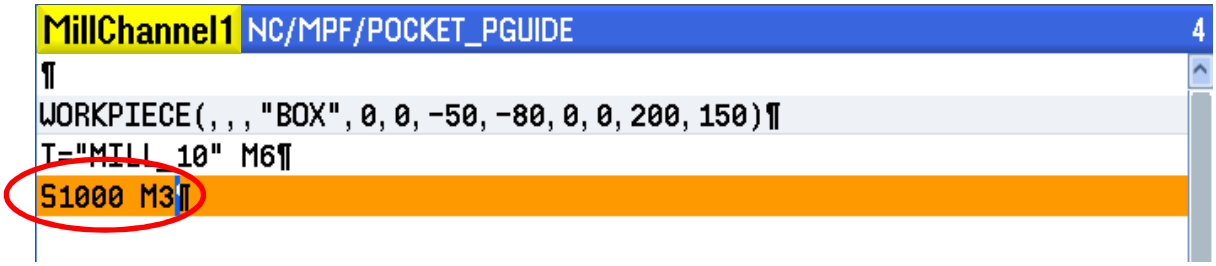
Add **M6** to the command sent **T="FRAESER_10"**

to switch to the correct tool for the next machining operation.



4. Program spindle speed

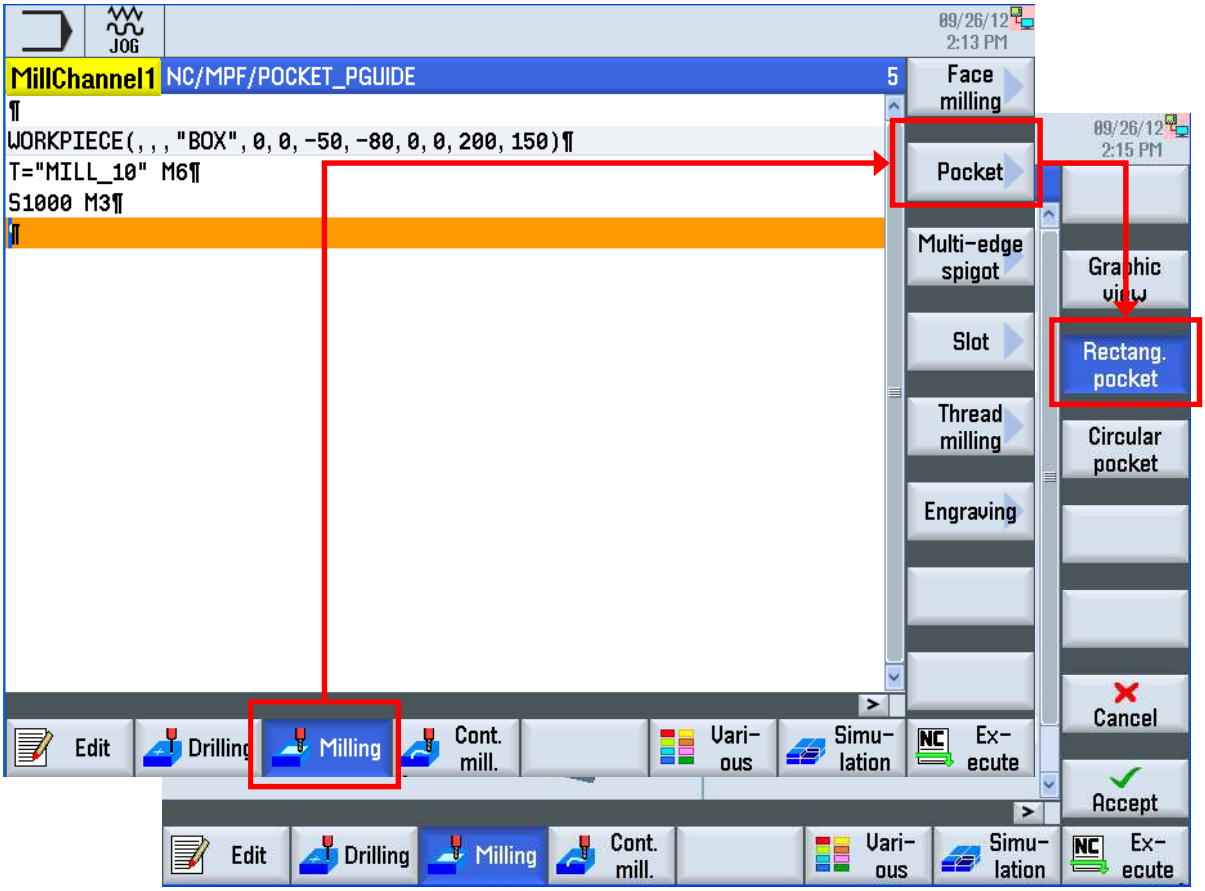
Program the required spindle speed



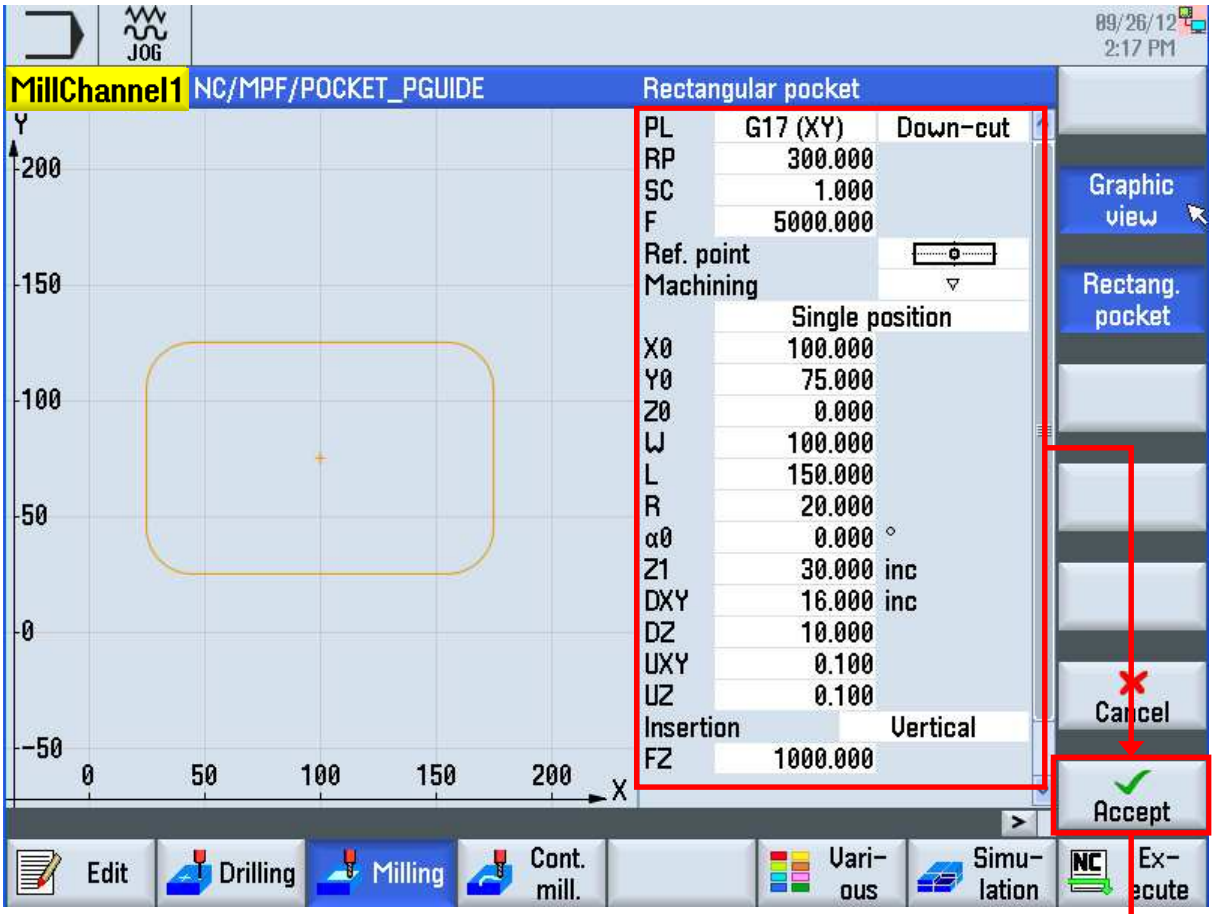
5. Create and parameterize the rectangular pocket

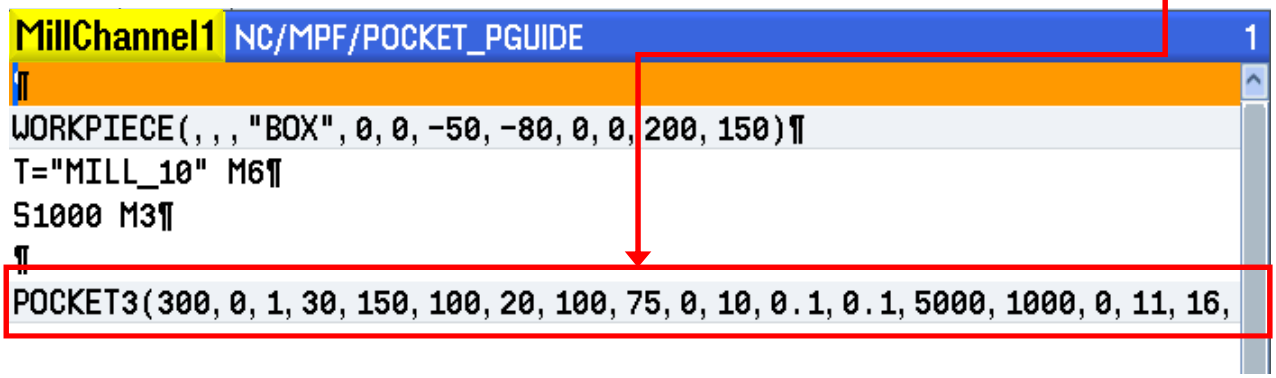
Next, create the rectangular pocket:

-> softkey: Milling-> Pocket -> softkey: Rectangular pocket



Parameterize the rectangular pocket according to the specifications.
This produces the cycle call **POCKET3(...)** as shown below.





6. End of program and test

Complete the program with M02.

Simulation or **simultaneous recording** can be used to test the program.

• Simulation: Start from the program editor with **-> softkey**: **Simulation**

The following machine data should be supplied at this point:

MD 24030 = 1

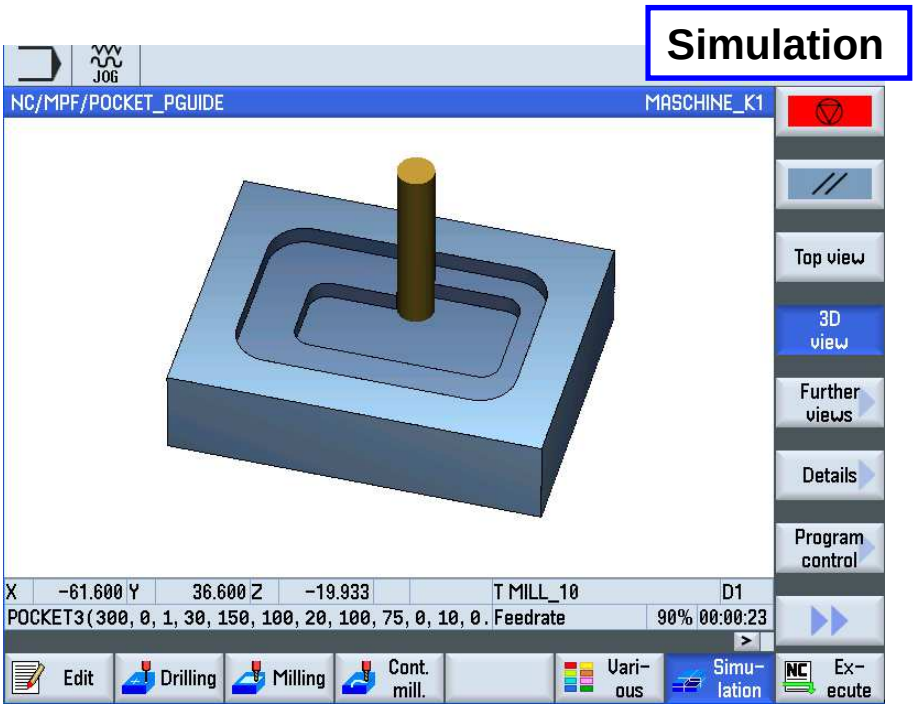
MD 53230 > 0 (X, Y, Z) Start position for simulation

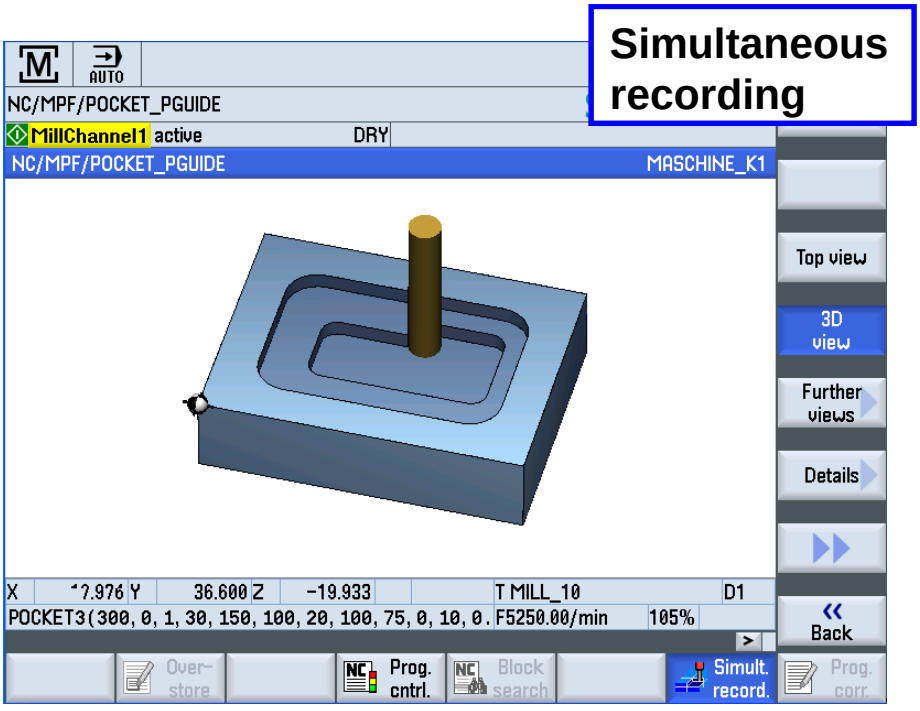
• Simultaneous recording: Select program:

-> OA: Program -> softkey: Selection

Start from **Machine** operating area, **AUTOMATIC** mode

via **-> softkey: Simultaneous recording**





SIEMENS



Chapter 4

Networks and tools

Course NC-84SL-SIP

SITRAIN © Siemens AG 2017

Content **Page**

Learning objectives 3
Motivation: 4
Network hardware: 5
Networking: Protocols 6
Start-up tools for 840D sl: Networks 7
Network settings with 840D sl 8
Networking with TCP/IP v4 9
DHCP Dynamic Host Configuration Protocol 10
Default IP addresses with 840D sl 12
Selected commands on the command line 13
Linux service shell login 14
Default users and passwords with 840D sl 15
Finding the IP Address of the target station 16
Setting IP and subnet mask 17
Alternate IP addressing? □ Fixed IP is better! 19
Setting the computer name 20
PG/PCU: Creating Windows user and group assignment 21
PG/PCU: Setting-up Windows file share (1) 22
Setting-up Windows file share (2) 23
PG/PCU: Password policies 24
PCU Service Center: Network settings for Ghost 25
Step 7: Online with TCP/IP 26
SINUMERIK Operate on PCU: SK: "HMI" – “NCU link” 27
HMI on PG/PC: SINUMERIK CP 28
BOOTP (Bootstrap Protocol) 29

Content **Page**

VNC: Virtual Network Computing 30
840D sl: Multiple servers (BOOTP, DHCP, VNC) 31
Using VNC Scanner and VNC Viewer on the PG 32
Important service commands (examples) 33
Bus TCP/IP diagnostics in operating Area diagnostics 34
TCP/IP diagnostics 35
TCP/IP configuration in operating area diagnosis 36
TCP/IP configuration in operating area diagnostics 37
Network settings in operating area startup 38
System Network settings 39
Company network settings 40
OPC UA server settings 41
Operator panel configuration 42
VNC – configure connections 43
T:M:N Configuration 44
WinSCP overview of functions 45
Access MyMachine /P2P overview of functions 46
Networks and tools 47
Exercise: 48
Exercise: Check TCP/IP setting for PCU 51
Exercise: Set the default VNC server for an NCU+PCU+TCU configuration 52

Learning objectives

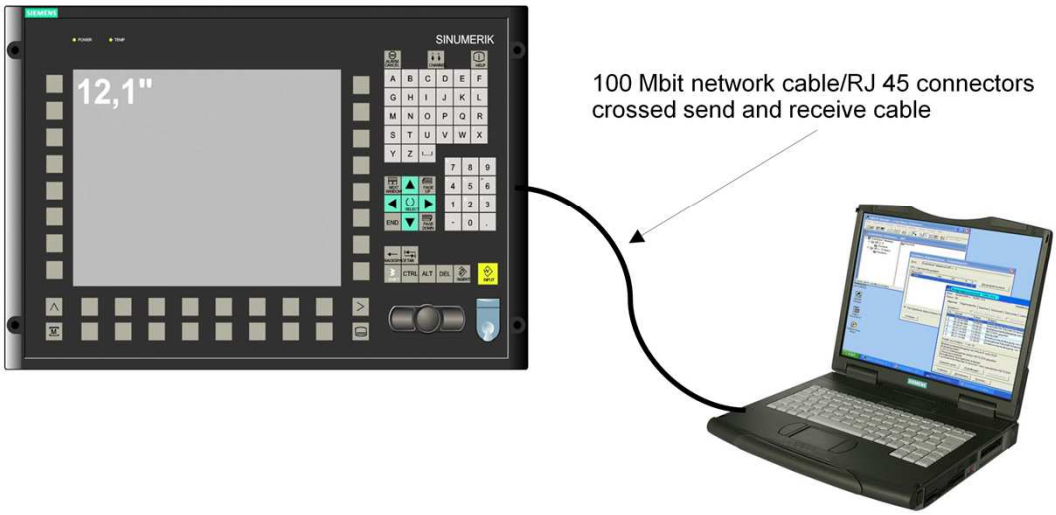
- You obtain knowledge of networks
- You have knowledge which protocols are important for SINUMERIK
- You are able to set up a network connection
- You know about the network settings for NCU and PCU



SITRAIN © Siemens AG 2017

Notes

Motivation:
Software installation/Data backup/Online access



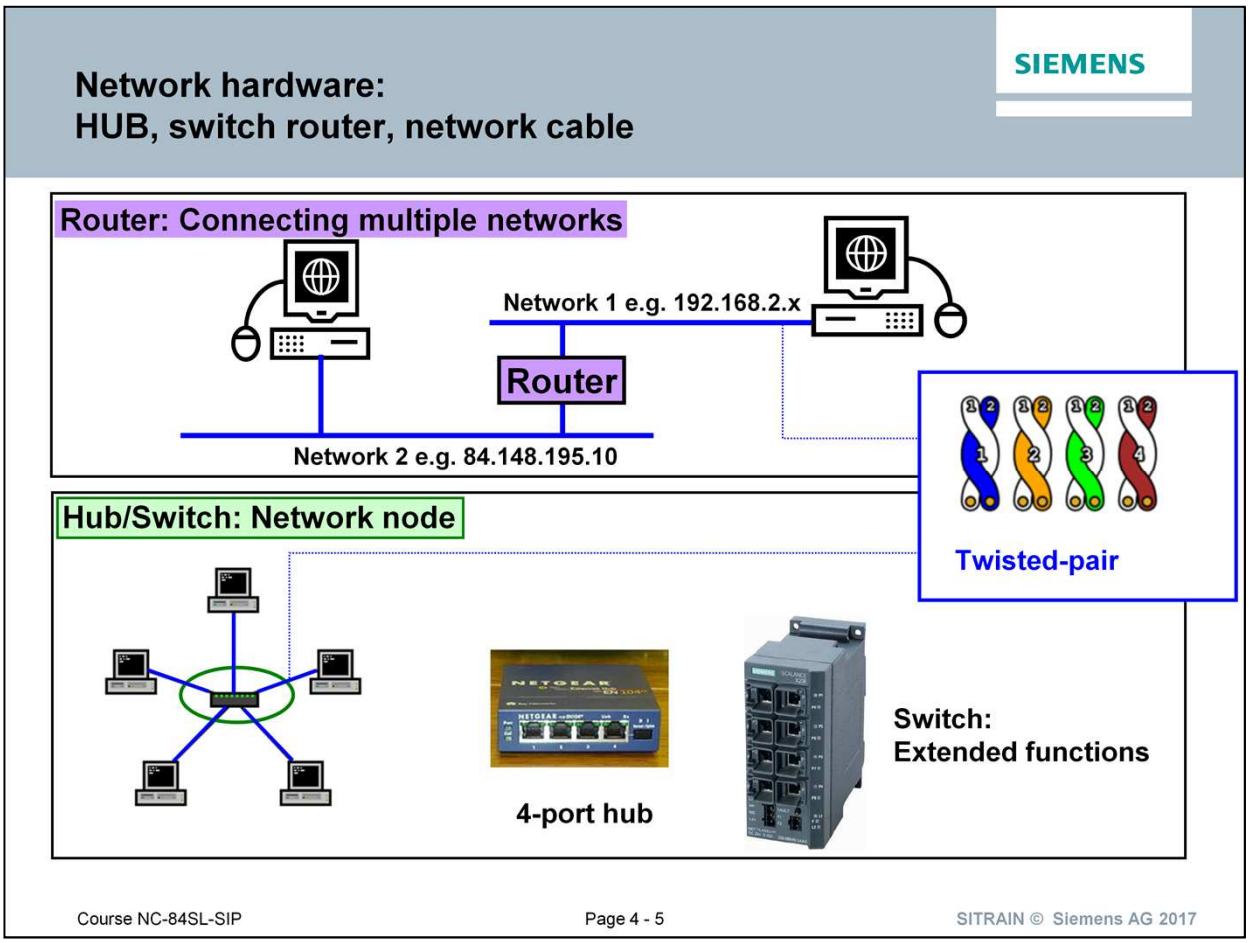
Course NC-84SL-SIP Page 4 - 4 SITRAIN © Siemens AG 2017

Modern control computers allow the user to run software packages in addition to the operating system and operator interface. The quantity of data and software can easily amount to several gigabytes. Realistically this quantity of data can only be transferred via a network. With a parallel or even external serial connection, data transfer would take several hours.

These lengthy periods of time would occur in particular when performing software installation or data backups. Therefore, with modern systems, only network interface cards (Ethernet) represent a useful solution for data transmission. Alternate high-speed interfaces such as firewire or USB2.0 are not intended for data backup or installation with SINUMERIK systems.

With the 840D sl control generation, the online connection for Step7, for example, is implemented via Ethernet. Other connection options are no longer being prioritized. This technology accelerates access to the PLC and data update operations.

It is, therefore, necessary to have fundamental knowledge of the network technology.



Hub

A **hub** is used in telecommunications to connect network segments or individual hubs e.g. via Ethernet.

A hub only has one connection type, called a port. Hubs are also referred to as multiport repeaters or repeating hubs. The hub passes the signal from one network node to all other network nodes.

Switch

A **switch** is an electronic device for connecting several computers or network segments in a local network (LAN); similar to a hub. Switches are also referred to as intelligent hubs.

Ethernet switch is the short form of switching hub and indicates the ability to operate switched Ethernet.

- Advantages

If two network nodes are sending simultaneously, there is no data collision as the switch can send multiple packets simultaneously via the backplane.

Router

A **router** is a switching device in a network that directs information arriving on an incoming protocol to the correct destination network or subnet.

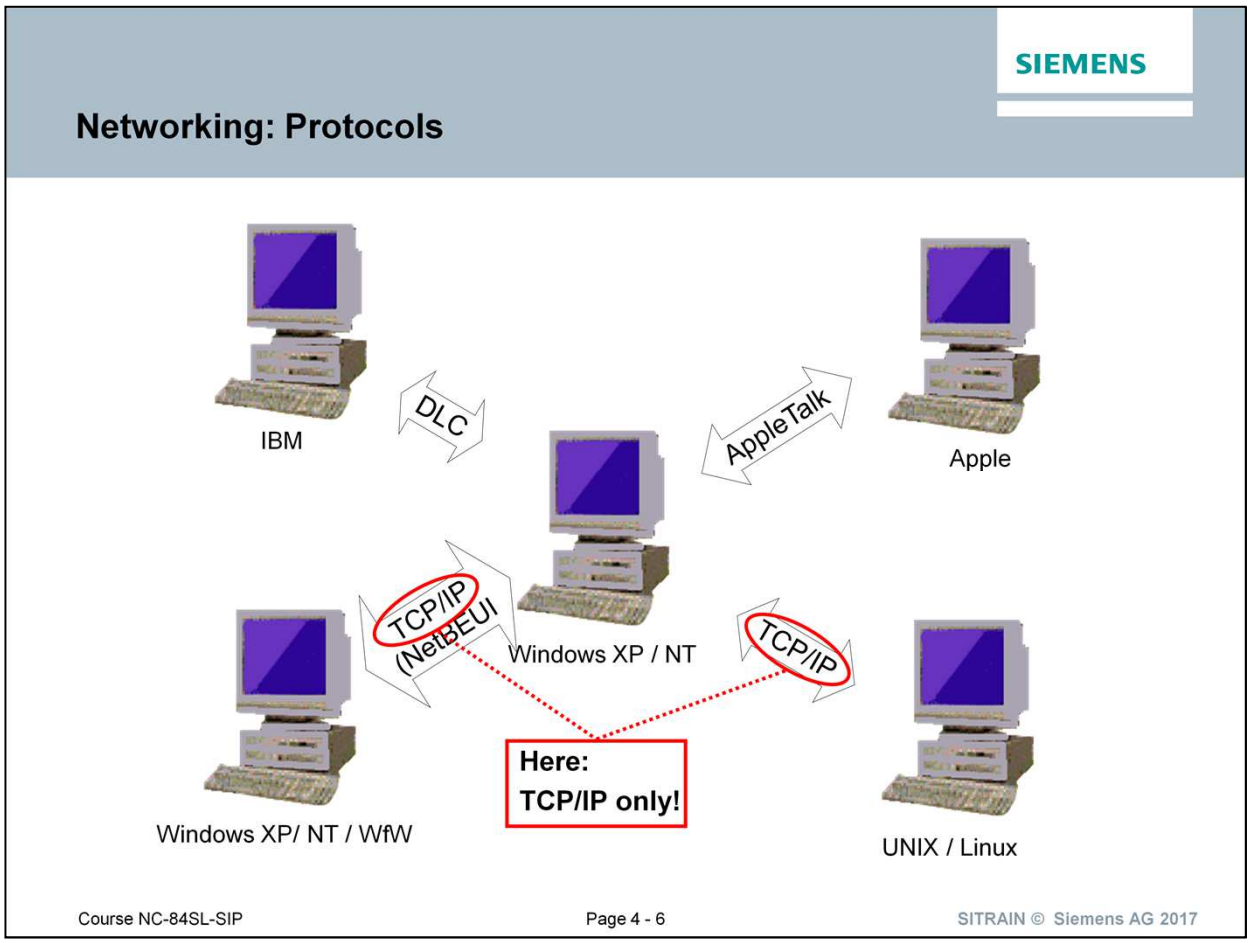
Network cables

- Twisted-pair cables

Twisted-pair cables are cables consisting of independently insulated wires that are twisted around each other. Interference and crosstalk is significantly reduced during data transmission by twisting the signal wire and return wire in a current loop (the wire pair). Twisted-pair cables come in two-pair and four-pair versions. Recent network installations mainly use four-pair cables.

- Crossover cables

Crossover cables are twisted-pair cables with 8 conductors. Each end is fitted with an RJ45 connector, but the pairs are swapped in *one* connector. They are used, for example, to directly connect two personal computers.



Windows supports various network protocols. Which network protocol you select depends on the requirements of the current network configuration. While the concurrent use of multiple network protocols allows more flexibility on the one hand, on the other hand loading multiple drivers will use more CPU capacity. Uninstalling protocols that are not required is quick and easy.
The TCP/IP network protocol is used as standard for communication with the 840Di/DiSL/Dsl.

NetBEUI

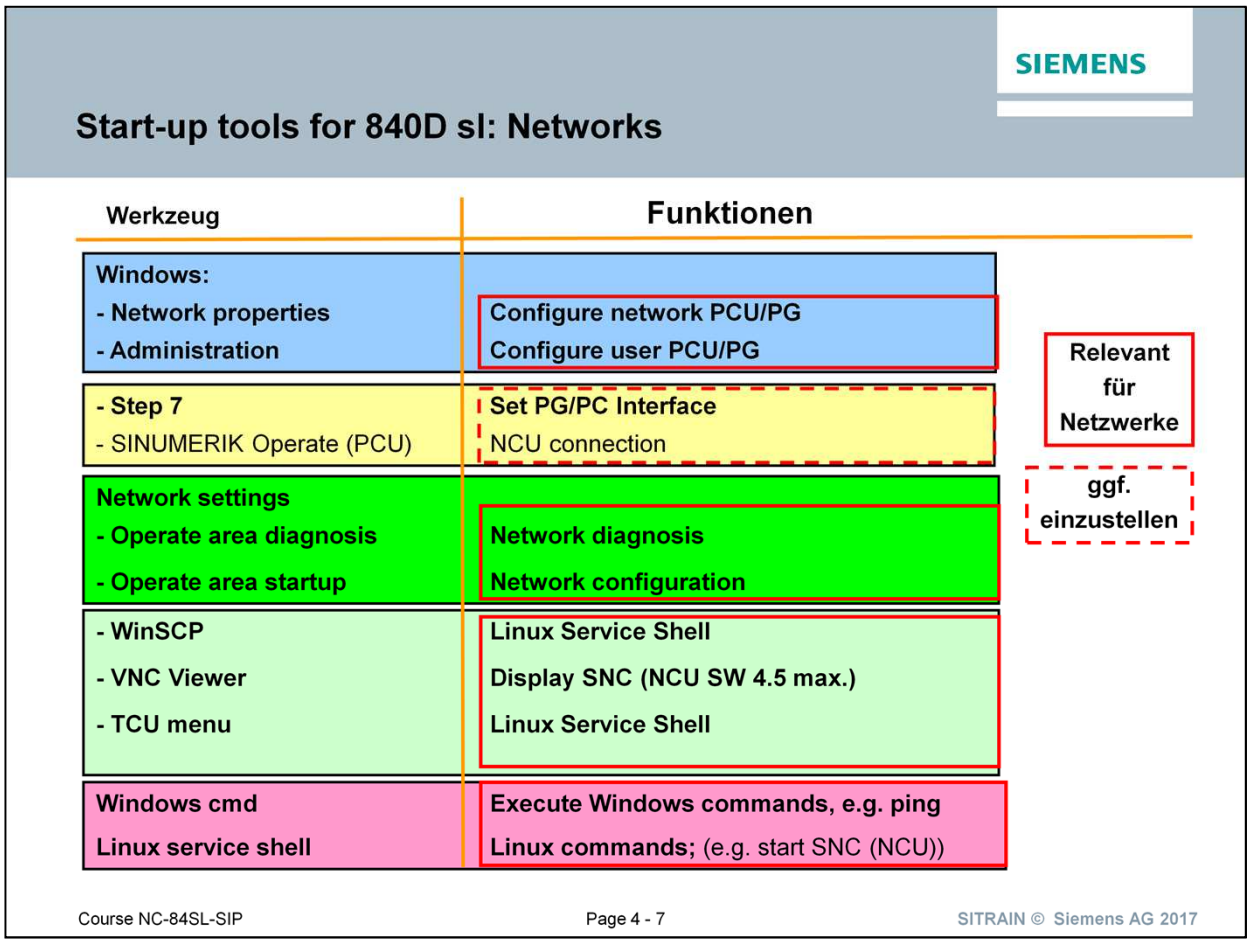
NetBIOS **E**xtended **U**ser **I**nterface. The NetBEUI protocol was developed by IBM. Although its performance is impressive (the size of the header protocol is just
14 bytes) but cannot be routed. It is suitable for smaller networks with fewer than 10 workstations on which all computers are located in the same segment. NetBEUI was the standard protocol for Windows for Workgroups, but the data volume generated in larger networks means that it is now no longer as popular as it once was.

TCP/IP

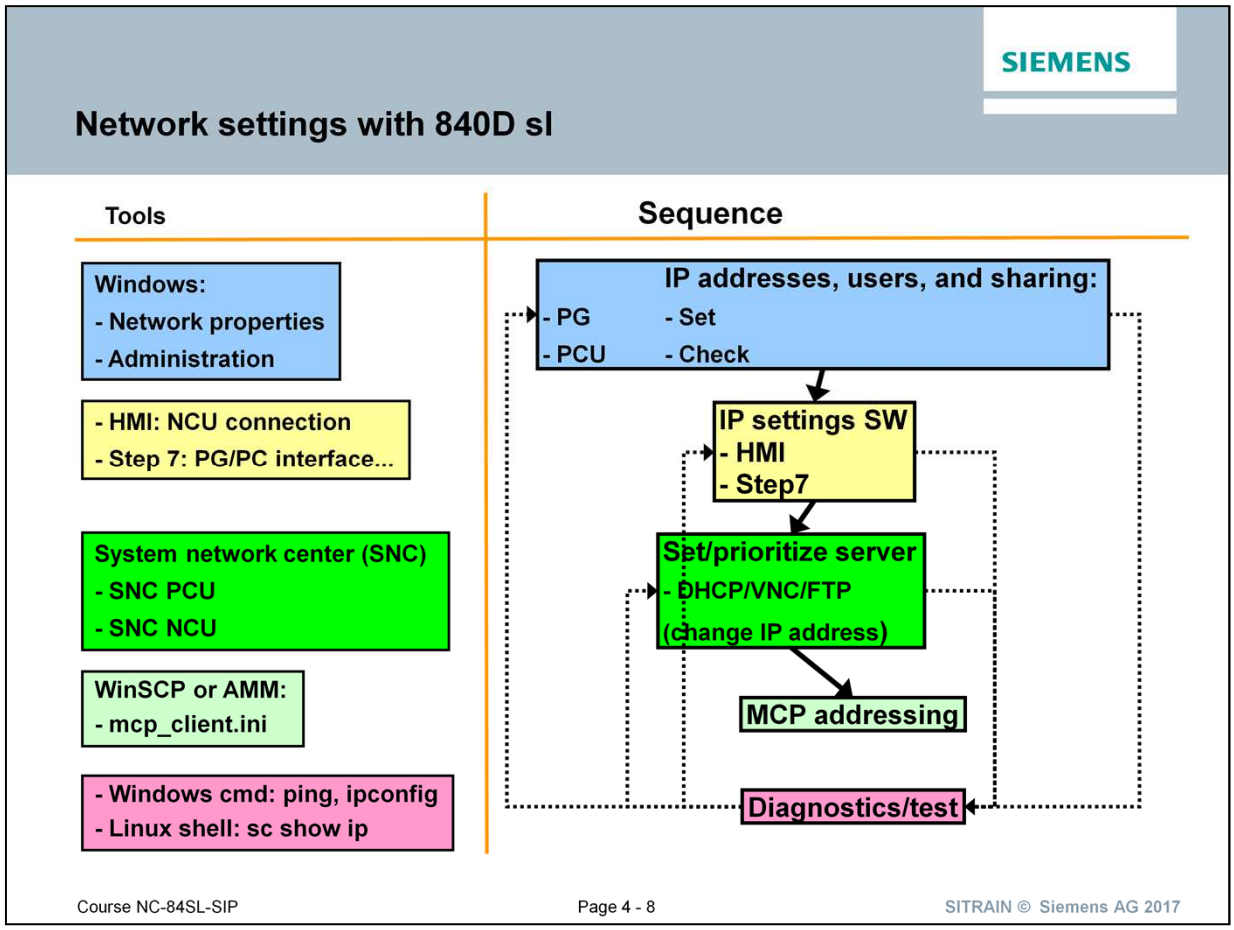
Transmission **C**ontrol **P**rotocol / **I**nternet **P**rotocol. Originally developed for WANs, TCP/IP has also established itself in LANs. Many utilities have been ported to Windows NT.
This protocol is recommended for larger networks with hierarchical structures. It is the standard protocol for UNIX networks. A disadvantage that is worthy of note is the relatively large header (64 bytes), However, although this reduces performance, it represents no real hindrance on today's high-speed networks. The CSMA/CD **C**arrier **S**ense **M**ultiple **A**ccess/**C**ollision **D**etection access method is used.

DLC

Data **L**ink **C**ontrol protocol. The Primary Setup Tool uses the DLC protocol when accessing Simatic components. DLC stands for **D**ata **L**ink **C**ontrol.


SIEMENS
Start-up tools for 840D sl: Networks
Werkzeug
Funktionen
Windows:
- Network properties
- Administration
Configure network PCU/PG
Configure user PCU/PG
- Step 7
- SINUMERIK Operate (PCU)
Set PG/PC Interface
NCU connection
Network settings
- Operate area diagnosis
- Operate area startup
Network diagnosis
Network configuration
- WinSCP
- VNC Viewer
- TCU menu
Linux Service Shell
Display SNC (NCU SW 4.5 max.)
Linux Service Shell
Windows cmd
Linux service shell
Execute Windows commands, e.g. ping
Linux commands; (e.g. start SNC (NCU))
Relevant für Netzwerke
ggf. einzustellen
Course NC-84SL-SIP
Page 4 - 7
SITRAIN © Siemens AG 2017

Notes



Network settings:

PG/PCU The following settings must be made on the PG and/or PCU:
- IP address
- User and authorizations (e.g. for Ghost backup of PCU)
- Sharing

They are made using standard Windows tools like the network settings and user administration and/or in the Explorer.

Software HMI, Step7 Access paths must be declared for other software such as Step7 and HMI (on NCU/PCU or PG). This is done using:

- SK: NCU connection for HMI-Advanced or SINUMERIK Operate on PCU

- NCU Connection Wizard for HMI-Advanced or SINUMERIK Operate on PG/PC

- Set PG/PC interface for Step 7

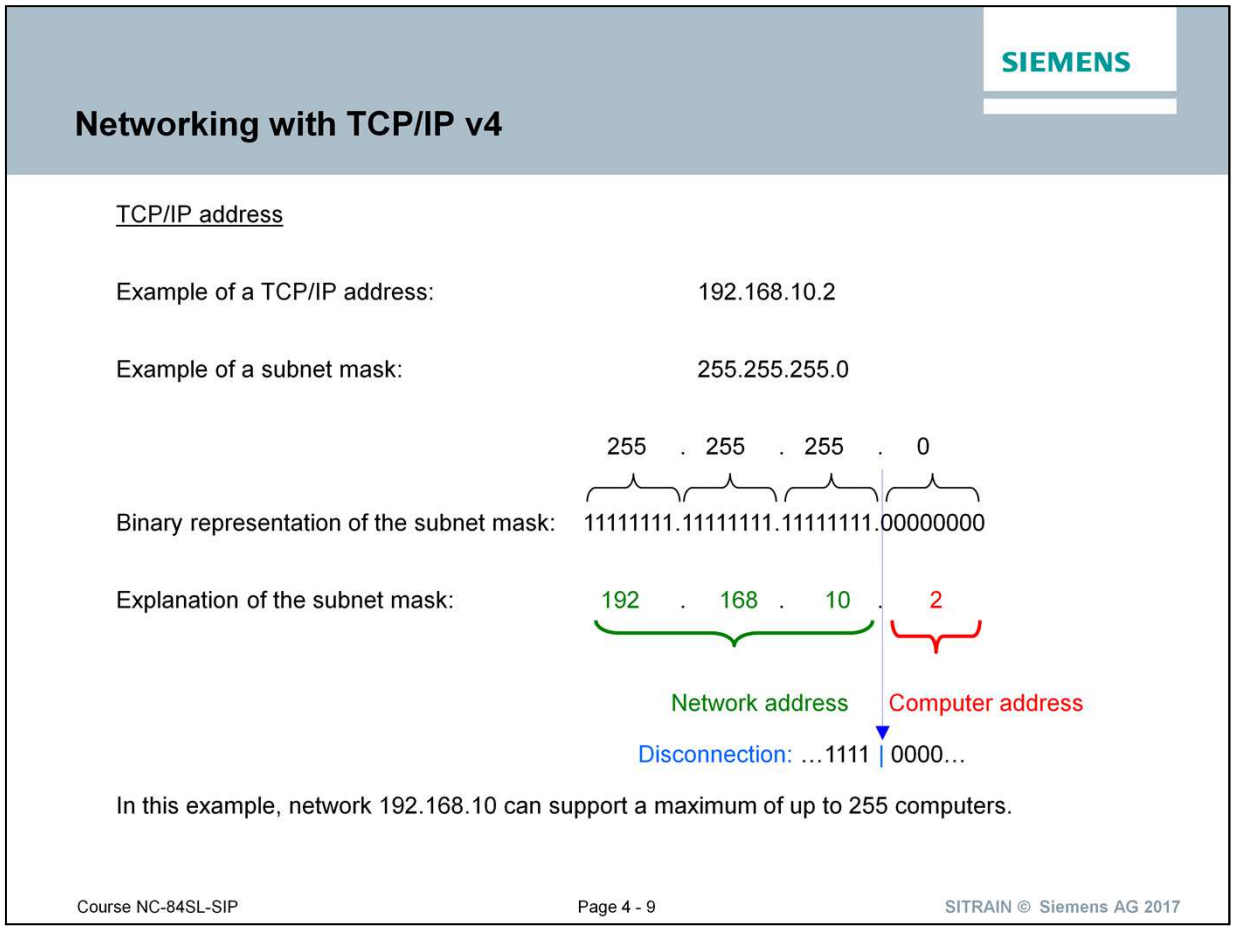
Server settings If there is more than one DHCP server for the TCU(s) and MCP(s), one server must be given priority.
The same applies if there is more than one boot server.

- SNC PCU/NCU This setting can be made with the user-friendly system network center (SNC) for the PCU or NCU.

The IP address of the NCU can also be set on the NCU SNC.

MCP addressing Normally, no settings have to be made here.
However, the configuration file is mentioned for the sake of completeness.

Tests Command processor cmd (Windows) or service shell (Linux) commands are available to test the network settings.



TCP/IP:

The main feature of TCP /IP networking is that the networked computers use IP addresses consisting of 4 three-digit numerals separated by a period. This combination of numbers identifies the network where the remote computer is located as well as the computer number.

Only computers within the same network can communicate with one another, unless routers are used.

Subnet mask

The subnet mask divides this number combination into network and computer number.
The binary representation is from left to right. The contiguous "1"s in the subnet mask represent the network address. The first "0" in this row of numbers is a separator. This is followed by the computer address. It is also possible for the separating point between the network and computer addresses to be located within a byte.

Example

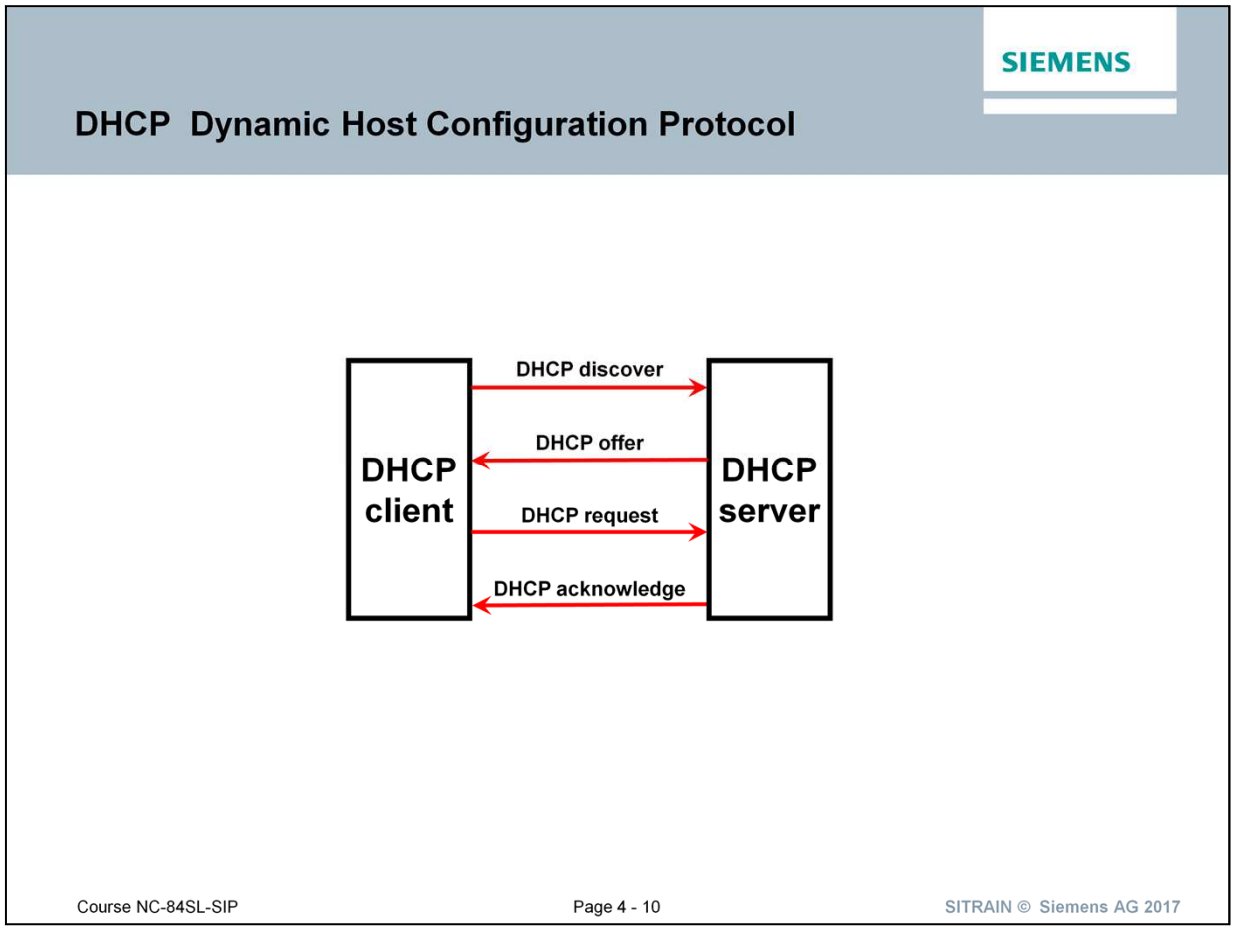
The subnet mask is 255.255.255.0
This segmentation ensures that this network can support up to 254 computers, as the separating point (0) is at the end of the third digit. Therefore, the total final byte is available for computer addresses.

Note

Computers in a network group must not have the same name, the same IP address, or the same MAC address; if they do, correct network operation will not be possible!

IPv4/IPv6

IPv4 supports an address space of just over four billion IP addresses (2^{32} = 4.294.967.296) for addressing computers and other devices. It was estimated that the final IPv4 networks would be assigned to Regional Internet Registries in January 2011 and the Internet community would run out of addresses approximately 1 year after this. This was indeed the case. IPv6 extends the address space to 2^{128} (≈ 340 sextillion ≈ $3.4 \cdot 10^{38}$) addresses, i.e. expansion by a factor of 2^{96}. An IPv6 address looks like this: 2001:0db8:85a3:08d3:1319:8a2e:0370:7344. This is equivalent to several hundred million addresses per m² of the earth's surface, including the sea.



DHCP - Dynamic Host Configuration Protocol

DHCP is capable of fully automatic administration and dynamic distribution of IP addresses in TCP/IP networks. There is now no longer a need to assign a fixed IP address to a station.
For a TCP/IP network, the following settings must be made on every station:

- Assignment of a unique IP address
- Assignment of a subnet mask
- Assignment of the default/standard gateway

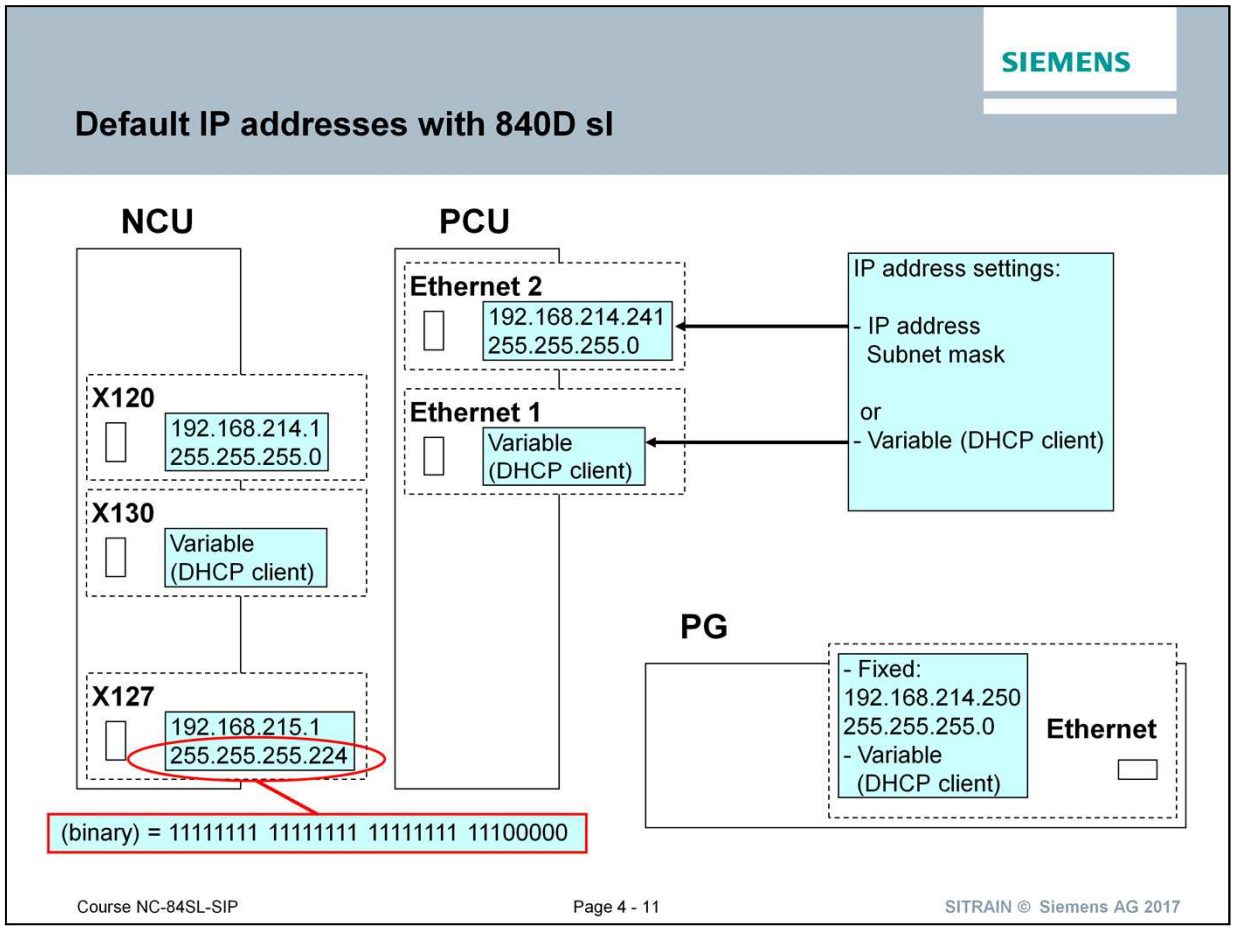
Mode of operation

DHCP is a client/server architecture. The DHCP server has at its disposal a pool of IP addresses which it can freely assign to DHCP clients. When a station starts up and a DHCP client is activated, a TCP/IP stack with restricted functionality is running. The stack does not have a valid IP address, a subnet mask, or a standard gateway. The only thing the client is able to do is to send IP broadcasts. The DHCP client sends a UDP package with destination address 255.255.255.255 and source address 0.0.0.0. This broadcast serves as an address request to the available DHCP servers. The UDP package contains the hardware address (MAC address) of the station. Every DHCP server addressed responds by sending back a UDP package containing the following data:

- MAC address of the client
- Possible IP address
- Validity period of the IP address
- Subnet mask

IP address of the DHCP server

The DHCP client searches for an IP address from the selection of DHCP servers (in some cases there will be more than one server to choose from). It then sends a positive acknowledgment to the corresponding DHCP server. All other servers also receive the acknowledgment and assume that the client has taken an IP address from another server. Finally, the DHCP server must confirm the assignment.



Reserved IP addresses at X120 on the NCU

IP address	Network station	Remark
192.168.214.1	NCU on X120	Default
192.168.214.2 – 9	For additional NCUs with a fixed IP address on the system network	Unassigned
192.168.214.10 – 240	For additional TCUs, subsequently for additional PCUs, NCUs, MCPs, MPPs	DHCP clients
192.168.214.241	Fixed IP address of the PCU on Eth 2	Default
192.168.214.242 – 249	For additional PCUs with a fixed IP address	Unassigned
192.168.214.250 – 254	For PGs with a fixed IP address (service connection)	Unassigned

IP addresses at X127 on the NCU

IP address	Network station	Remark
192.168.215.1	NCU on X127	Default
192.168.215.2 – 9	For PGs with a fixed IP address	Unassigned
192.168.215.10 – 30	E.g., for programming device	DHCP clients

IP address PG

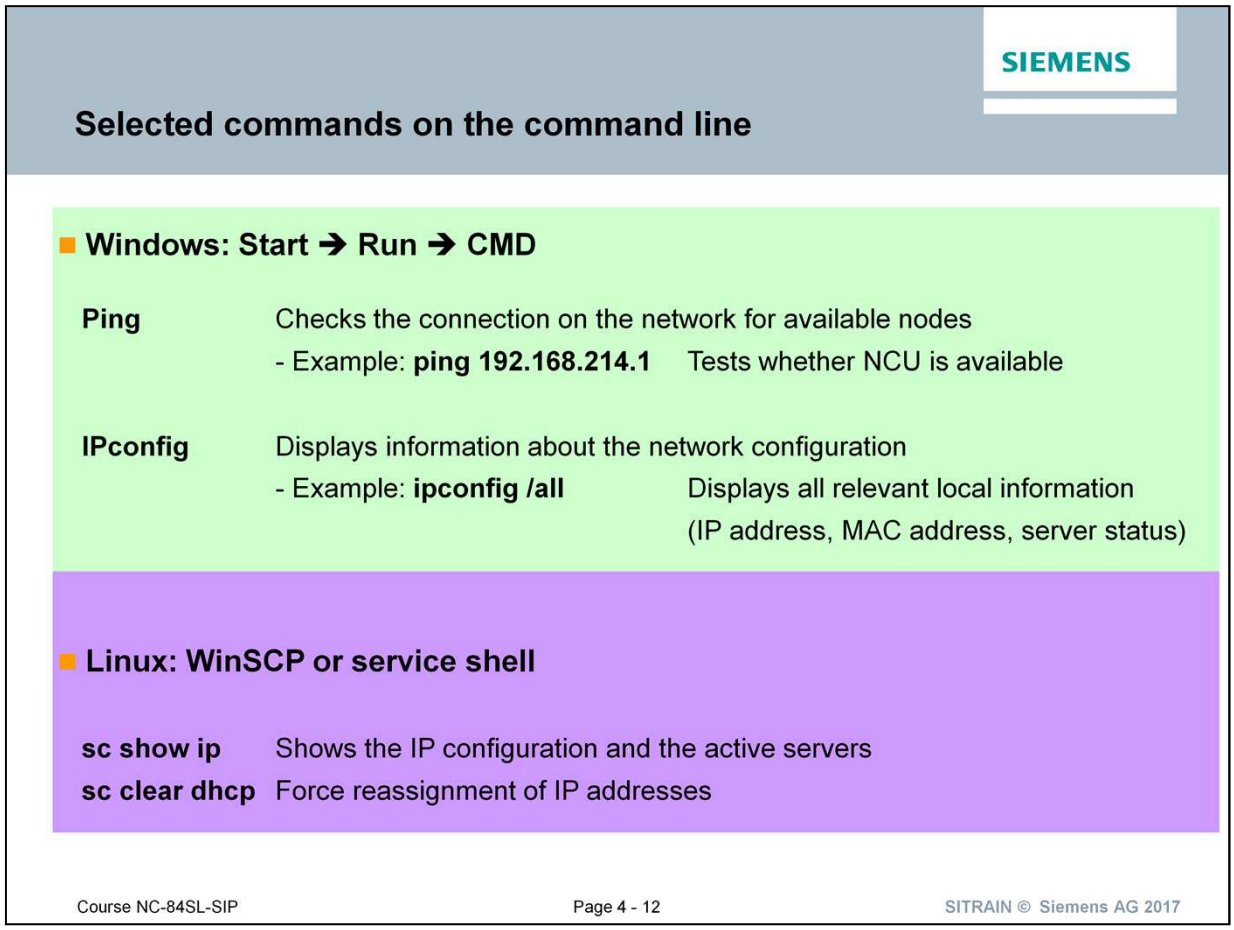
In this case, the ideal setting for the PG is:

- Fixed addresses: IP address 192.168.214.250
  Subnet mask 255.255.255.0

This makes access to the NCU possible at X120 and access to the PCU possible at Ethernet 2.

- Variable: DHCP client ("Obtain an IP address automatically")

This makes access to the NCU possible at X127.



Windows commands:

To execute these commands, select:

➔ **Start** ➔ **Run** ➔ **cmd**

- Ping

The 'ping' is an echo request that is sent to the target computer.
The simplest ping syntax is:
ping <target computer ID>
where <target computer ID> is the address or computer name of the target computer.

- Ipconfig

Displays information about the network configuration.
The simplest way to use this is:
ipconfig /all

Linux commands:

Linux commands require that you access the Linux service shell via WinSCP or the TCU menu. Log in as follows:

➔ **Service shell** ➔ **WinSCP:**

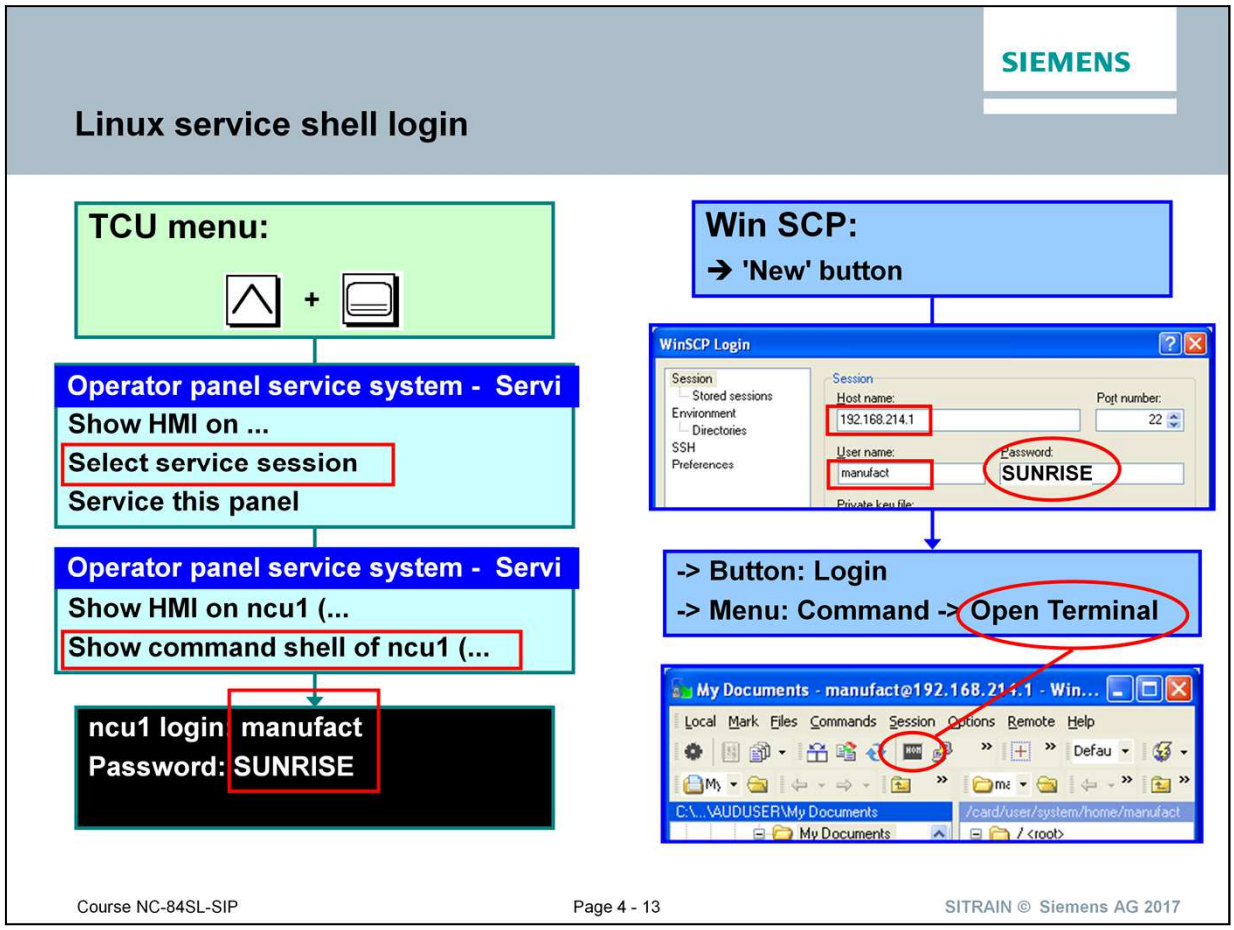
HostName = IP address of target computer
Login = **manufact** UserName = **manufact**
Password = **SUNRISE** Password = **SUNRISE**

- sc show ip

This command shows up-to-date information about the network configuration. It corresponds to ipconfig /all in Windows.

- sc clear dhcp

Forces the reassignment of the IP addresses.
This is necessary after NCU address changes (X120/X130).



Linux service shell In principle, there are three ways of logging in on the Linux service shell.

- TCU menu This only works if a TCU is installed on the operator panel.

The sequence of operations is as follows:

 **RECALL + MENU SELECT**

➔ **Select service session**

➔ **Show command shell of ncu1 (192.168.214.1)**

➔ ncu1 login: **manufact**

➔ Password: **SUNRISE**

- PG The Linux service shell can also be accessed from the PG with VNC Viewer using IP address: port = 4 (login as above):

192.168.214.1:4

- Win SCP Linux commands can also be sent with this tool.

The sequence of operations is as follows:

➔ Button: **New**

➔ Complete the login dialog as follows:

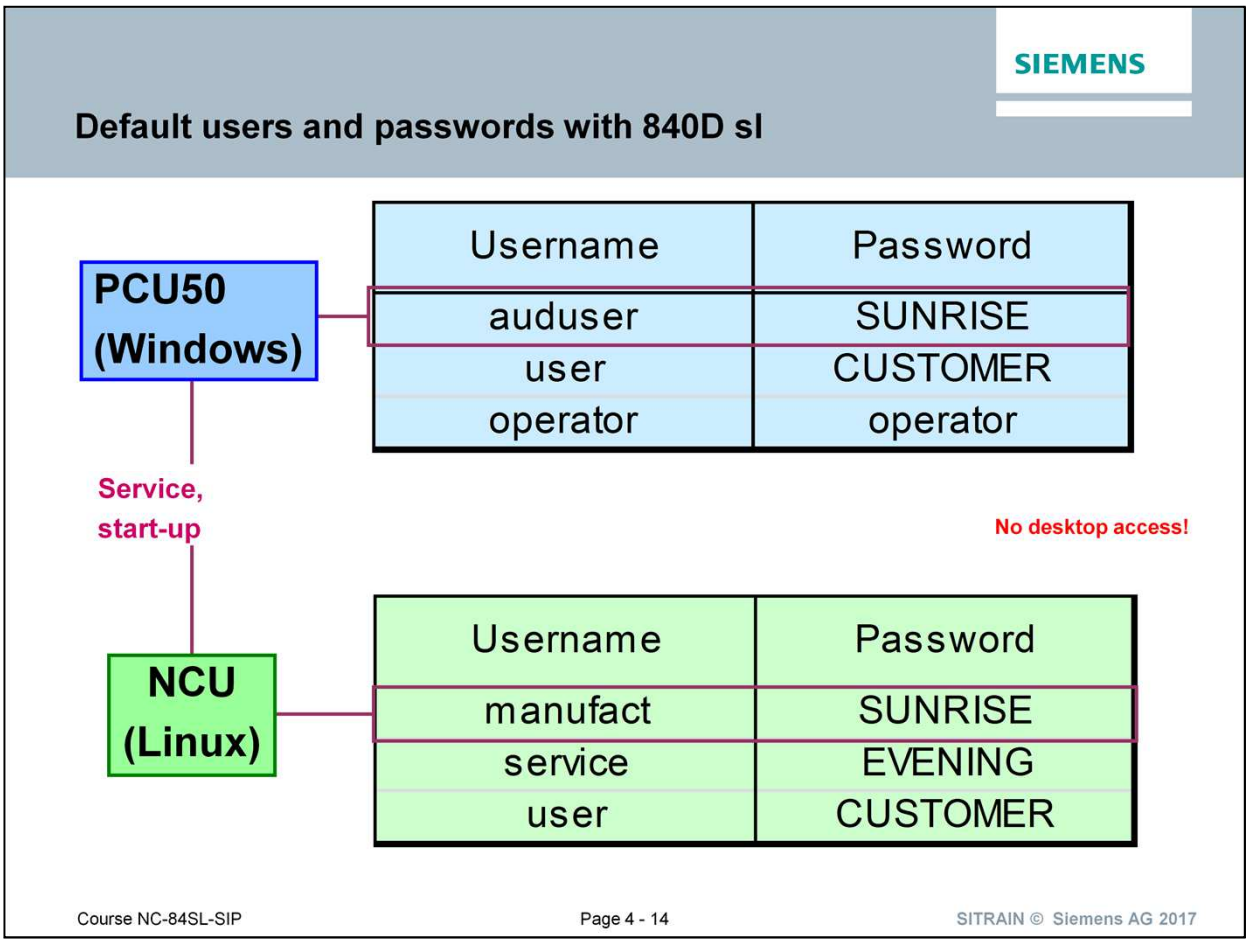
➔ HostName = IP address of target computer

➔ UserName = **manufact**

Password = **SUNRISE**

➔ Menu option: **Command ➔ Open Terminal**

-- Remark Click the **'Save'** button to save the login settings.



Users on PCU 50 (Windows XP)

The default users on PCU 50 are:

- **auduser** Password: SUNRISE
- **user** Password: CUSTOMER
- **operator** Password: operator

➔ See “Commissioning Base Software and Operating Software" in DocOnCD.
Only the 'auduser' user is recommended for service and start-up.

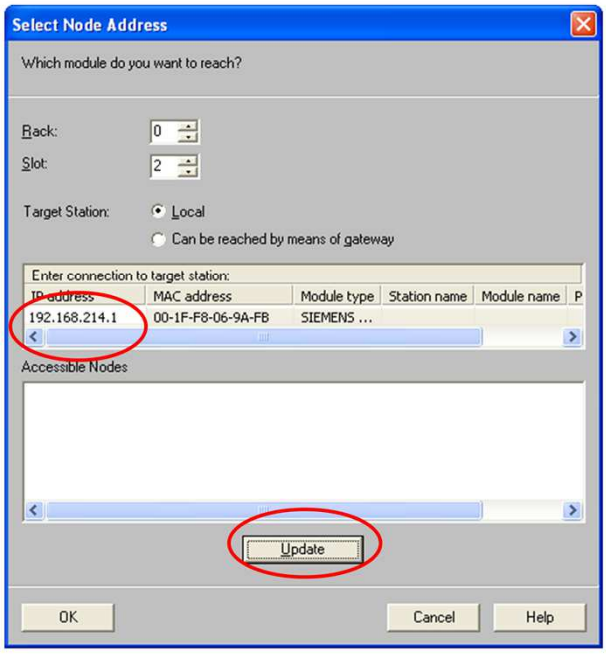
Users on NCU (Linux)

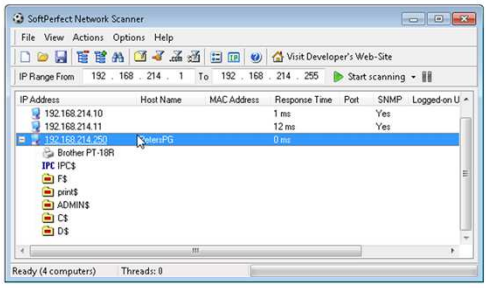
The default users on NCU are:

- **manufact** Password: SUNRISE
- **service** Password: EVENING
- **user** Password: CUSTOMER

➔ See “Commissioning Base software and HMI Advanced" in DocOnCD.

Finding the IP Address of the target station





Course NC-84SL-SIP Page 4 - 15 SITRAIN © Siemens AG 2017

One way to find the IP address of the target station is to take a look on the corresponding device.
Alternatively, other methods are available.

IP scanners

IP scanners are programs (Internet downloads) that support the automated sending of PINGs to a series of IPs and automated checking of responses. If a positive acknowledgment is returned, this computer is then displayed - according to user-friendliness and scanner settings - in a "Live Hosts" list. Many of these scanners have additional functions to allow access to the located computer.

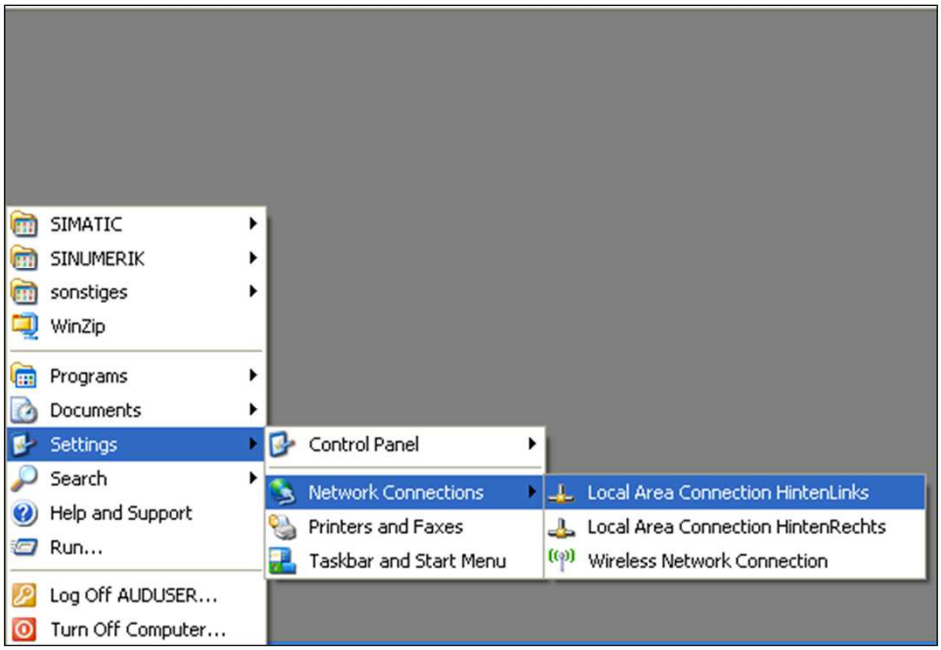
SIMATIC Manager

The SIMATIC Manager can be used to start a configuration upload in the component hardware configuration. More information about accessible nodes is displayed by clicking the "Update" button.

- Note

With SINUMERIK NCUs 7x0.3 you have to enter the IP address manually in the corresponding field of the dialogue ‘Select Node Address’!

Setting IP and subnet mask



 SITRAIN © Siemens AG 2017

The TCP/IP address is set for the network adapter you want to use.
The address is a property of the TCP/IP protocol which is used by the network card of the PC/PG.

Classes

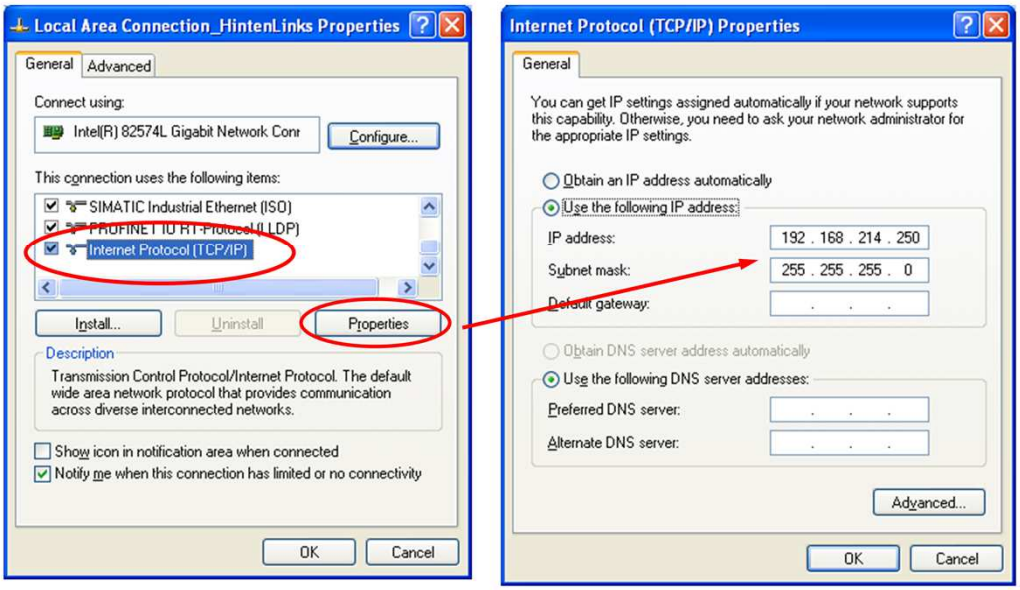
The subnet mask defines 3 network address classes.
Class A Networks with the subnet mask **255.0.0.0**
Class B Networks with the subnet mask **255.255.0.0**
Class C Networks with the subnet mask **255.255.255.0**

2 combinations are never used within the classes for the IP network identifier.
Bits are never all set to 1 or all set to 0.
Consequently, only 127 networks remain for a Class A network.
Another special IP address is 127,

Loopback address

which is a loopback address which is reserved for internal purposes (e.g. connecting an HMI Advanced on SINUMERIK 840Di sl to the NCU).
Therefore, only 126 networks remain for Class A networks. The remaining bytes represent the addressable area of the hosts. This amounts to 255*255*255 combinations which corresponds to address space for more than 16 million computers within this network scheme. The other network classes have correspondingly more networks but as a consequence fewer hosts.

Setting IP and subnet mask



 SITRAIN © Siemens AG 2017

Select the TCP/IP protocol for the adapter and click Properties. The input screen for the IP address appears. You may need to select the option "Use the following IP address".

DHCP

The option "Obtain an IP address automatically" is used if the network contains a DHCP server.
DHCP stands for Domain Host Configuration Protocol. If a PC/PG is connected with a network containing a DHCP server, the PC sends the server its MAC address and receives an IP address from it if there is no access restriction.

Dynamic IP

Each time the PC logs on, it is assigned the same or a new IP address depending on the settings of the DHCP server.
The behavior of the server depends on the lease time setting (server setting). If a PC logs in again within the time set here, it is assigned the same IP address. If the lease time is infinite, the IP address is "virtually static". Outside the lease time, the PC is assigned a new address when it logs on.

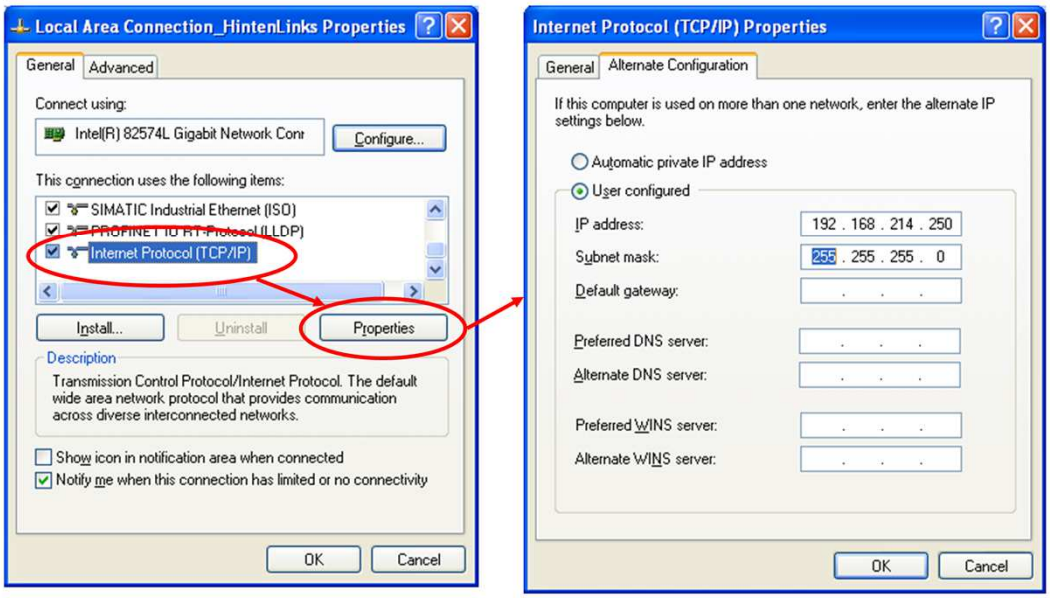
Remark

After making changes to the TCP/IP configuration, it may be necessary to unplug the Ethernet cable from the PG and plug it back in again.

This applies in particular if the cable on the NCU has been plugged into different port:

Switch (X120) <-> Service port (X127)!

Alternate IP addressing? ➔ Fixed IP is better!



 SITRAIN © Siemens AG 2017

You can set alternate IP settings on recent Windows operating systems. If the network connection is not established under the first setting, the alternate setting is used. If a connection cannot be established here either, a corresponding message is issued.

Problems Several problems might occur here:

1. Timing
- The IP address does not change over immediately when the network cable is plugged back in.

Example: Reconnect NCU X120 ➔ X127 Immediate connection
X127 -> X120 connection after 60 s

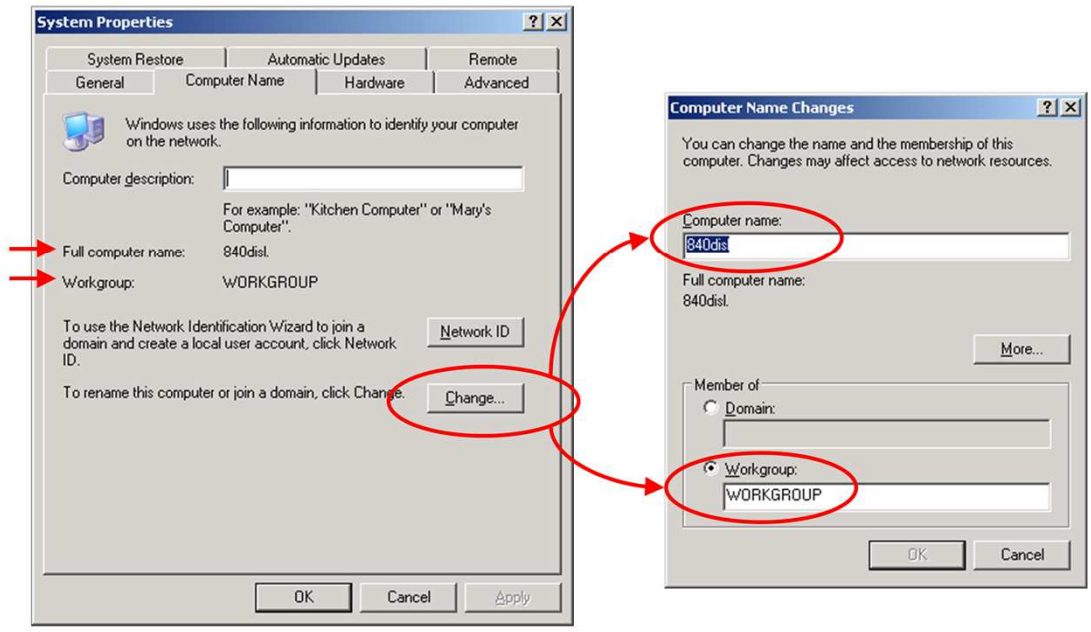
2. Old IP address
- In some cases, following automatic address assignment (DHCP server) and switching to a connection without a DHCP server, the old address is retained and a connection cannot be established.

3. NC reset
If the NC is reset, communication will fail. However, in this case, the alternate IP address is set and the HMI will no longer be connected, for example.

2./3. ➔ Remedy:
- ➔ Command: **ipconfig /release** Release IP address
- ➔ Disconnect/connect network cable

➔ Summary
Rather than using an alternate IP address for X127,
it is better simply to activate automatic address assignment.

Setting the computer name



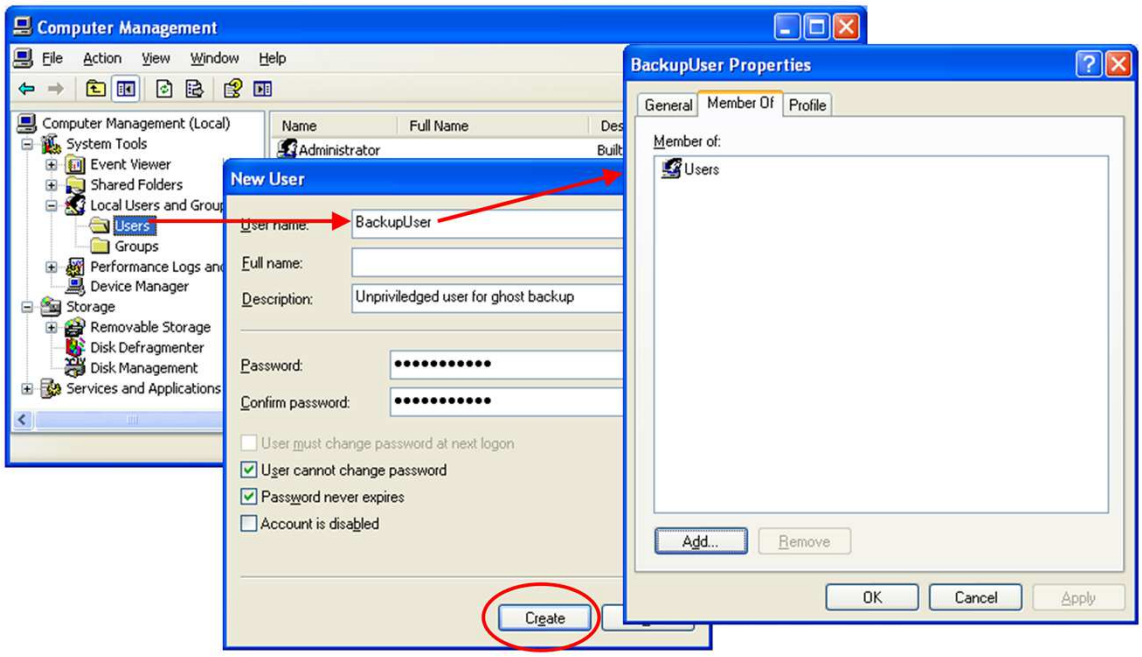
 SITRAIN © Siemens AG 2017

The dialog boxes shown above are accessed via "Start > Settings > Control Panel > System".

Note!

Please note that in administered networks generally only domain administrators with appropriate rights can change the domain! In general it is possible to leave the domain, but it is only possible to return with the administrator's help.

PG/PCU: Creating Windows user and group assignment



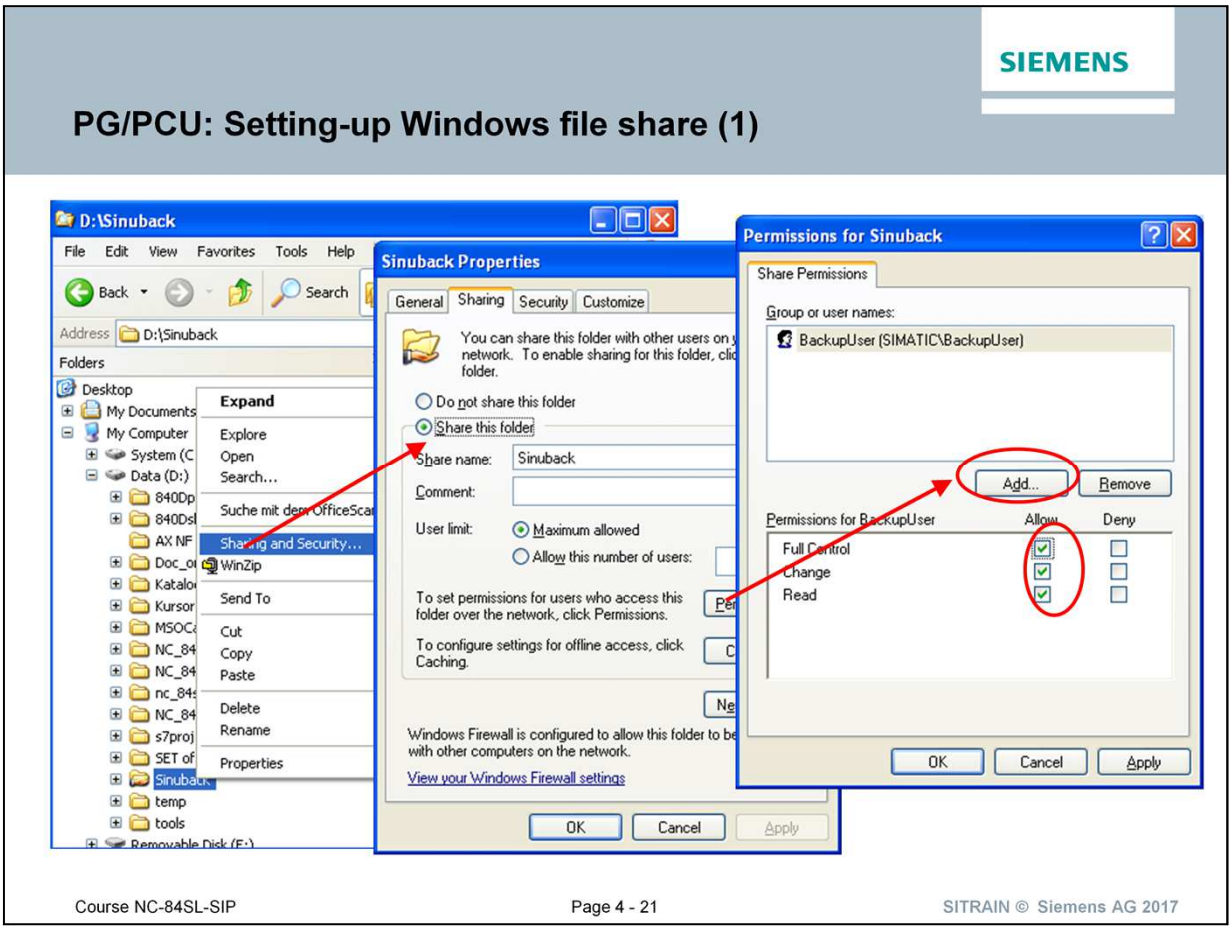
 SITRAIN © Siemens AG 2017

Computer management is accessed via the Control Panel. If you have permission to create new users, you can create new users in the "Local users and groups" subdirectory and assign these to user groups.

Important

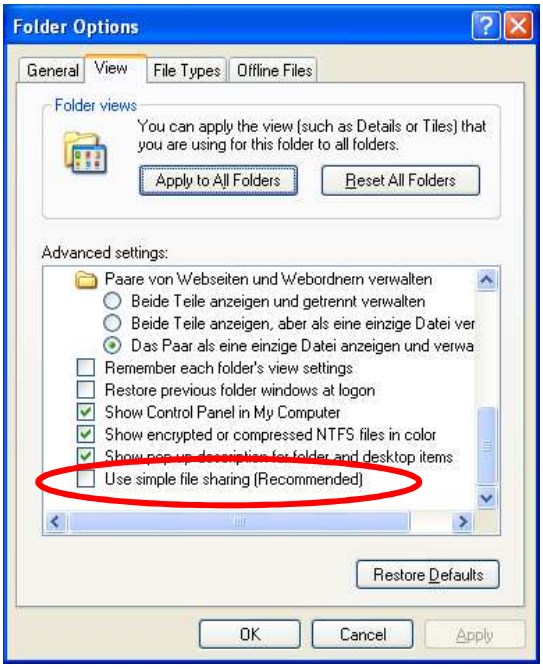
The newly created user account should be used only as access for the Windows file share to be created in the next step. Therefore, for security reasons, the following settings should be observed:

- The user cannot change the password (in order to avoid unauthorized access)
- The user should belong to a privileged user group that should be as small as possible (i.e. **not** "administrators"!)
- The password should be a real password - i.e. not "SUNRISE" ! (min. 8 characters, upper and lowercase letters, numbers + special characters)

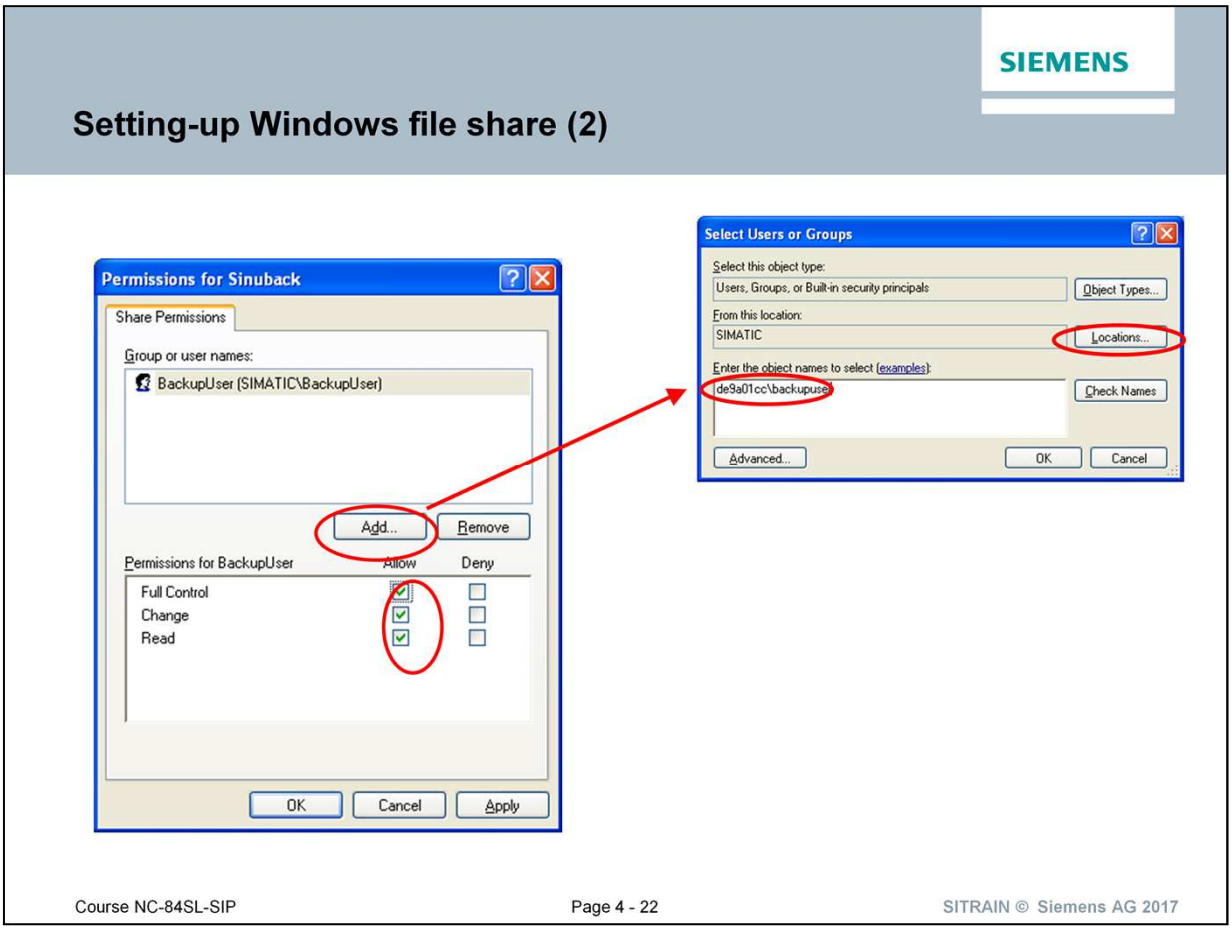


For **security reasons**, when setting up the file sharing strategy in Windows, the following issues should be given careful consideration:

- Standard file sharing: in Explorer via "Extras > Folder options > View" Deactivate "Use simple file sharing" (see the diagram under "Folder options")



- Under "Authorizations", users authorized to access the system are configured (see the next overhead)



- Add the previously created (non-privileged) user via "Add"

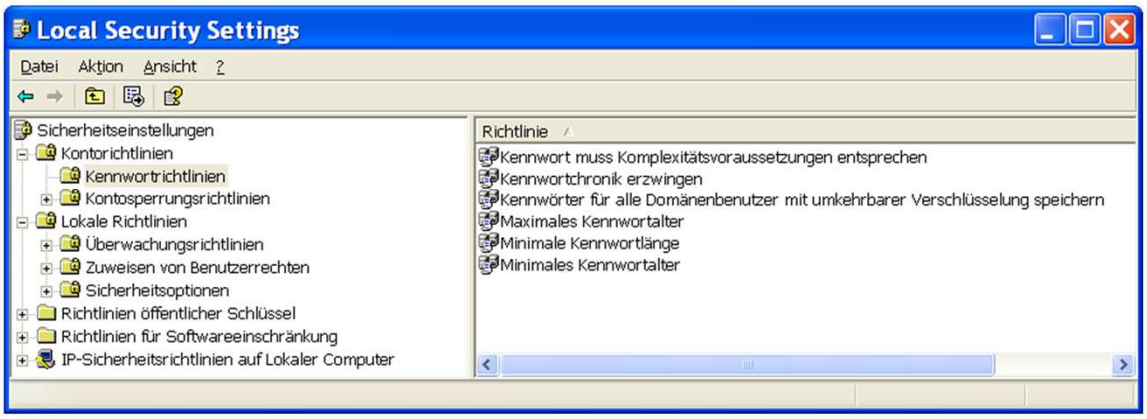
Note for PGs in a Windows domain

The previously created user was created locally on the computer - i.e. is not known to the domain. If only the user name is specified when using the search function ("check name"), Windows will not find it. That is why the computer name must be specified in front of the user name so that Windows knows where to look for the user:
Syntax: **<Computer name>\<user name>**

As an alternative, the search path can be set via "Paths" (select your own computer name).

- The added (non-privileged) user is now allocated full access
- If another user is already listed (e.g. the user presently logged on)
  => then this user must be deleted with "Remove" (important!)
- Also, the "Everyone" user must be deleted => otherwise this represents a major security vulnerability!

PG/PCU: Password policies

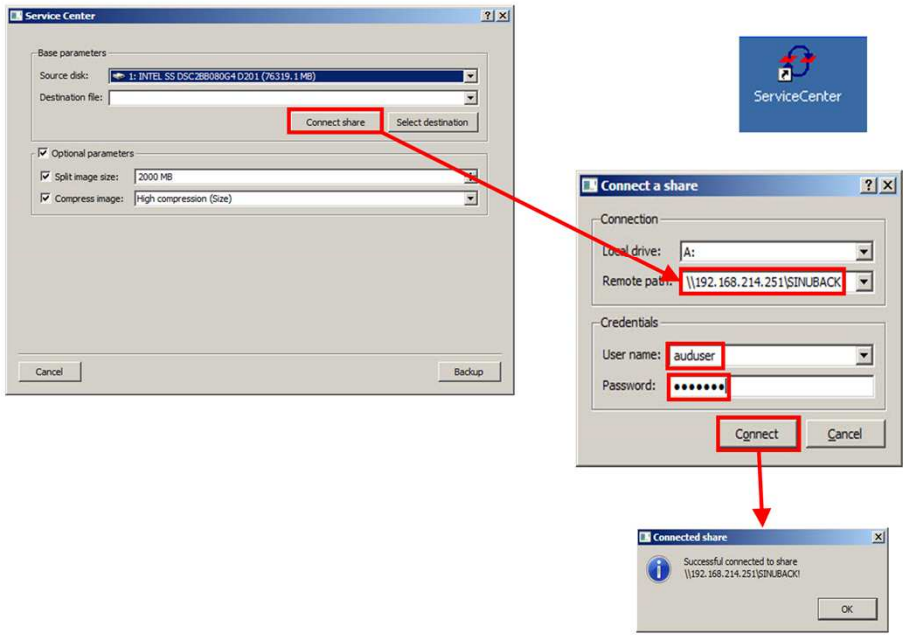


 SITRAIN © Siemens AG 2017

Both with local and domain-based login, Windows offers different options for setting different policies. These include meeting requirements for complexity, for example. For example, it could be necessary to include a special character in the password to help prevent brute force/dictionary attacks on passwords. Passwords that can be guessed, e.g. children's name, birth dates, etc. are vulnerable to attacks as the attacker can take the path of social hacking and find out information about their target before attacking.

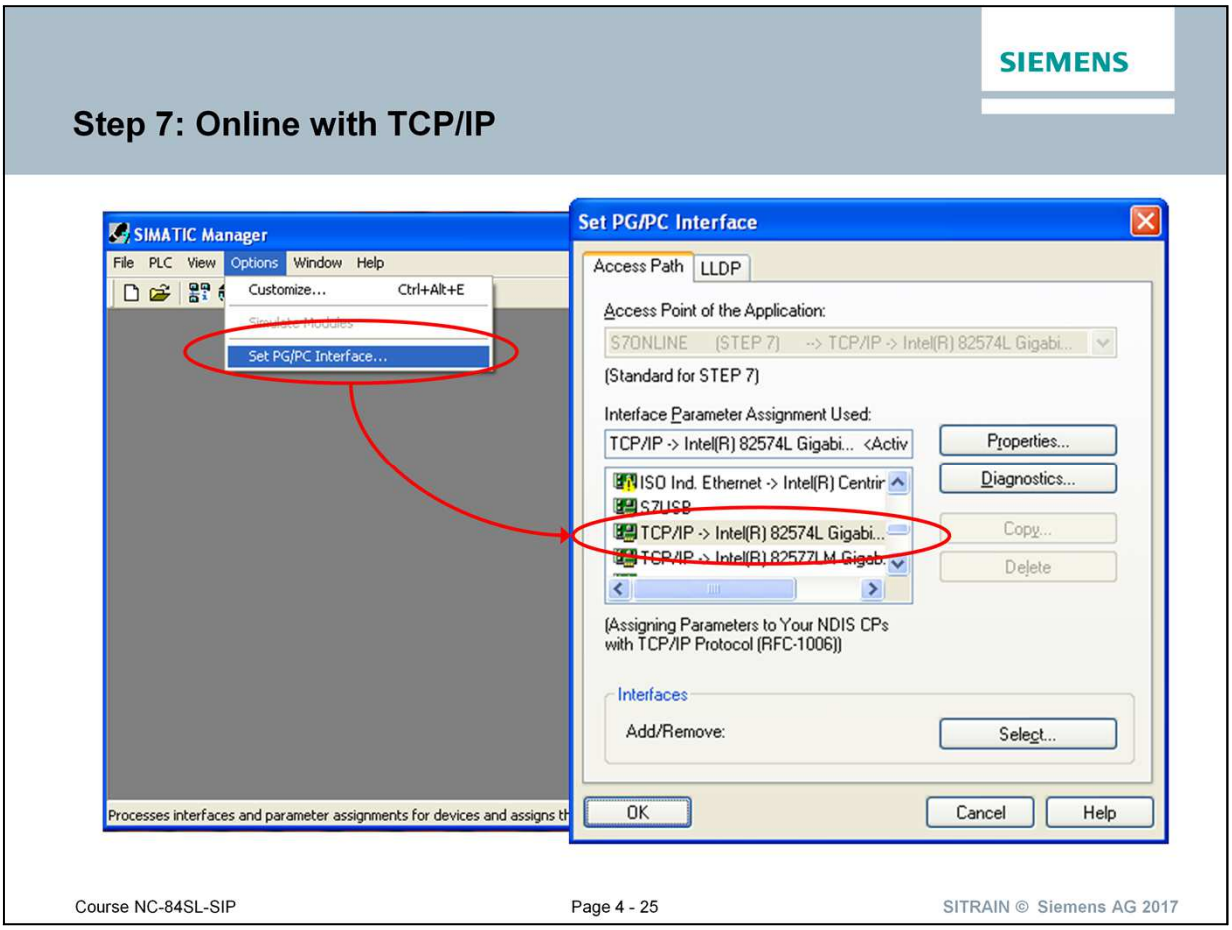
Another security mechanism is to force a password change at specific time intervals instead of assigning permanent passwords. The interval is selected so that a potential attack would take longer than the time interval before the password expires. Passwords that are too short do not provide adequate defense against attackers. To provide added security, Enforce Password History can be set to prevent users from changing between two passwords that are easy for them to remember.

PCU Service Center: Network settings for Ghost



SITRAIN © Siemens AG 2017

You can access the network settings for service purposes via the "Service Center" icon. After setting the correct addresses, you can trigger service mode for backing up and restoring hard disk images.



S7 Online via TCP/IP

We need to modify the access path for S7 online for the subsequent download.

This is achieved by selecting the following setting:

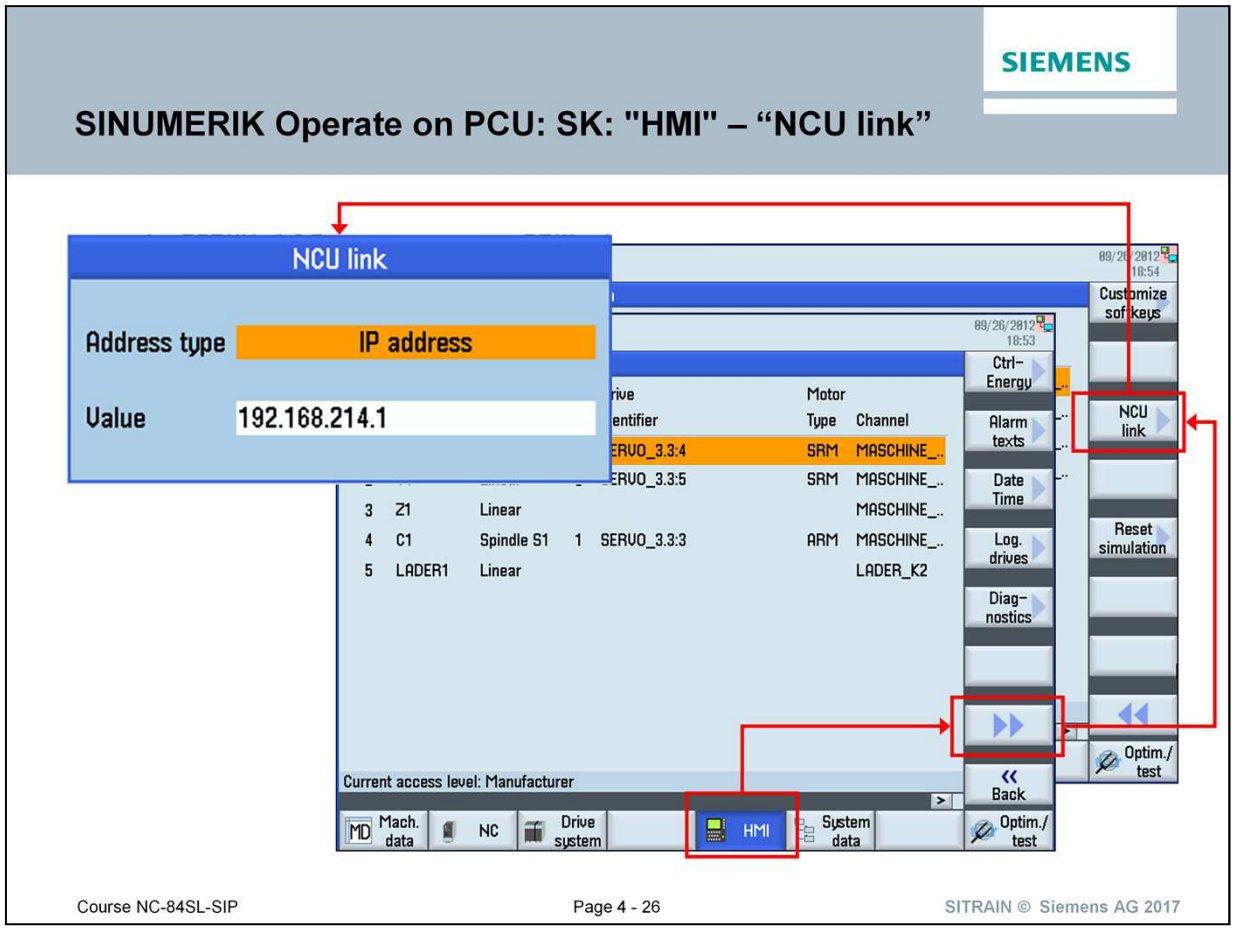
➔ SIMATIC Manager ➔ Extras ➔ Set PG/PC interface

In the dialog box with this name, select:

- **TCP/IP ➔ ETH-HW**

where ETH-HW designates the actual Ethernet hardware.

Example in the screenshot above: ETH-HW = Intel(R) 82577L Gigabit Ethernet.



"NCU link" SK

The IP address of the target NCU must be specified here.

With SINUMERIK Operate on PCU, this softkey can be found as follows:

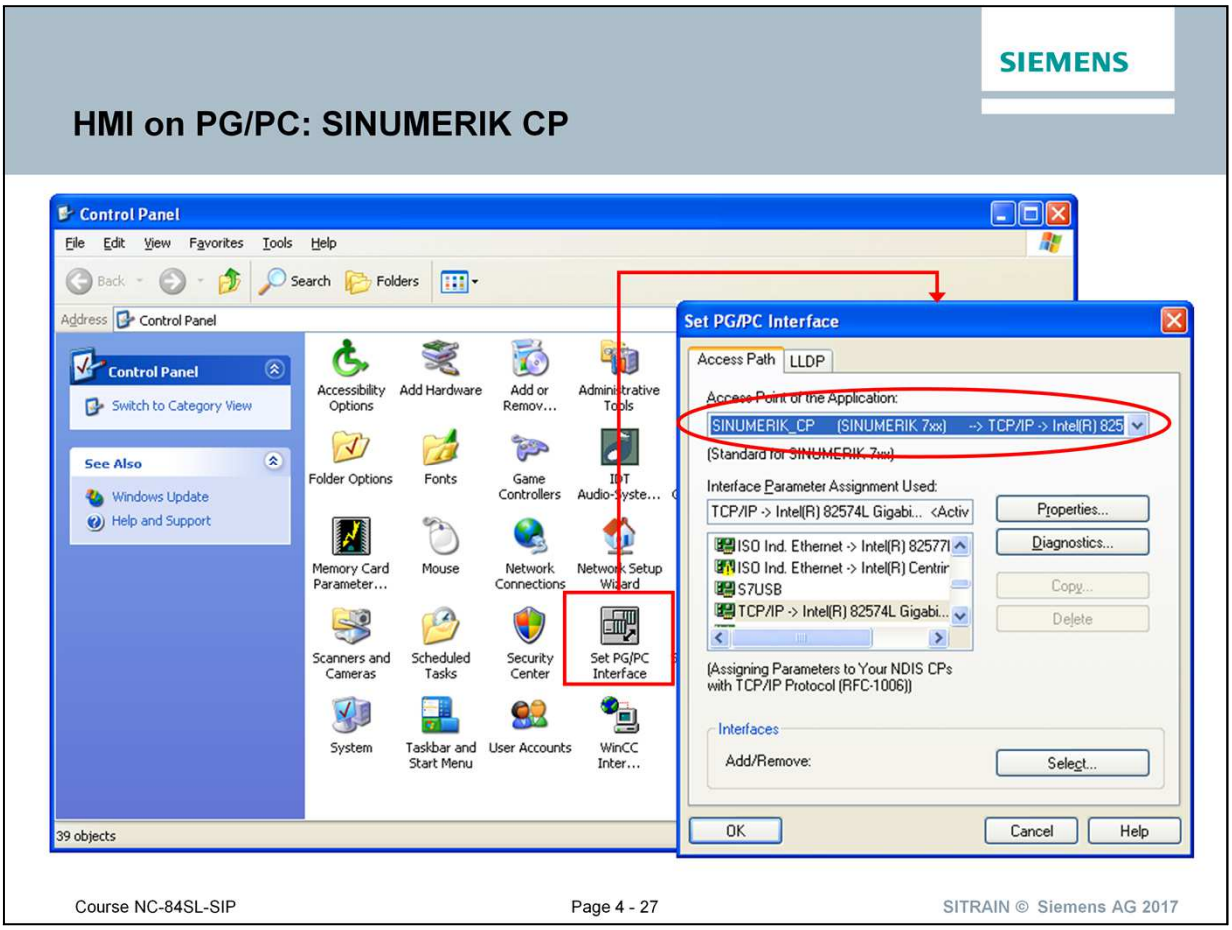
➔ OA: Setup ➔ SK: HMI ➔ SK: >> ➔ SK: NCU link

Remark

NCU and OP must have the same network address, i.e. the bit-by-bit multiplication of IP address and subnet mask must result in the same address.

Example:

	IP address	Subnet mask	IP addr. x subnet mask
NCU	192.168.214.1	255.255.255.0	192.168.214.0
PCU/PG	192.168.214.250	255.255.255.0	192.168.214.0



HMI: SINUMERIK CP

For SINUMERIK Operate on PG, the access point for the SINUMERIK CP on TCP/IP must be set.

This setting can be found here:

➔ **Control Panel ➔ Set PG/PC interface**

On most computers, the setting is made correctly by the Connection Wizard, but in isolated cases it must be entered by hand.

Remark about WLAN

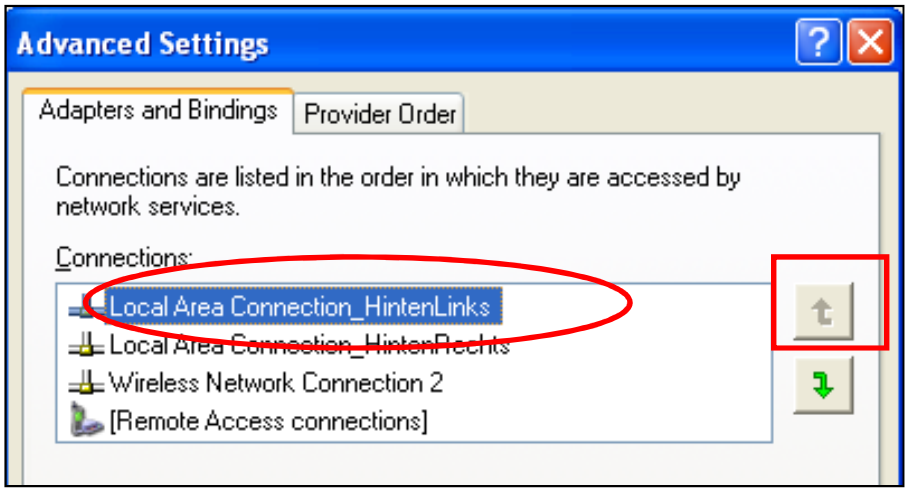
If WLAN is active, the following setting may be important:

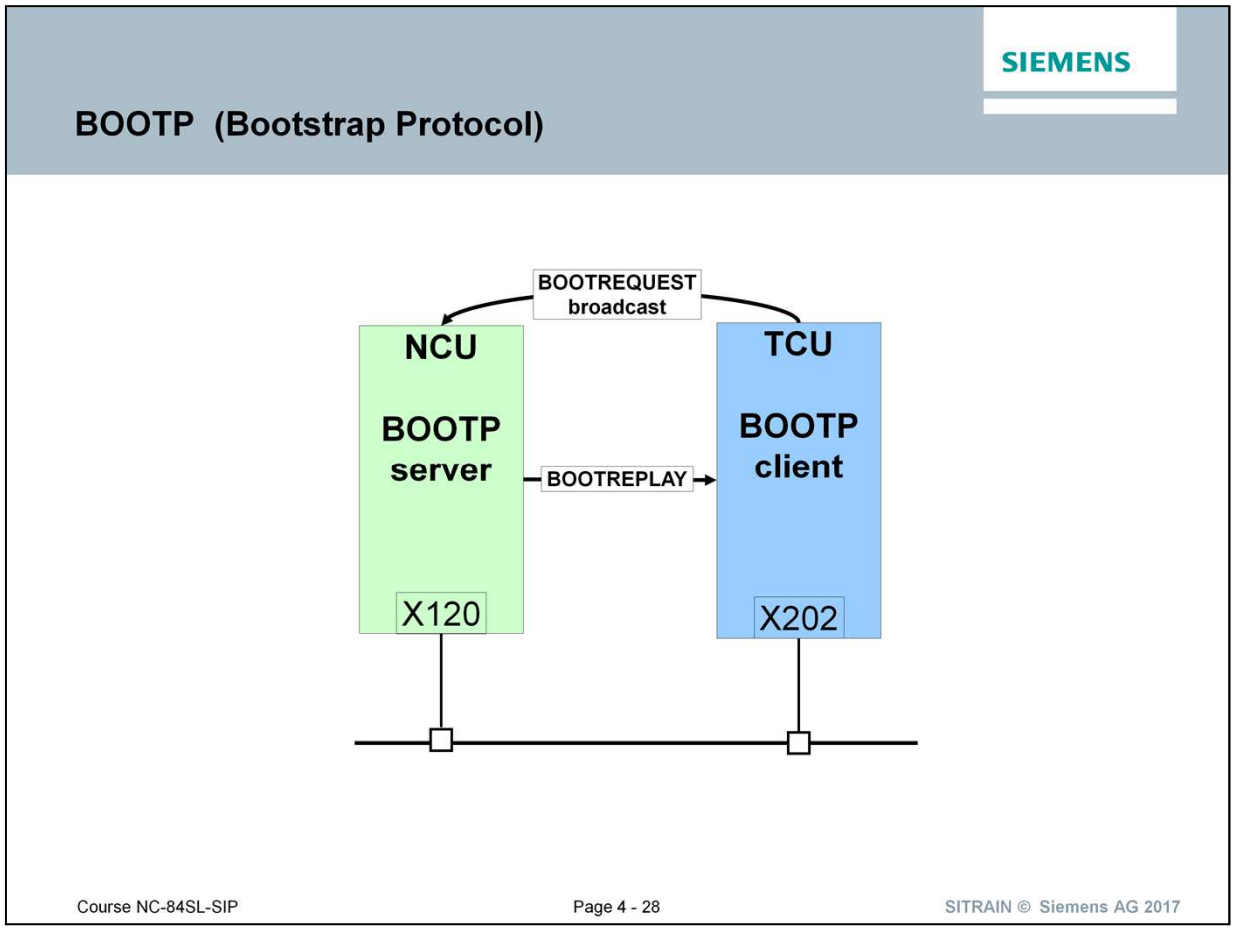
➔ **Network properties**

➔ **Menu: Advanced ➔ Advanced settings**

Use the button to move the interface you are using up the list (priority 1).







BOOTP

(Bootstrap Protocol) is a client/server protocol that allows computers to obtain the parameters they require from a server on system start-up.

The protocol was originally motivated by the desire to operate diskless workstations. Diskless workstations generally use a network file system.

The computer to be started first obtains a kernel from a TFTP server (TFTP stands for Trivial File Transfer Protocol). The operating system is then executed and uses the network file system provided by a server (possibly a different server) as its file system. This procedure is called bootstrapping.

BOOTP tasks

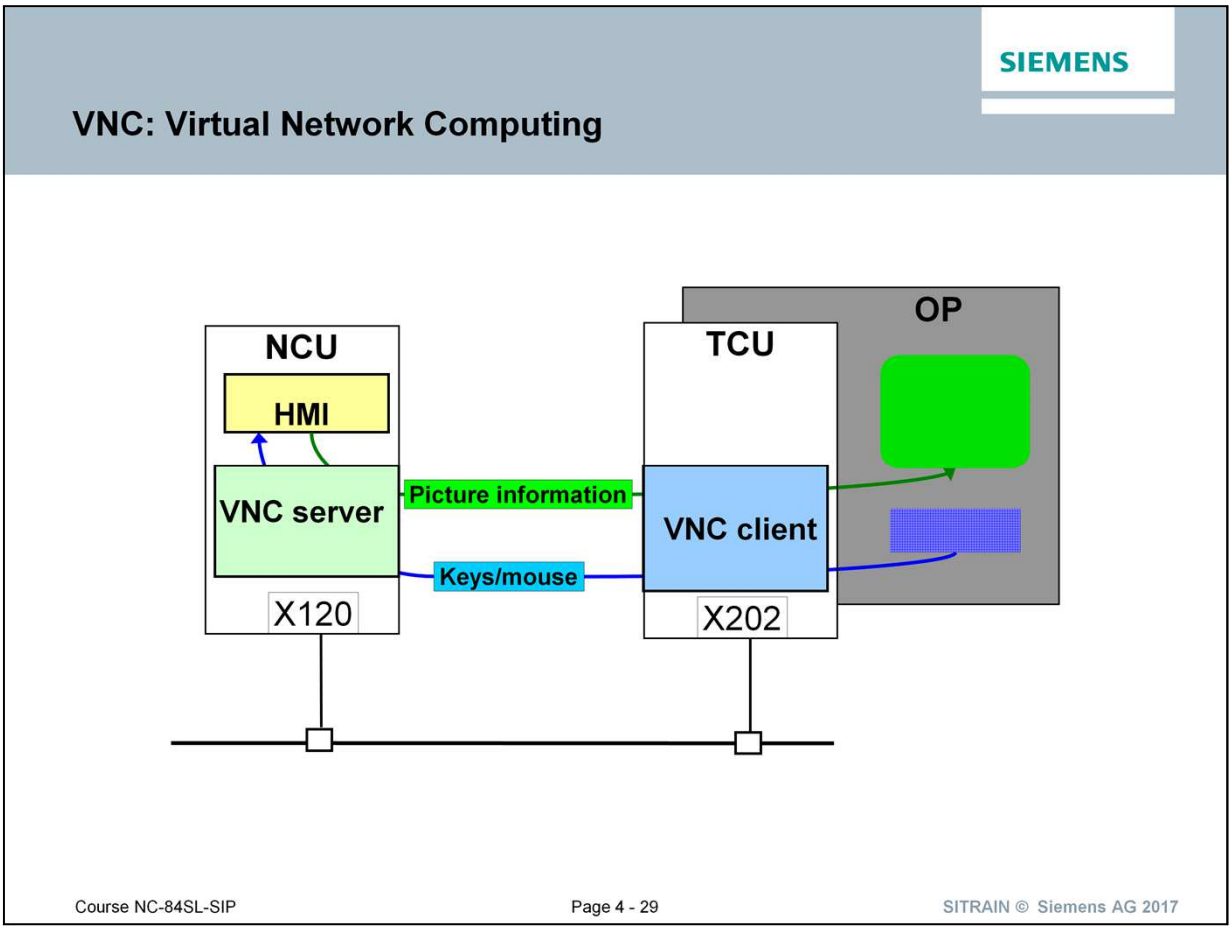
- To find a responsible BOOTP server through a client and send a request
- To transfer the necessary configuration parameters from server to client

BOOTP makes provision for clients that know neither their own IP address nor the IP address of the BOOTP server to send a broadcast request that can be received by all stations on the local network. This means that a client is able to communicate with a BOOTP server without knowing its IP address.

BOOTP protocol

There are two types of message: BOOTREQUEST and BOOTREPLAY. A client sends a BOOTREQUEST message to request the parameters it needs. So that clients (which still do not have IP addresses) can be told apart, they include an ID in their request, usually the hardware address of the network adapter (e.g. the Ethernet adapter). A server receiving a message of this type can use the ID information sent to identify the appropriate parameters for the client concerned and send these in a BOOTREPLAY message.

If a certain period of time elapses without a client receiving a response, that client will simply send the request again. Generally, the BOOTP client functionality is implemented by a type of "miniature operating system" which uses BOOTP to identify the necessary parameters on system start-up and then, in the case of diskless workstations, obtains the kernel via TFTP.



VNC

The VNC (Virtual Network Computing) mechanism can be used to operate a VNC server (computer: NCU or PCU) on the VNC client (OP with TCU).

- Requirement

This requires that the VNC client is already networked (via TCP/IP) with the VNC server.

The standard mechanisms for achieving this are:

- BOOTP Provides the basic software (VNC Viewer) to the client in BOOTREPLAY.
- DHCP Assigns the client a dynamic IP address.

- VNC server

The VNC server provides the HMI picture information for display purposes. A corresponding software program (also called "VNC Server") controls the HMI software on the server with the operator actions.

- VNC client

The "VNC Viewer" software, which receives the picture information and reports the operator actions (keyboard, mouse) back to the server runs on the VNC client.

TCU+OP

These mechanisms mean that the TCU with OP operates like an operator terminal.

PG

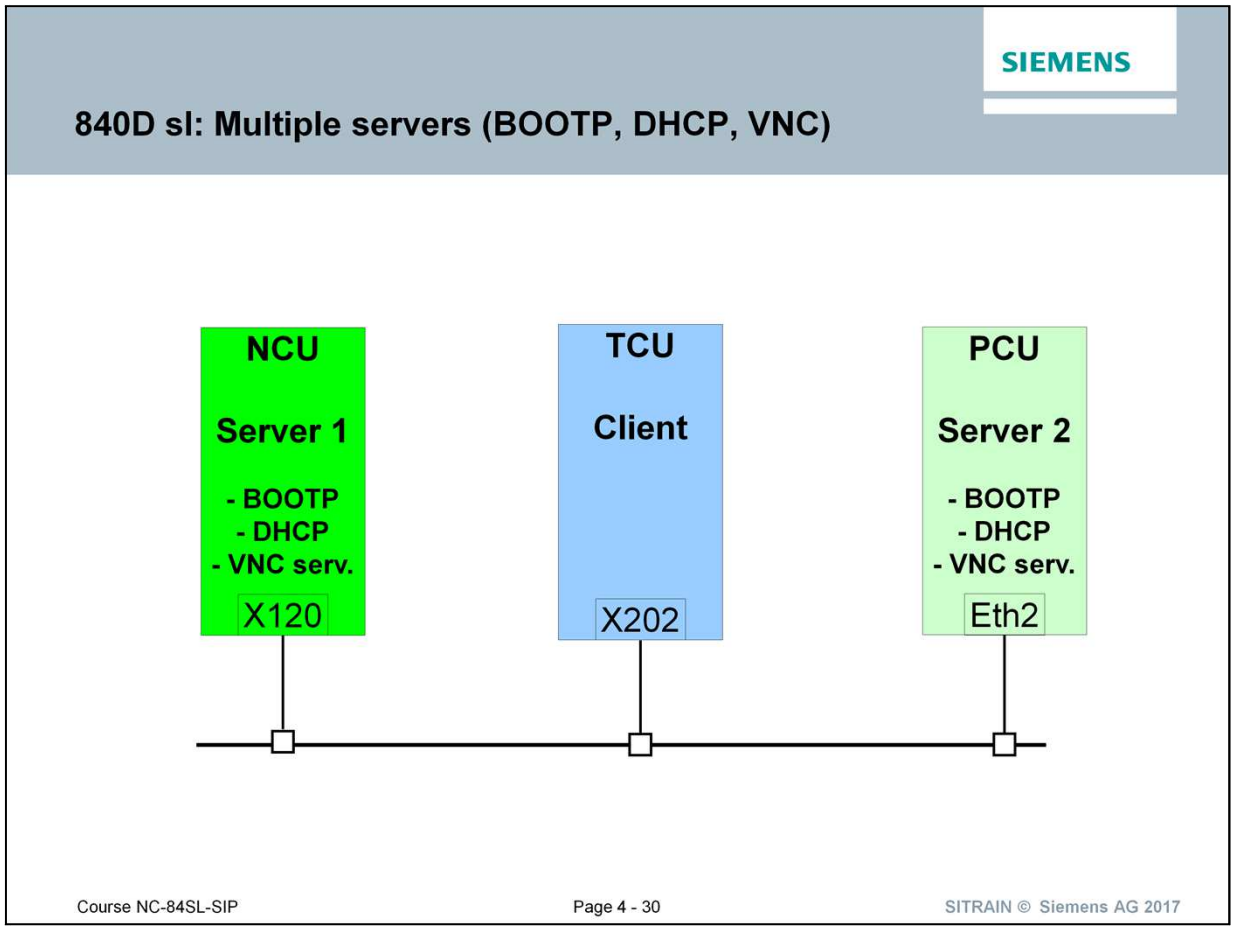
The VNC Viewer can also be used for "remote operation" of the HMI from the PG.

This requires that this remote control is enabled.

➔ **OA: Diagnostics** ➔ **Remote diag.** ➔ Set **“Rights for remote access”** to **“Allow remote operation"** .

The setting is saved in file F:\user_base\system\etc\tcu.ini (or /card/user/system/etc/tcu.ini) under:

[VNCViewer]
ExternalViewerSecurityPolicy=2 ; Allow remote operation

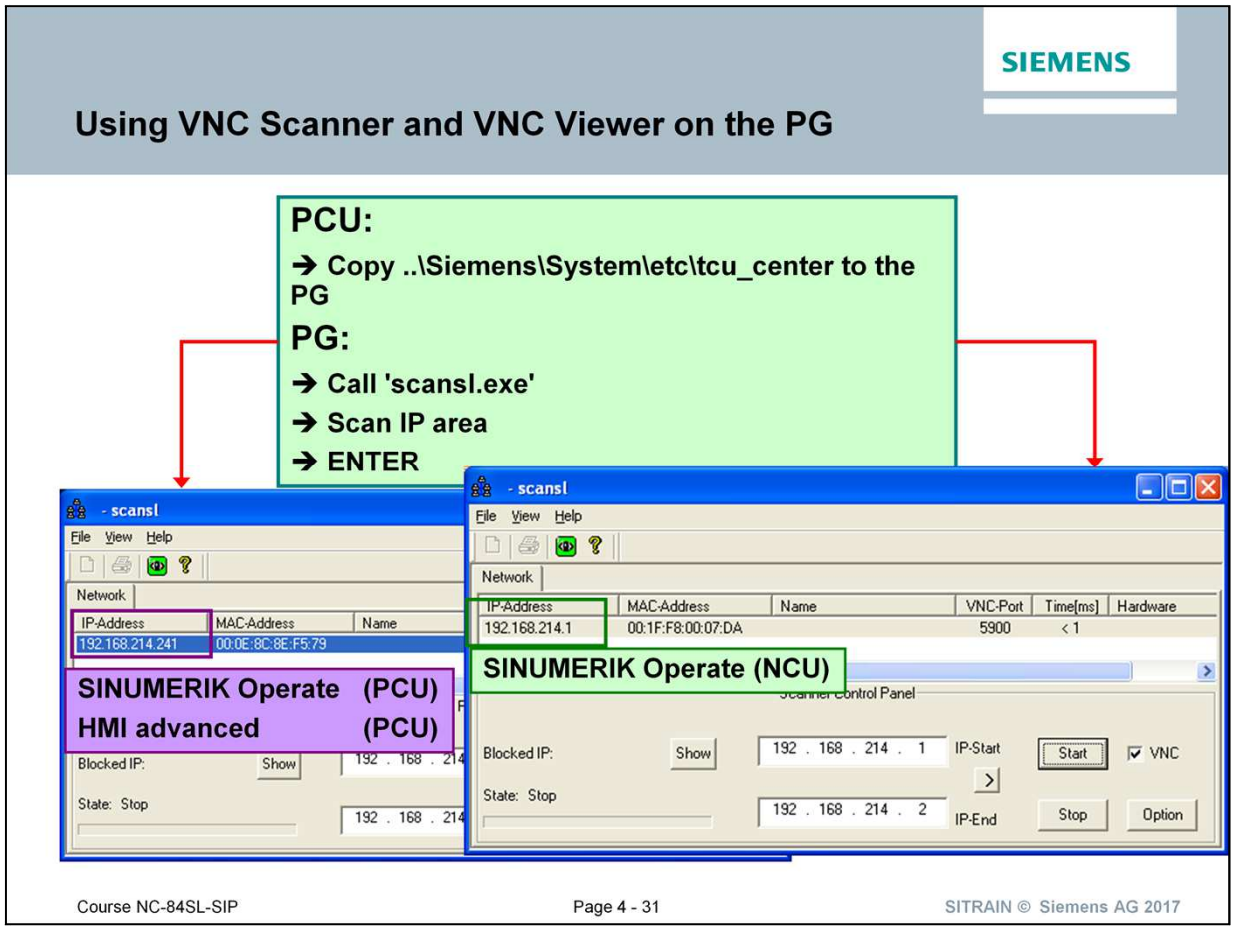


Hardware assembly

If, for technical reasons, a PCU 50 is not mounted on the rear of an OP but in the control cabinet, for example (to avoid vibration, for example) or the assembly of a slimline operator panel would take up too much overall depth, a TCU (Thin Client Unit) is used as the display module.
The TCU is mounted on the rear of the OP. It provides a remote operator control and visualization interface via VNC (Virtual Network Computing). The VNC server takes the form of a system integrated in the NCU which forwards the HMI Embedded user interface to the OP, or a VNC server which runs on the PCU 50 and forwards its picture content.

In a configuration, both the NCU and the PCU add a DHCP server, a VNC server, an HMI system, etc. to the network. However, for some components, this duality is not required, advisable, or even permitted.

So that the end user is not forced to make a selection, various settings must be made.



tcu_center

The VNC Scanner (scansl.exe) and VNC Viewer (vncviewer.exe) tools are available on the PCU on the following path:

C:\\Program files\Siemens\MotionControl

\siemens\system\etc\tcu_center

Once the entire directory has been copied to the PG, the Viewer can be started with scansl as follows:

- Select address space, e.g.: IP = 192.168.214.1 ... 2
- Press the **Start button**
- Press the **ENTER key**

Depending upon the selected IP address, you will see:

IP = 192.168.214.1 SINUMERIK Operate on NCU

IP = 192.168.214.241 SINUMERIK Operate (or HMI advanced) on PCU

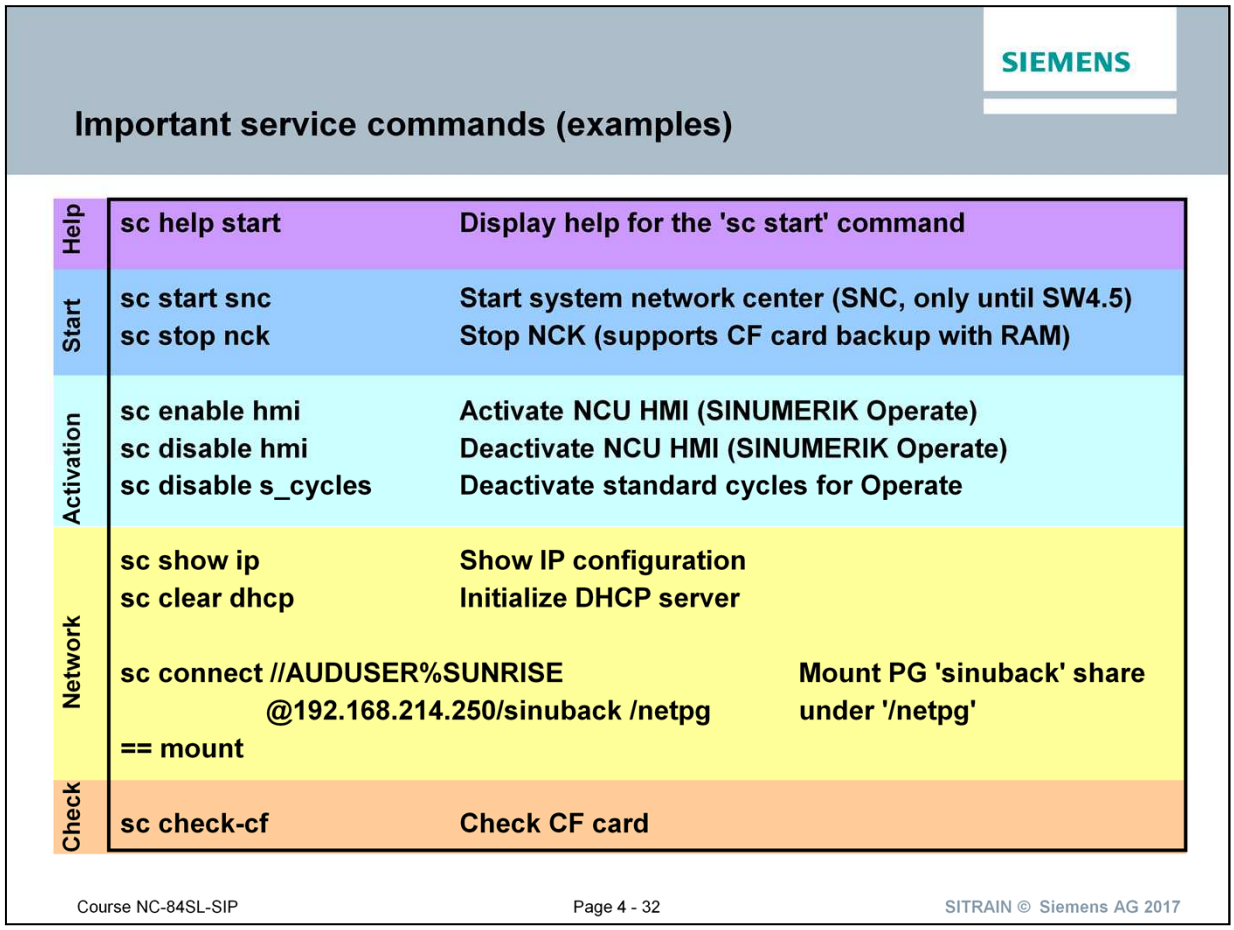
Operation

For remote control, the operator authorization must be changed over in the NCU/PCU SNCs:

- ➔ **SNC (NCU) or SNC (PCU)**
- ➔ '**TCU mode**' tab
- ➔ External Viewer security policy **= Administrator mode**

Note

If the scanner only displays 3 IP bytes, check the screen setting of your PC and reduce the DPI number to a smaller display.



Service commands

The service commands are a simple service interface for the Linux command processor. They are entered in the command shell.

sc <Parameter>

where <Parameter> represents one or a number of parameters.

Examples

- **sc help start** Display help for the 'sc start' command.
- **sc stop nck** Stop NCK (supports CF card backup including SRAM).
- **sc enable hmi** Activate HMI on NCU (Operate) **important for SINUMERIK Operate (NCU)!**
- **sc enable s_cycles** Activate standard cycles for Operate **for SINUMERIK Operate (NCU)!**
- **sc disable hmi** Deactivate HMI on NCU (Operate). **important for HMI on PCU!**
- **sc disable s_cycles** Deactivate standard cycles for Operate **important for HMI-Advanced (PCU)!**
- **sc show ip** Show IP configuration.
- **sc clear dhcp** Initialize DHCP server.
- **sc connect //AUDUSER%SUNRISE@192.168.214.250/sinuback /netpg**

  Mount PG 'sinuback' share (IP=192.168.214.250) under mount point '/netpg'.
  The user is 'AUDUSER' and the password is 'SUNRISE'.

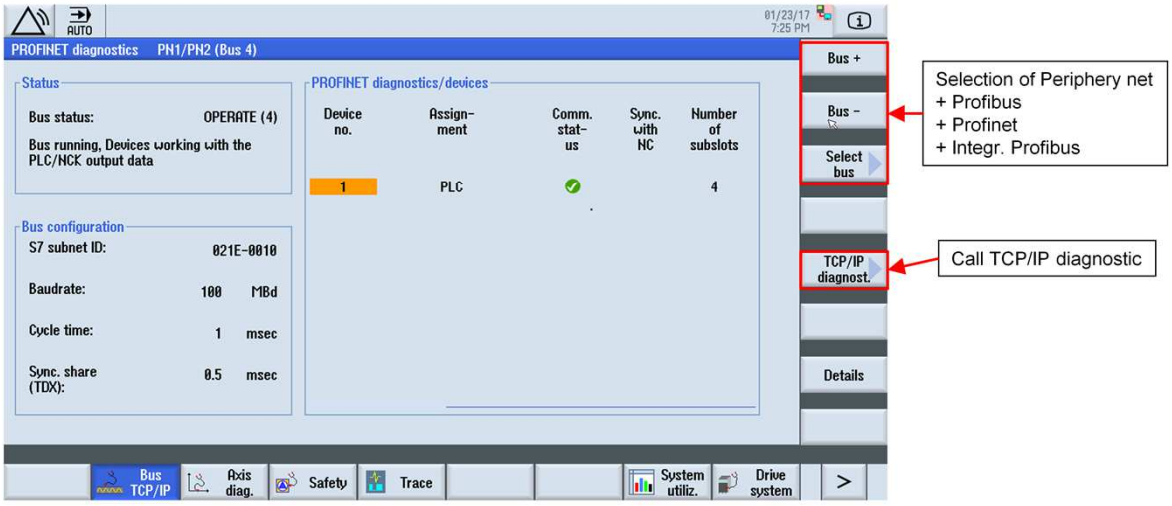
  This corresponds to the 'Map network drive' function in the Explorer. The only difference is that Linux uses a 'mount point' (=directory) instead of a drive.

  This mount point must be generated first with **mkdir /netpg**.
- **sc check-cf** Check CF card (check-cf written together as **one** word).

Further information

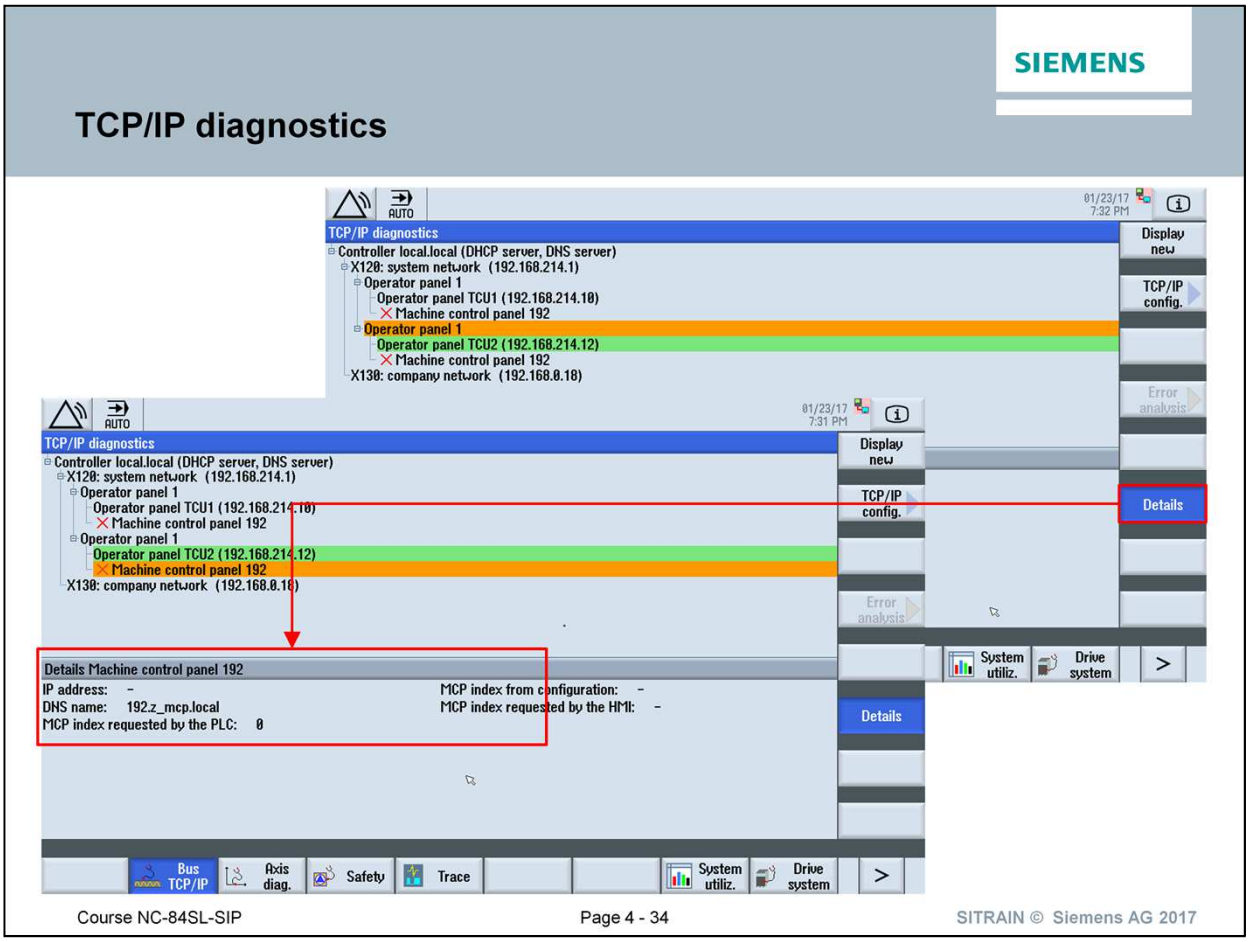
DocOnCD ➔ 'Commissioning Manual Base SW'. ➔ 'NCU operating system (IM7)'.

Bus TCP/IP diagnostics in operating Area diagnostics



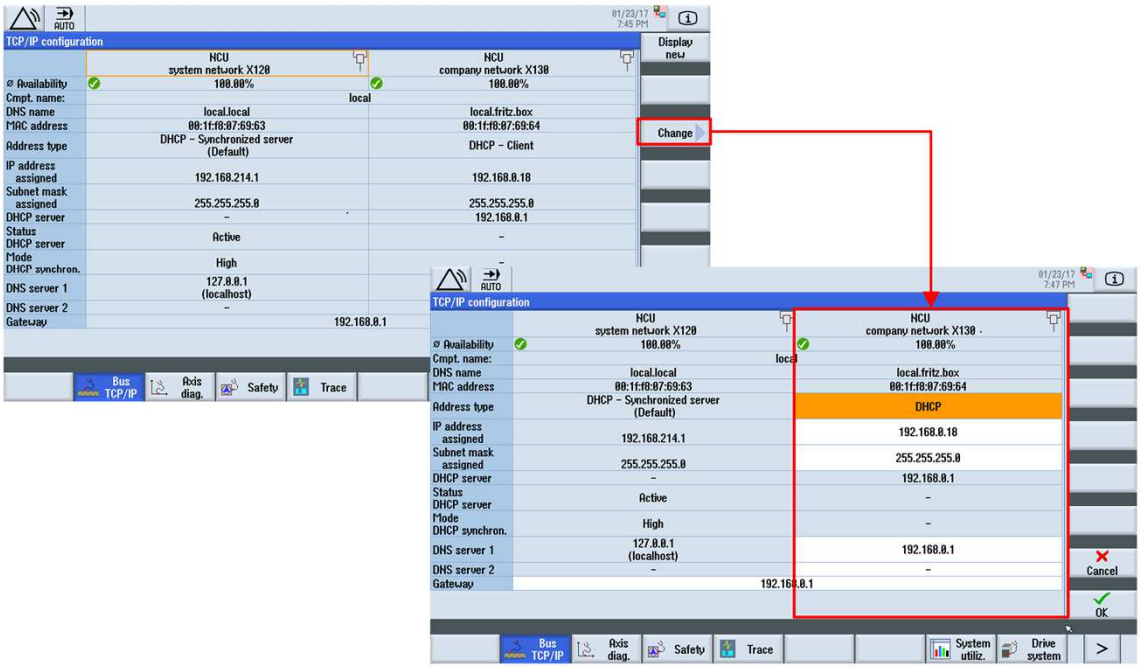
SITRAIN © Siemens AG 2017

In the operating area diagnostics you got the softkey BUS TCP/IP within you have different diagnosis functions.
Specially interesting for Networking is the TCP/IP diagnostic.



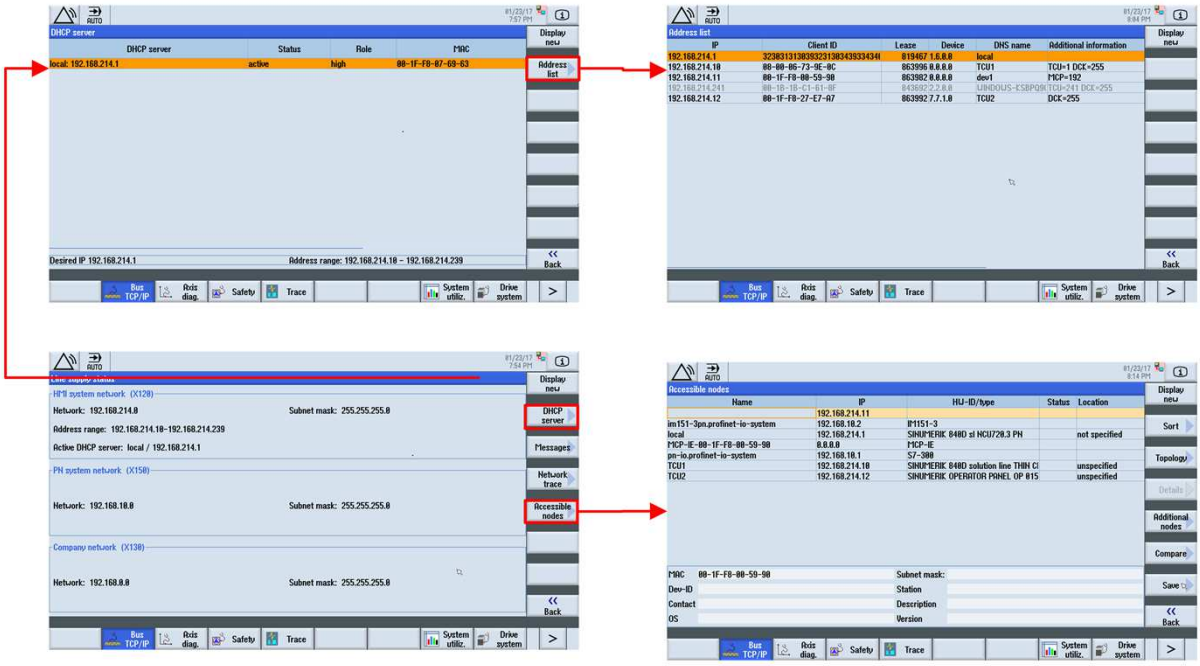
With the Softkey „Details“, you get further information about the network participants which are selected with the cursor.

TCP/IP configuration in operating area diagnostics



You can find the configuration of the machine and company network by using the softkey „TCP/IP configuration“.
With the „Change“ Softkey the settings of the company network can be configured.

TCP/IP configuration in operating area diagnostics



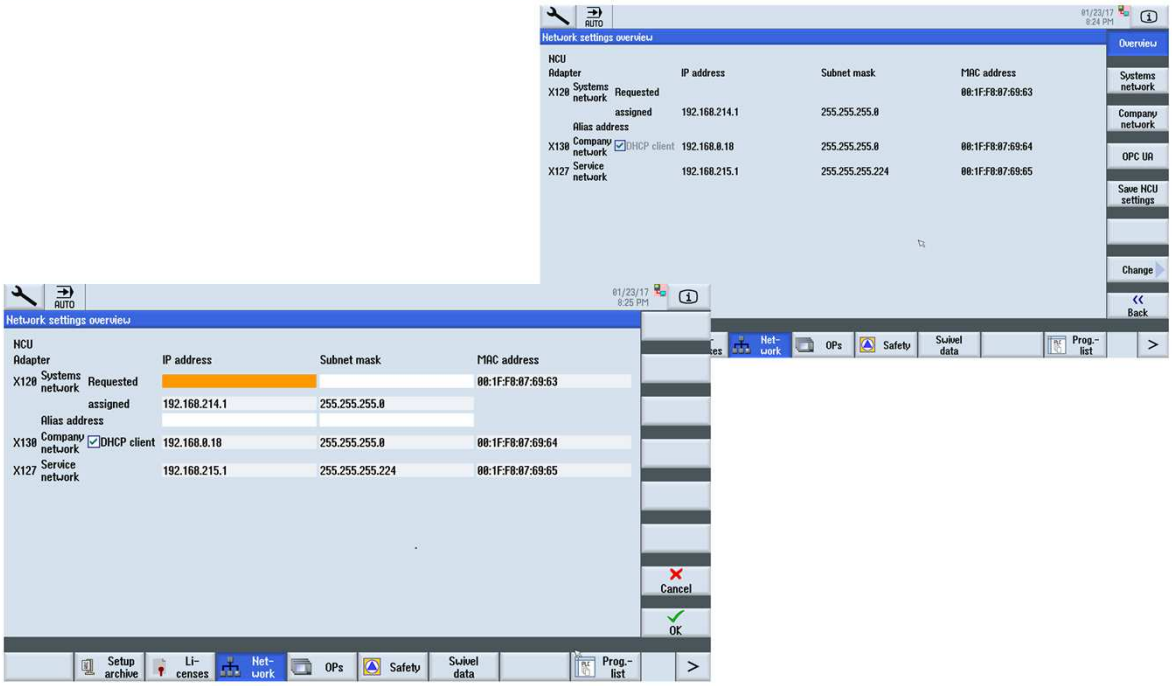
By pressing the softkey TCP/IP diagnostic you find the above displayed screens.

Here you gain access to the DHCP-server und further on to the lease list.

You can also browse additional nodes in the screen called by this softkey.

The Network trace is accessible only after entering an additional password which is „NETWORK“. Here you can record a network communication to a file which later could be analyzed with e.g. wireshark.

Network settings in operating area startup



You can find several network settings on the 2nd. Softkey level in the operating area startup.

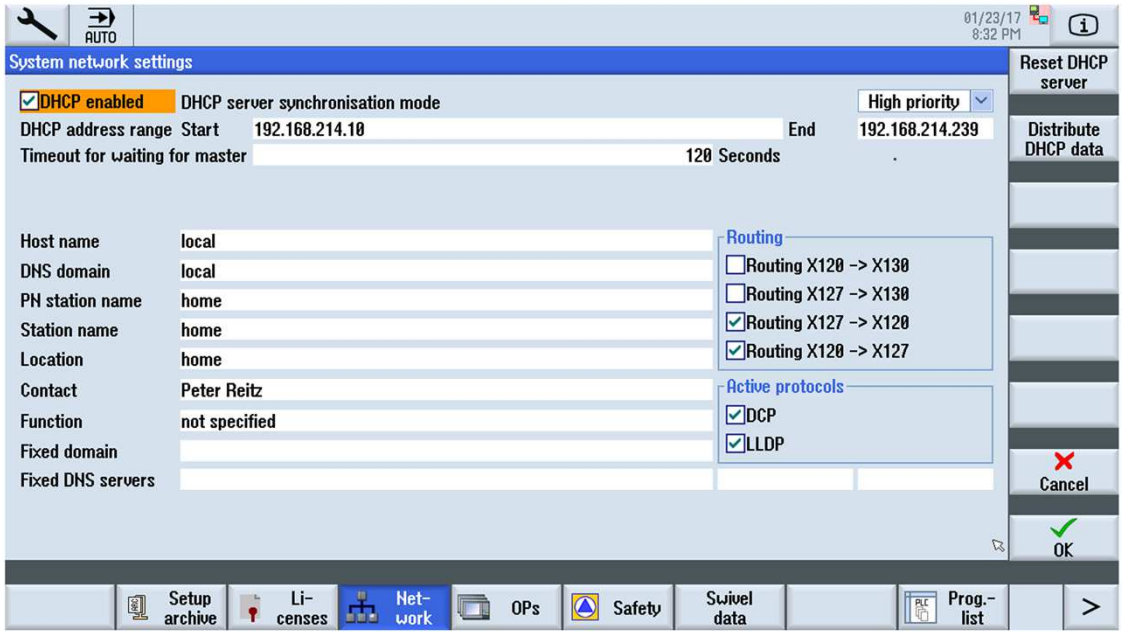
There are settings for the machine- company- and service-network.

You can configure the NCU IP address in the machine network for example.

For the company network it is possible to define whether the network node is addressed manually or by DHCP. Additional settings are e.g. for the NTP nodes.

If the option OPC/UA is activated you can do the OPC/UA server settings here.

System Network settings



 SITRAIN © Siemens AG 2017

In this screen you do the settings for the DHCP Server on X120. This includes the setting for the DHCP range and the server priority. The server itself has a static IP-address.

If the master server fails the next lower priority server becomes active after a configured waiting time and provides IP-addresses..

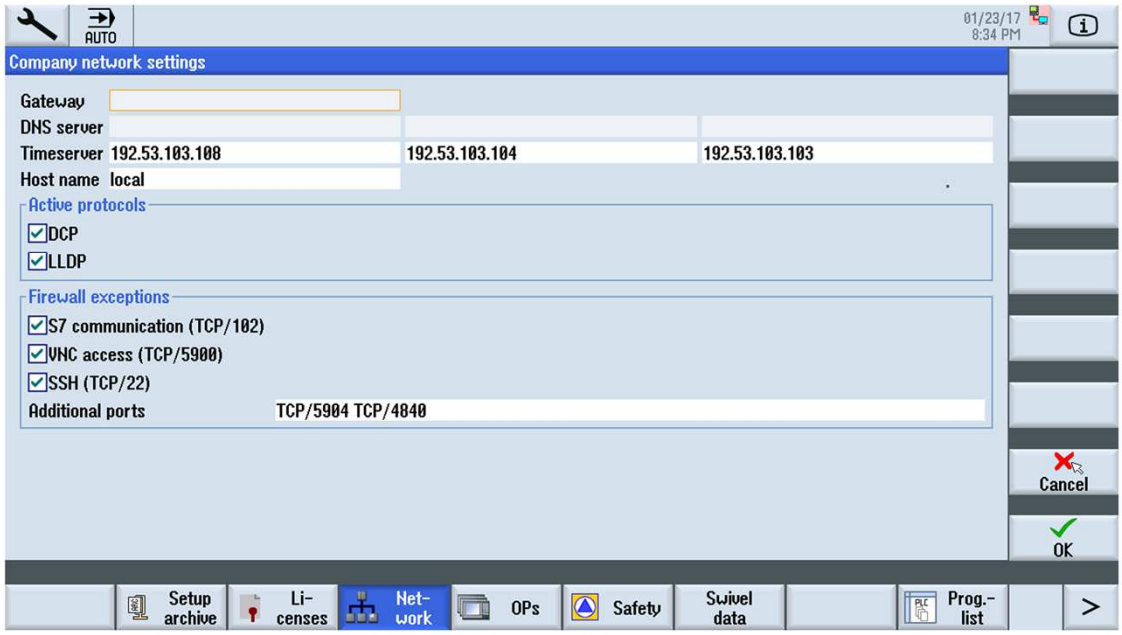
Several name settings can be made like the hostname, DNS-name PN-station name, routing configuration in between the interfaces.

Active protocols

The **Dynamic Configuration Protocol** (DCP) has the task to provide addresses and names to the profinet participants.

The **Link Layer Discovery Protocol** (LLDP**)** has the task to identify the devices in the network neighborhood.

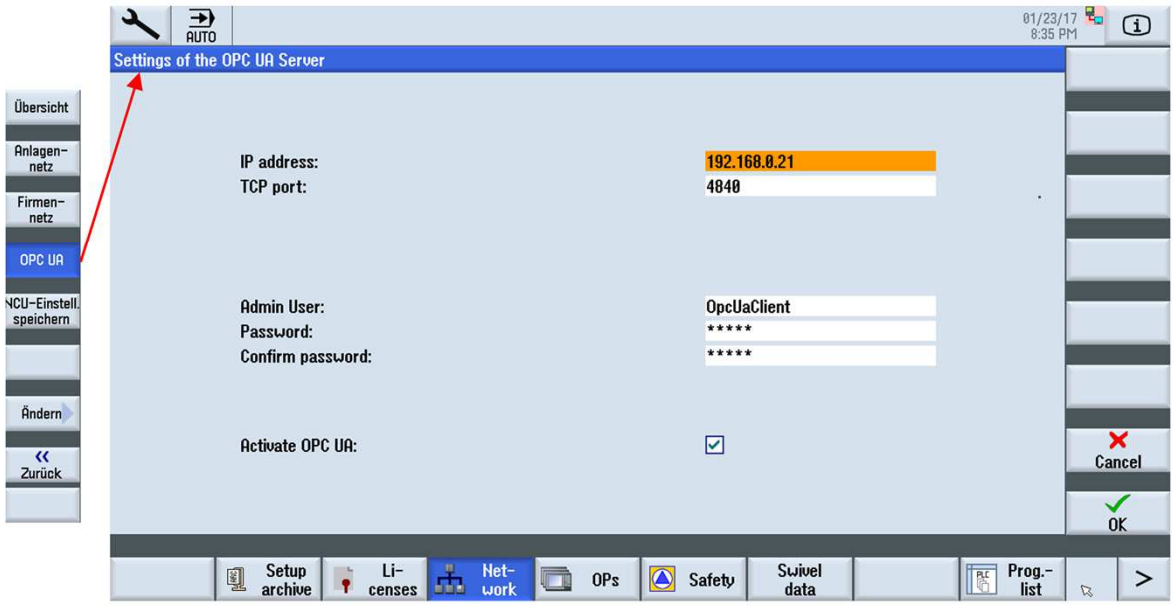
Company network settings



SITRAIN © Siemens AG 2017

The company network settings include configurations for the time system (NTP-servers, gateway, DNS- und timeserver.
Also firewall exceptions on X130 can be done here..

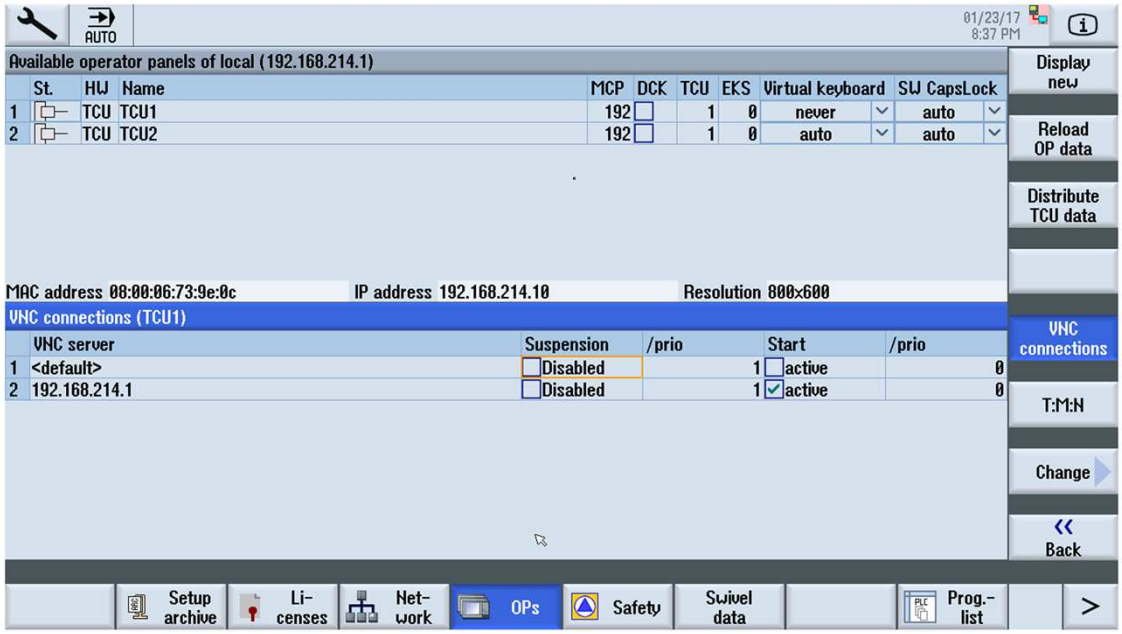
OPC UA server settings



 SITRAIN © Siemens AG 2017

OPC Unified Architecture,
short form **OPC UA**, is an industrial M2M-Communication protocol.
As a new OPC-specification of the *OPC Foundation,* OPC UA can access machine data and describe them semantically that they are machine readable.
API‘s are available for different programming languages like Java, C++, Ansi C etc .

Operator panel configuration



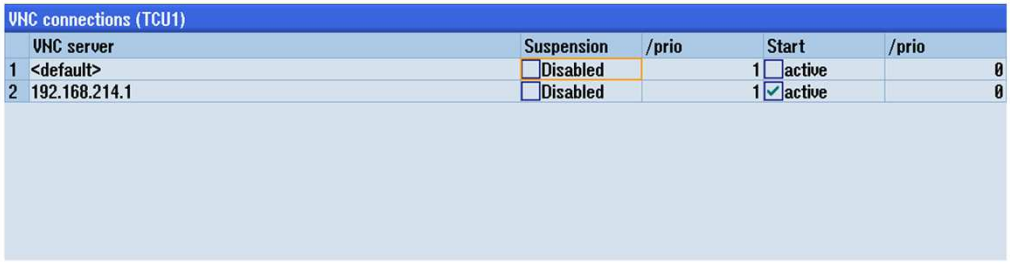
 SITRAIN © Siemens AG 2017

In the section „available operator panels“ you get a list of accessible operator panels.
A line specifies one Operator Panel and contains following informations:

Column	Description
	Consecutive number to identify the operator panel
Status	Status of the operator panel: : The operator panel is available on the bus. : The operator panel is not available on the bus.
HW	Information on the hardware of the operator panel Possible values are unknown (no information available), NCU, PCU, PPU, TCU (all TCU including HT 8, etc.). Default value: Unknown
Name	Name of the operator panel
MCP	DIP switch position for the machine control panel The values can be freely selected from 1 to 254 (0 means that the last active machine control panel is retained, 255 means that a machine control panel is not being used). Default value: 192
DCK	Availability of direct control keys Active: The TCU has direct control keys Inactive: The TCU has no direct control keys
TCU	Index for the TCU (Thin Client Unit) The value is freely configurable from 0 to 255 and is used for identification in the PLC.
EKS	DIP switch position for the chip reader for access levels (Electronic Key System) Possible values range from 1 to 255. The value 0 means that an EKS is not used.
Virtual keyboard	Use of a virtual keyboard Possible values are "Never", "Always" and "Auto", whereby "Auto" is set as default value.

Hint These information are read only and can‘t be changed in this screen.

VNC – configure connections



VNC connections (TCU1)

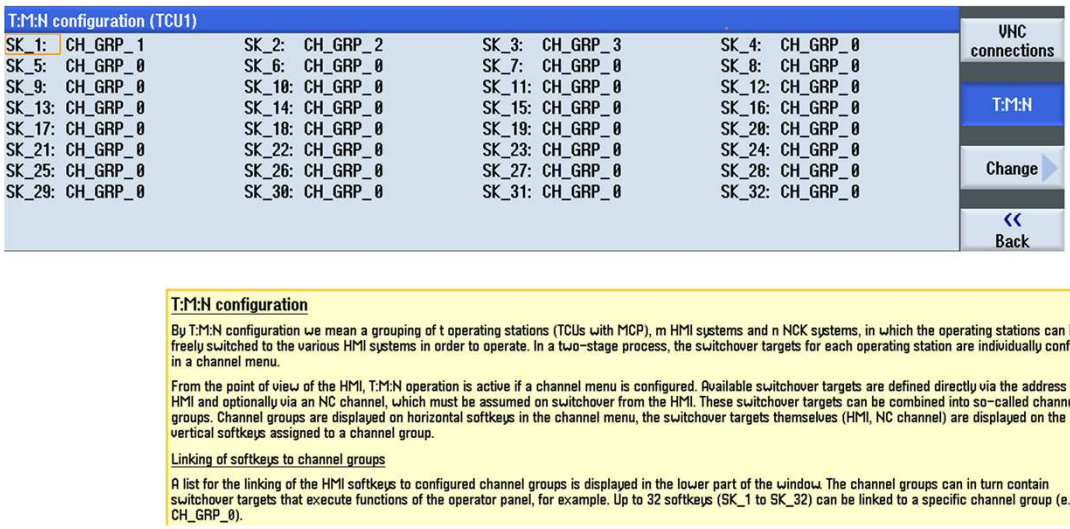
	VNC server	Suspension	/prio	Start	/prio
1	<default>	Disabled	1	active	0
2	192.168.214.1	Disabled	1	active	0

Column	Description
	Consecutive number to identify the VNC connection
VNC server	IP address or name of the NC
Displacement	Specifies whether it is possible that the TCU will be displaced by other nodes.
/Priority	Displacement priority: Specifies the priority which influences whether the TCU will be displaced by other nodes. The possible values lie between 0 and 10. The smaller the value, the faster the TCU will be displaced. If the displacement is blocked, the priority will be ignored.
Start	Specifies whether the TCU is to be started active, i.e. as operator panel.
/Priority	Start-up priority: Specifies the priority of the TCU connection to be used. Any value greater than 0 is permitted. The larger the value, the lower the connection priority
Display area	Specifies which HMI screen is displayed when activating the TCU.
Channel	Specifies which channel is displayed when activating the TCU.

SITRAIN © Siemens AG 2017

In the lower part of the available panels screen you get a list of VNC connections displayed.

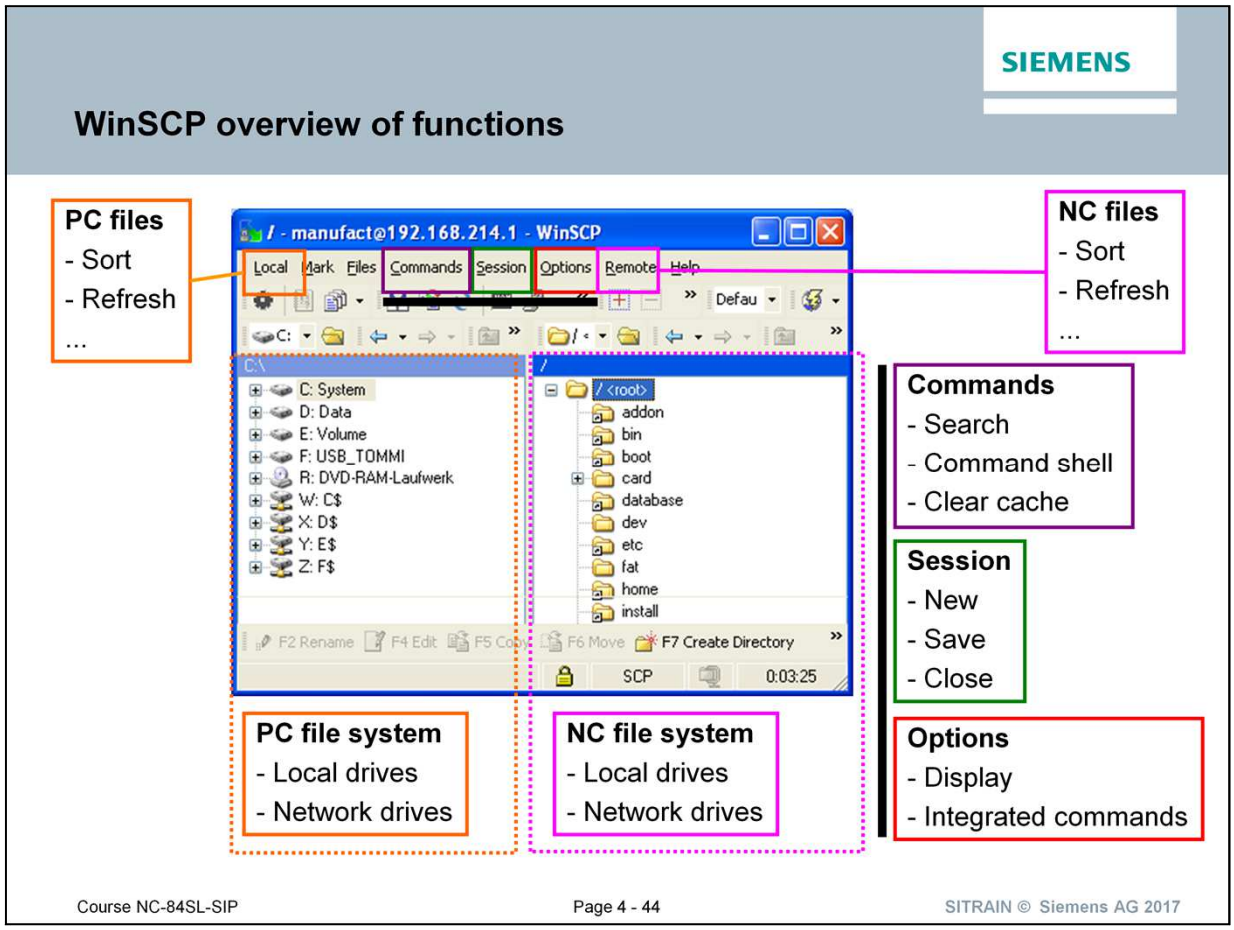
T:M:N Configuration



SITRAIN © Siemens AG 2017

T:M:N

A T:M:N-Configuration is a group of **t** Operator stations (TCU and MCP), **m** HMI-systems and **n** NCK-systems where each Panel can be connected to each HMI system during operation.

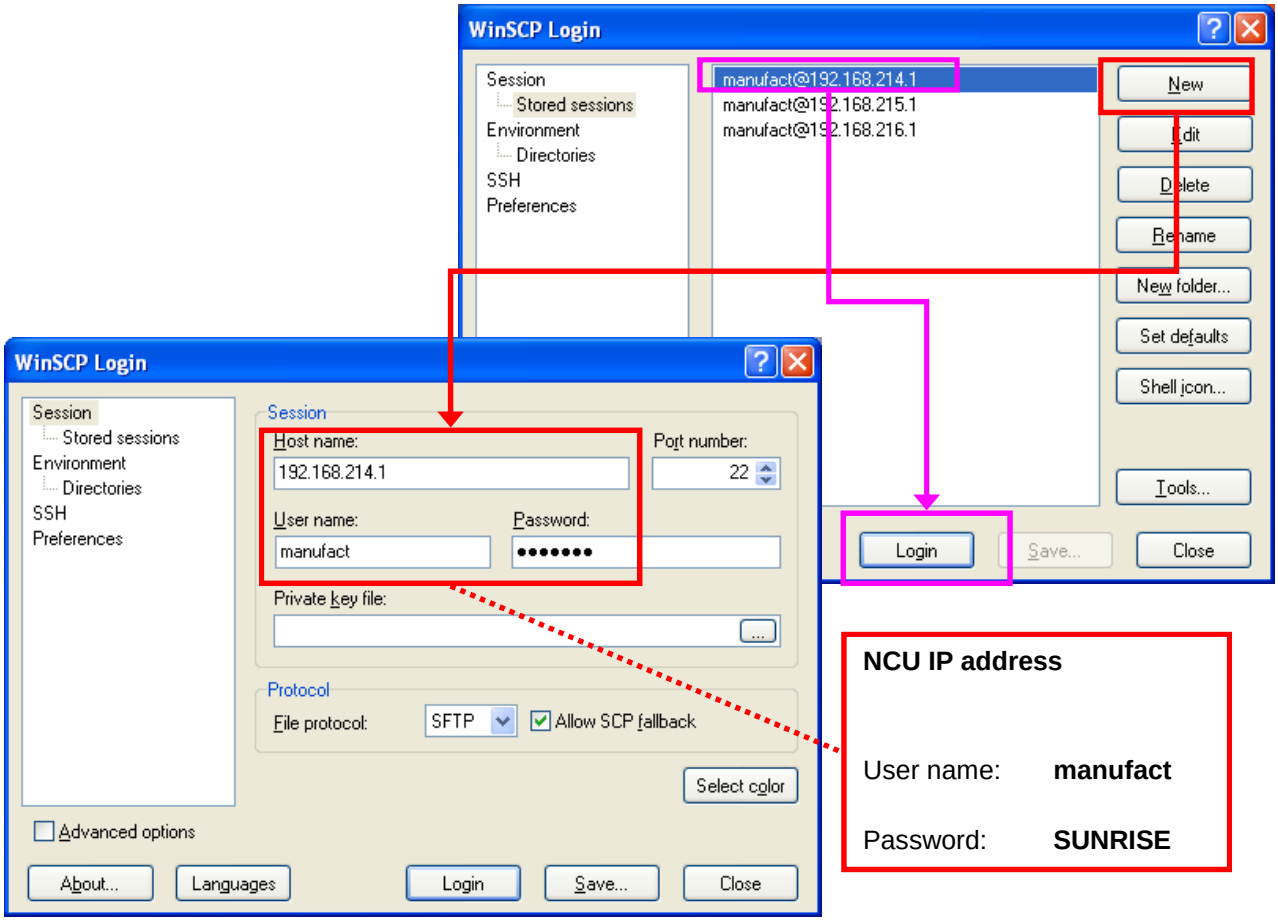


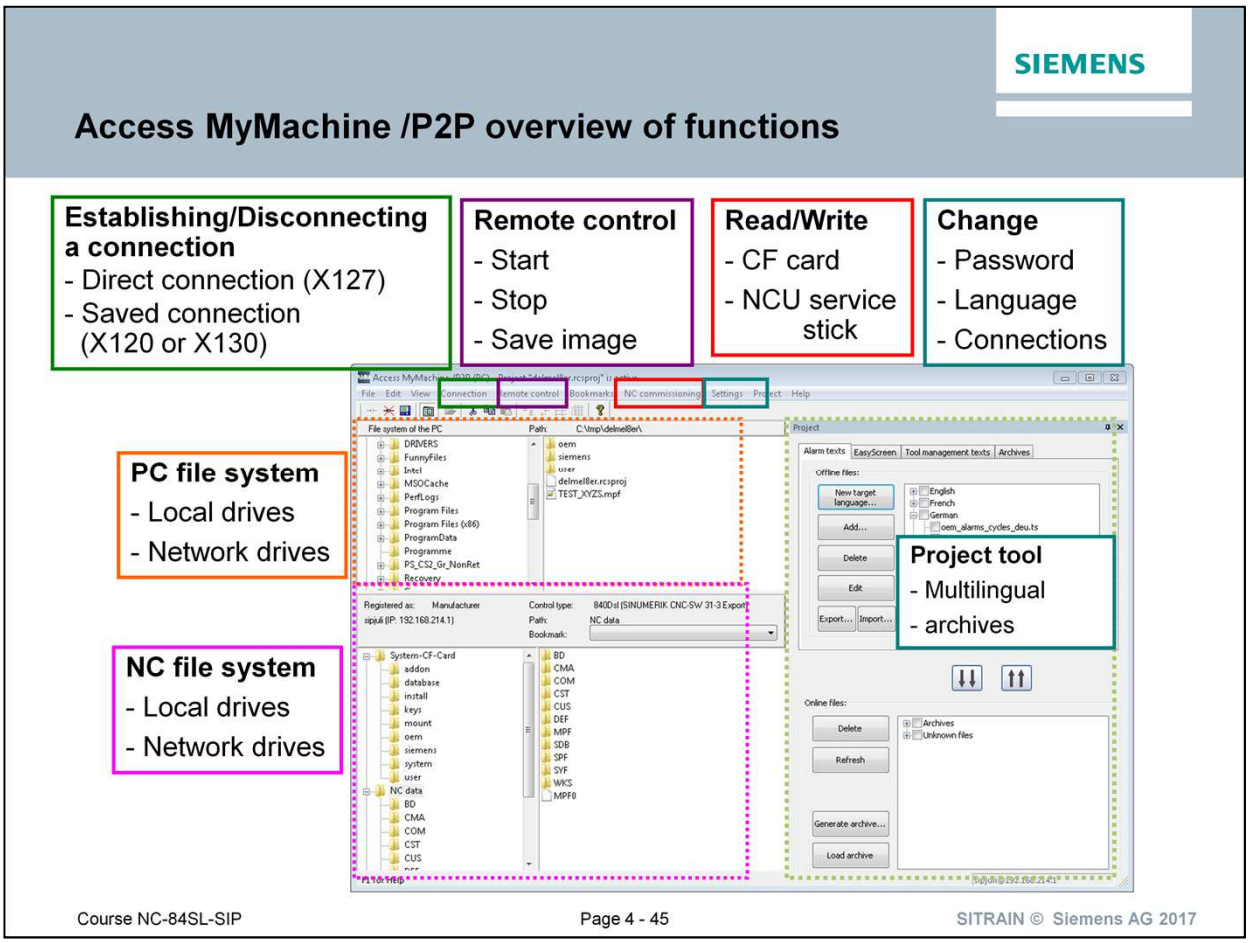
WinSCP

- Login on NC

Here is a brief reminder of how to log in:

Either select saved connection, **'Login' button,**
or press the '**New**' button:





AMM

- Login on NC

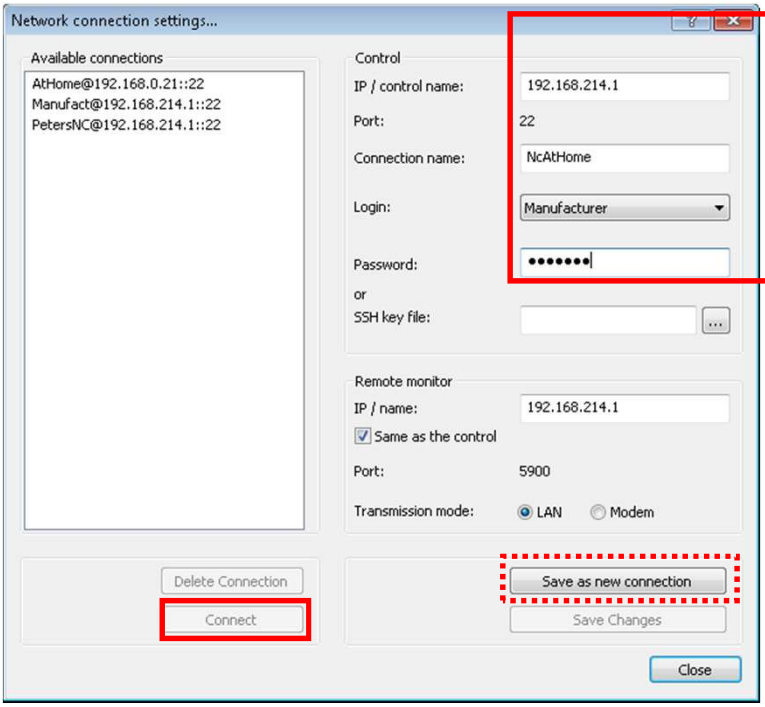
Here is a brief reminder of how to log in:

Either connect on start-up; example:

840Dsl_X120@192.168.214.1 (saved connection '840Dsl_X120')

or

Connection ➔ Connect ➔ New network connection



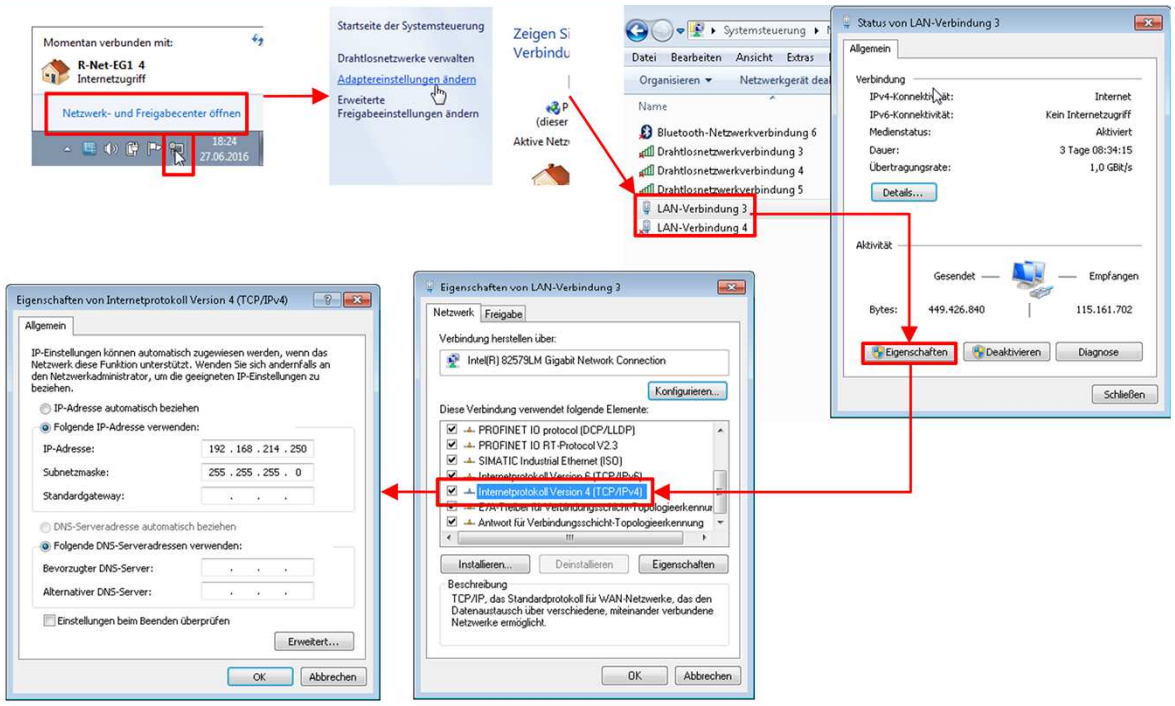
NCU IP address

Connection name

Login: **Manufacturer**

Password: **SUNRISE**

Exercise:
Check the TCP/IP settings of the Windows 7 PG



Course NC-84SL-SIP Page 4 - 46 SITRAIN © Siemens AG 2017

Objective

Prepare PG for backup/restore of the PCU with Ghost.

Setting dialog

Search for the following setting:

- ➔ **Network (task bar) ➔ left-click**
- ➔ **Network and sharing center ➔ Adapter settings**
- ➔ **LAN connection ➔ left- click ➔ Properties**
- ➔ **Internet protocol (TCP/IP) V4**
- ➔ **Key 'Properties'**

Check that a fixed IP address is entered here:

- PG IP address

IP address = **192 . 168 . 214 . 250** (feasible: .250 .. 254), and
subnet mask = **255 . 255 . 255 . 0**

Computer name

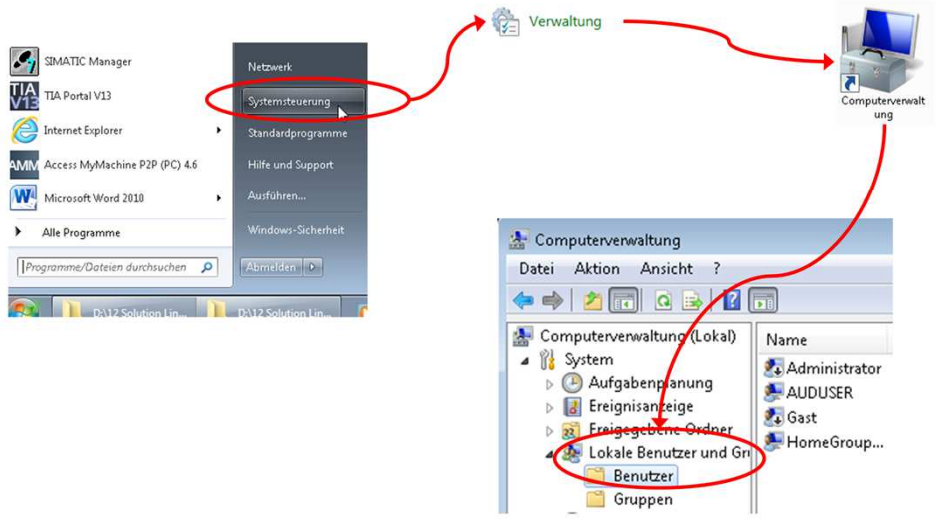
Locate the computer name as described below and make a note of it.

- ➔ **Workspace -> right-click (shortcut menu)**
- ➔ **Properties -> 'Computer name' tab**

The correct name is the one under '(Full) computer name'
and not the one under 'Computer description'!

Exercise:
Check AUDUSER user on PG

SIEMENS



 SITRAIN © Siemens AG 2017

Objective Check that AUDUSER on the PG is included in the 'Administrators' group.

Computer administration

Can be found in Windows XP under

➔ **Start ➔ Control Panel ➔ Administrative Tools**
➔ **Computer Management.**

User administration They are separated into two folders

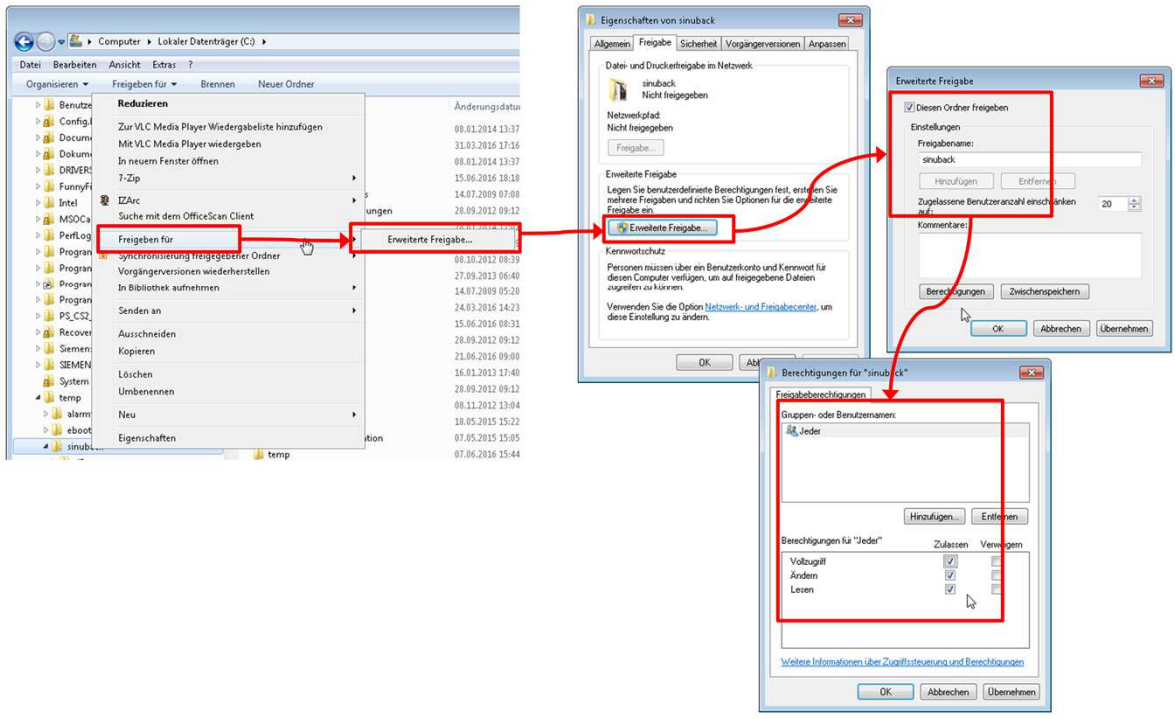
➔ **Users** and

➔ **Groups**.

Result On the PG you have a user called AUDUSER, who belongs to the 'Administrators' group.

The password settings are:
permanent password SUNRISE (password never expires and user cannot change password).

Exercise:
Share the SINUBACK folder on PG



Share on PG

For a folder like Sinuback, you can access this option in the shortcut menu under

➔ **Share with**

There, you should select

➔ **Advanced Sharing / Share**

and under

➔ **Permissions**

check or set the following:

- **Administrators** have access as a group
- **Full Control** is set (more precisely: Change, Read, Full Control)

Not simple file sharing

If these settings are not available,

make the following settings:

➔ **Explorer ➔ Organize ➔ Folder and search options ➔ View**
➔ **Use sharing wizard = not selected**

Share on PCU

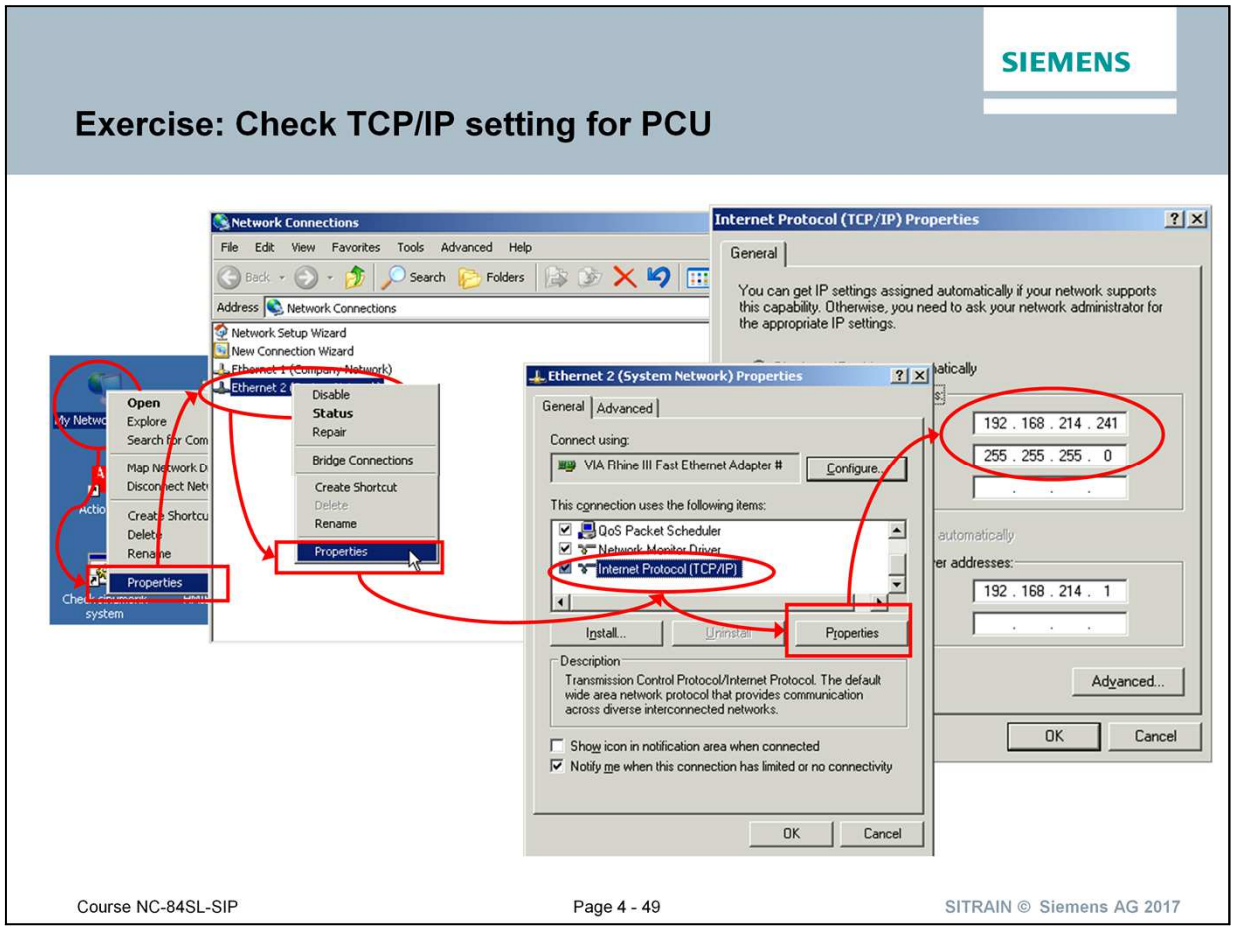
Although all drives C$ and D$: are shared by default,
there are problems with access from the PG due to the UAC.

This can be corrected with a registry entry.

HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows\CurrentVersion\Policies\System

Create a new 32-bit Dword with the name

LocalAccountTokenFilterPolicy
and set its value to 1. Access to the administrative shares should now be possible.



Objective

Check that the TCP/IP address setting of the PCU is suitable for backup/restore.

Easiest option for PCU

Use the following option, which is the easiest:

Connection with Ethernet 2 with fixed IP address on the PCU

Setting dialog

Proceed as follows to make the settings:

- ➔ **My Network Places (on the desktop) ➔ Right-click (=shortcut menu)**
- ➔ **Properties**
- ➔ **Ethernet 2 ➔ Right-click ➔ Properties ➔ 'General' tab**
- ➔ **Internet Protocol (TCP/IP)**
- ➔ **'Properties' button**

Check the following here:

PCU IP address Ethernet 2

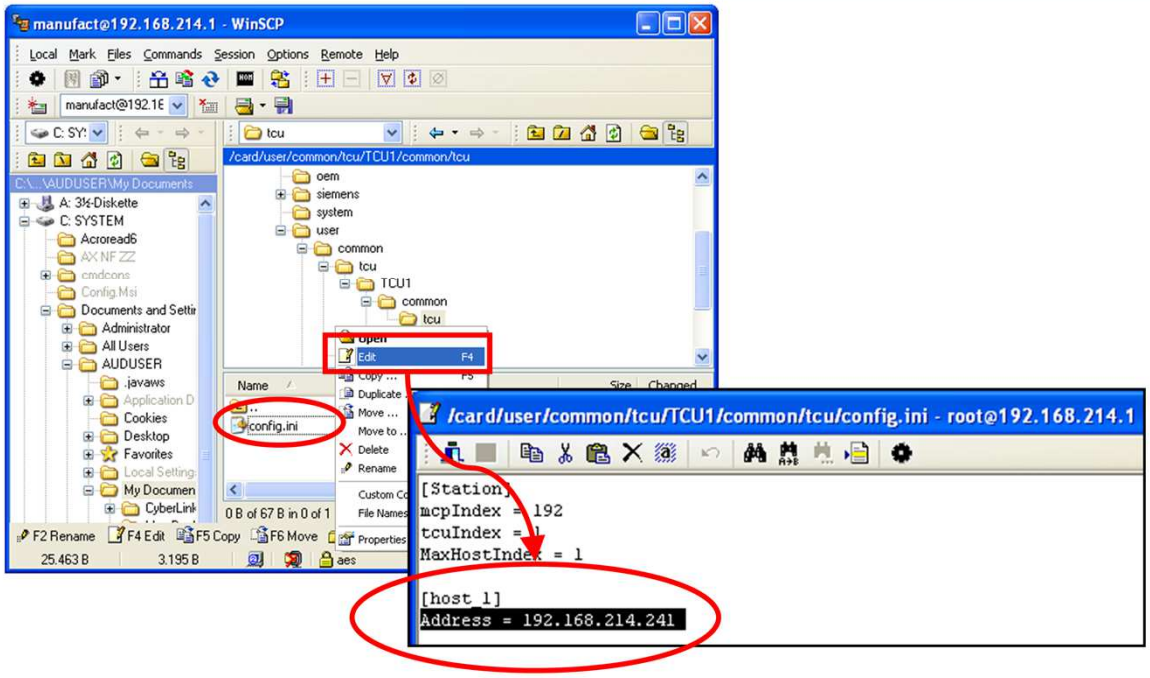
IP address = **192 . 168 . 214 . 241**for port Ethernet2

Subnet mask = **255 . 255 . 255 . 0**

Advantage

As this is the default setting, you no not need to make any changes.

Exercise: Set the default VNC server for an NCU+PCU+TCU configuration



Course NC-84SL-SIP Page 4 - 50 SITRAIN © Siemens AG 2017

Objective

The default VNC server is to be set for an NCU+PCU+TCU configuration. HMI-Advanced on the PCU must be displayed on the TCU automatically (without operator intervention) after run-up.

Requirements

The TCU is already registered with the NCU:
(the directory /user/common/tcu/TCU1 has been created).

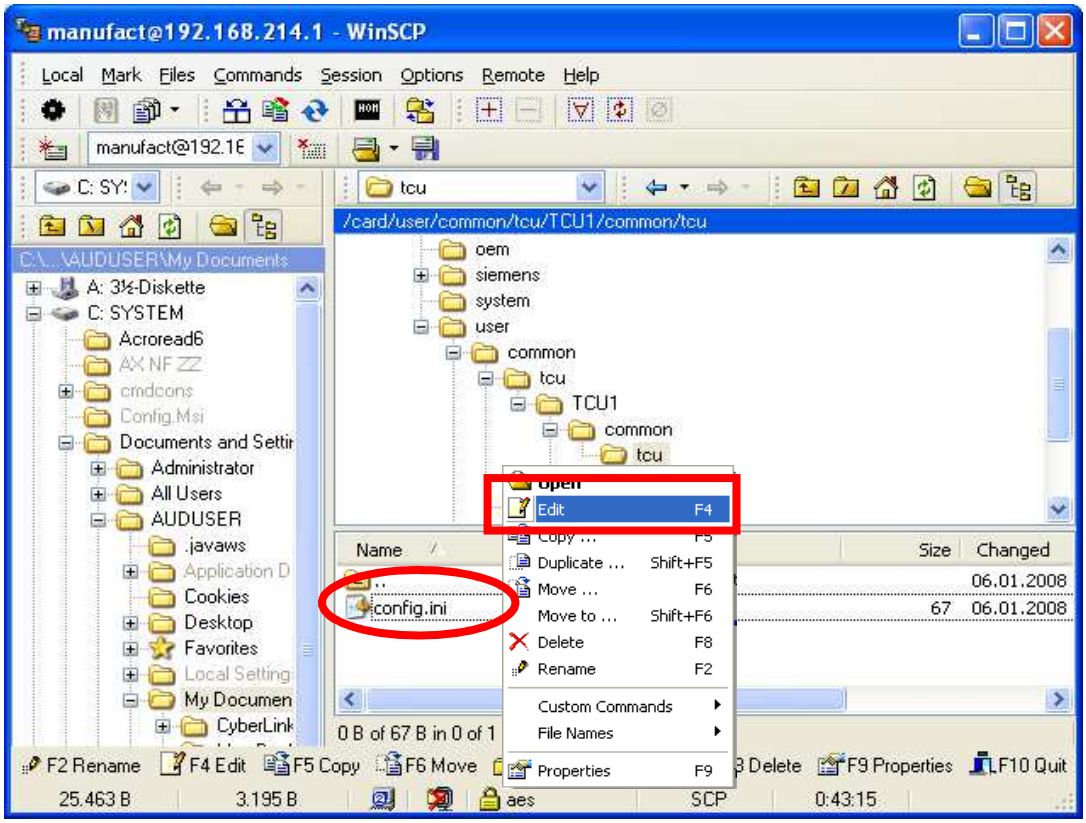
Steps

1. NCU: Open config.ini
2. config.ini : Enter address
3. Reboot NCU+PCU+TCU

1. NCU: Use WinSCP to connect to the control.

- Open Then open file

config.ini **/card/user/common/tcu/TCU1/common/tcu/config.ini**



2. config.ini: Check the following entries:

- Address entry

[Station]

mcpIndex = 192

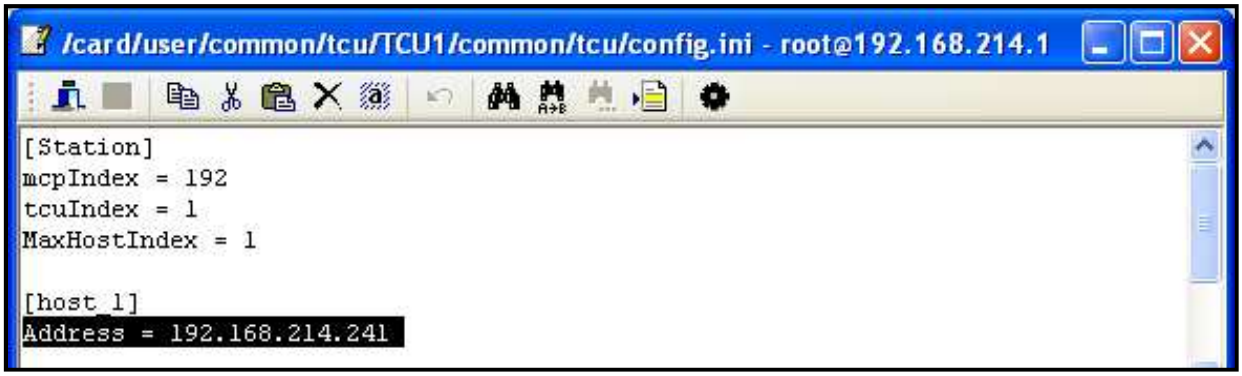
tcuIndex = 1

MaxHostIndex=1

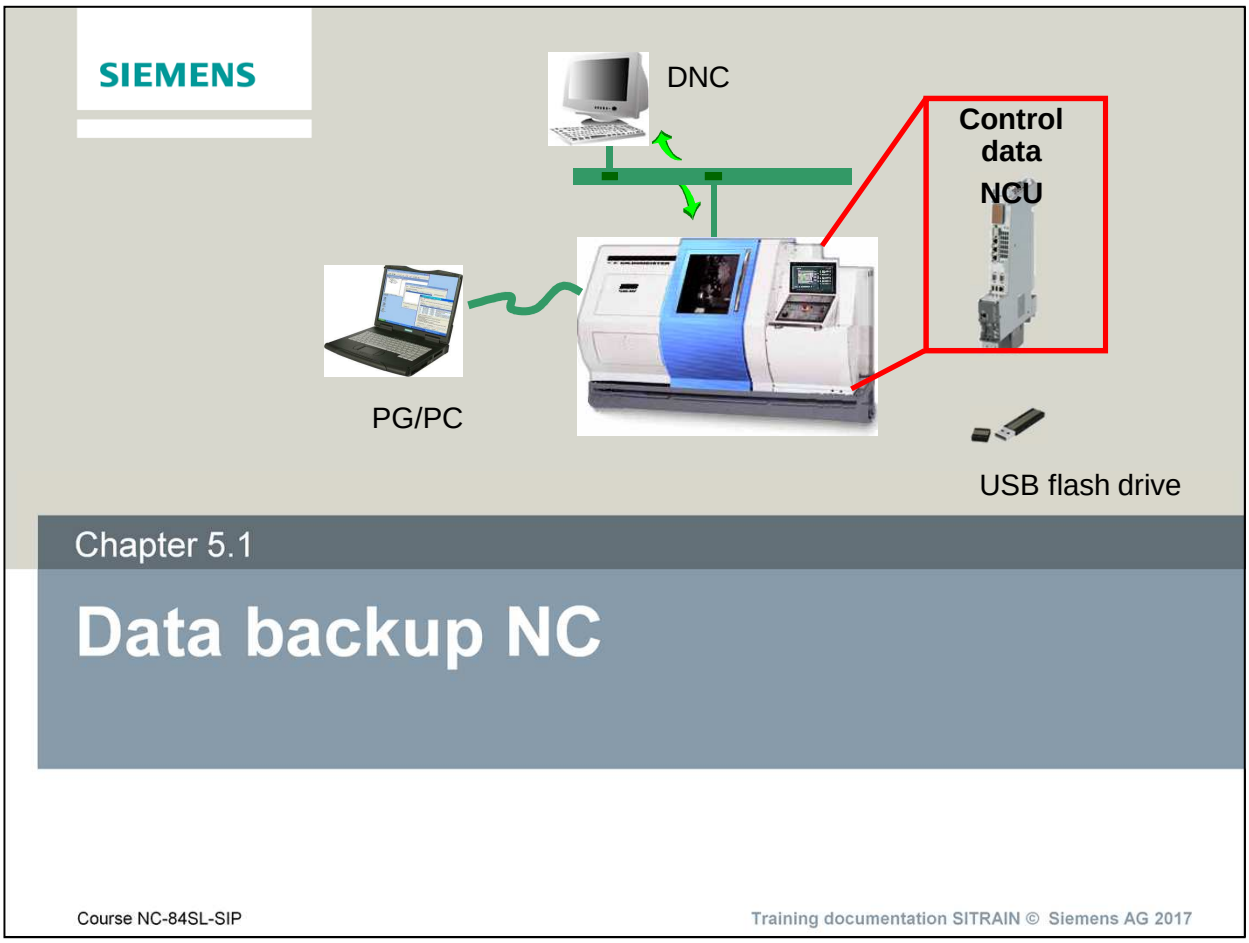
Create the entry:

[host_1]

Address = 192.168.214.241



3. Reboot NCU+PCU+TCU HMI-Advanced should now be displayed automatically after all the control components (NCU, PCU, and TCU) have rebooted.



Learning objectives 2
Protection levels 3
Data management 6
Archiving specific files 10
Punched-tape format 11
Machine data for data input/output 12
Copying and pasting files 13
Setup archive 14
HMI: Logical drives 16
Exercise: Create a setup archive 17
Exercise: Load SINAMICS factory setting 19
Exercise: General reset of the NC and PLC 20
Exercise: Import the setup archive 21
Exercise: Set up a network drive 23
Exercise: Archive a part program archive on a network drive 25

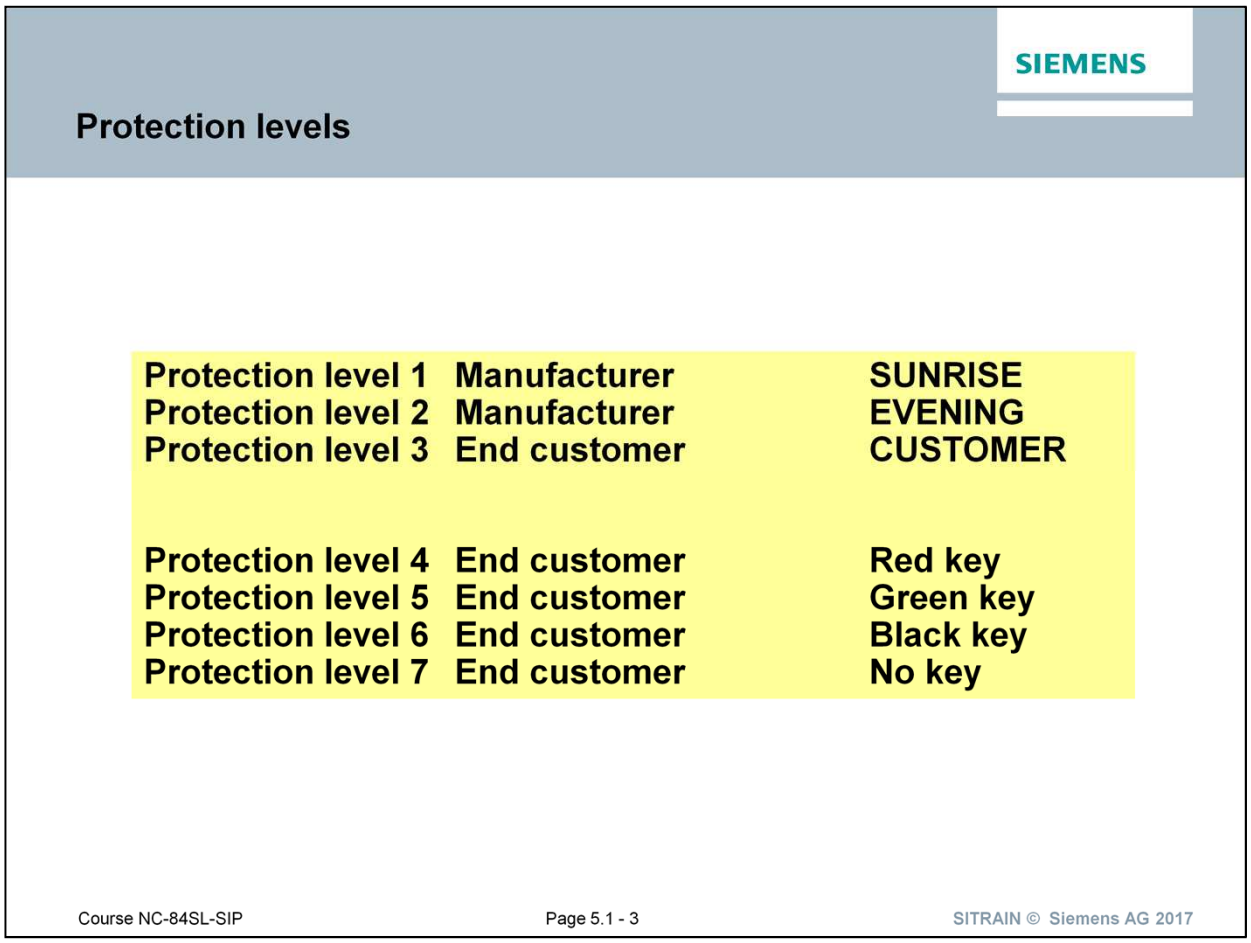
SIEMENS

Learning objectives

- You will be able to input and output data with SINUMERIK Operate
- You will be familiar with the protection level concept with password and key-operated switch
- You will be able to back up data and perform recommissioning with the setup archives



SITRAIN © Siemens AG 2017



Access rights

7 hierarchical protection levels are provided to protect access to programs, data, and functions in accordance with user-specific requirements. These are divided into:

- 3 password levels for Siemens, machine manufacturers and end users
- 4 key-operated switch positions for end users

Access properties

Protection level 1 has the highest access rights, protection level 7 has the lowest access rights.

The current protection level includes access authorization to the higher numbers.

Default passwords

The following passwords are set by default for protection levels 1 to 3:

- Protection level 1 SUNRISE
- Protection level 2 EVENING
- Protection level 3 CUSTOMER

Note

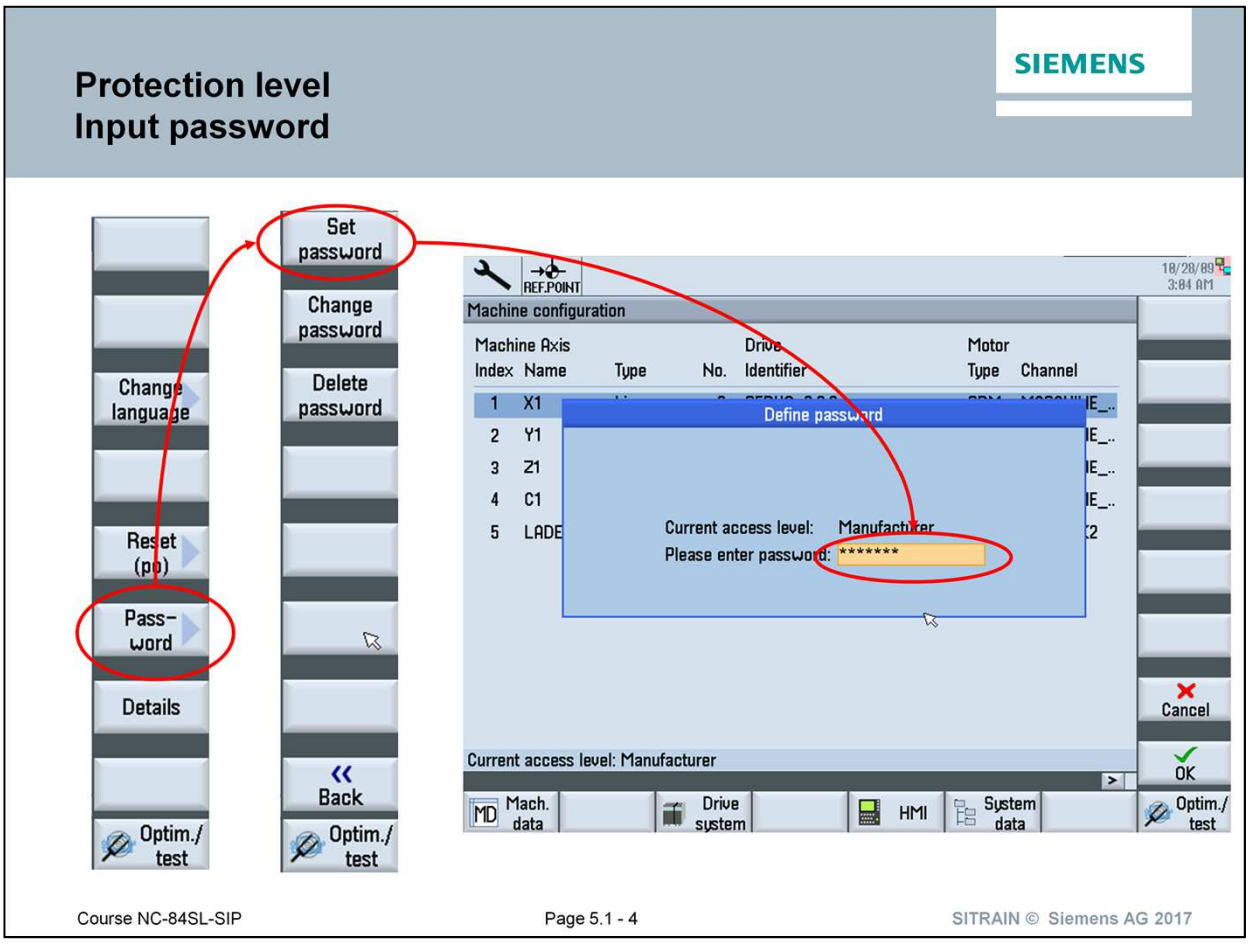
When the system boots in commissioning mode (NCK commissioning switch in position 1), these passwords are entered by default.

Further passwords

These are used for access to the Linux system and the VNC Viewer:

Linux (access with WinSCP)

User: **manufact**	Password: **SUNRISE**
VNC Viewer	Password: **password**



Softkey: Set password For the 3 possible password levels with their access permissions, the passwords can be entered in the "**Setup**" operating area by selecting the "**Set password**" softkey.

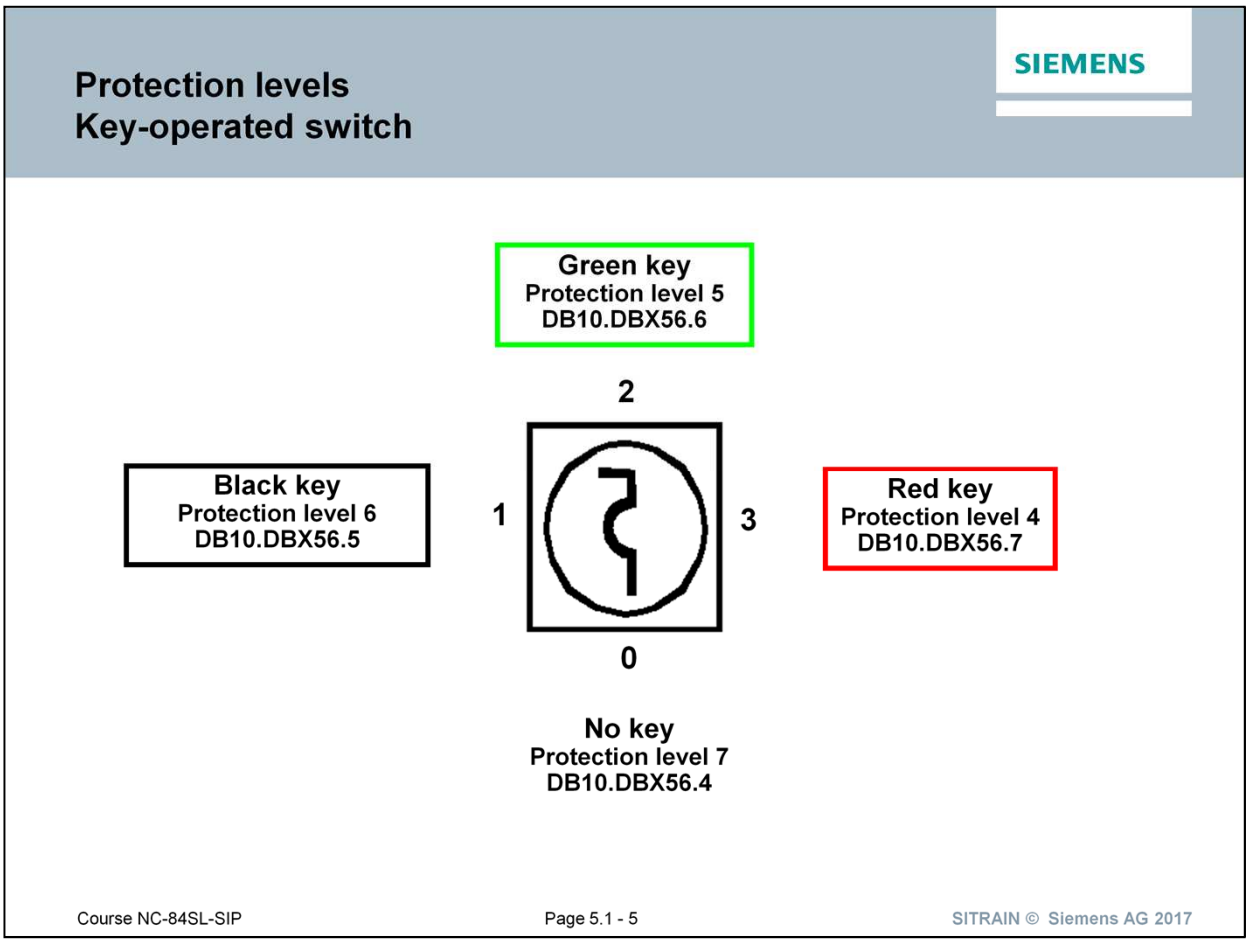
Softkey: Delete password

Please note that a password remains valid until access authorization is reset with the "**Delete password**" softkey. After service has been performed on the system, you need to press this softkey to delete the password.

Access authorization is not automatically deleted on POWER ON!

Softkey: Change password

The user can change the Siemens default passwords. Up to 8 characters can be used in the password. We recommend that you restrict yourself to the character set of the operator panel in selecting a password. Where a password consists of fewer than 8 characters, the additional characters are interpreted as blanks.



Key-operated switch

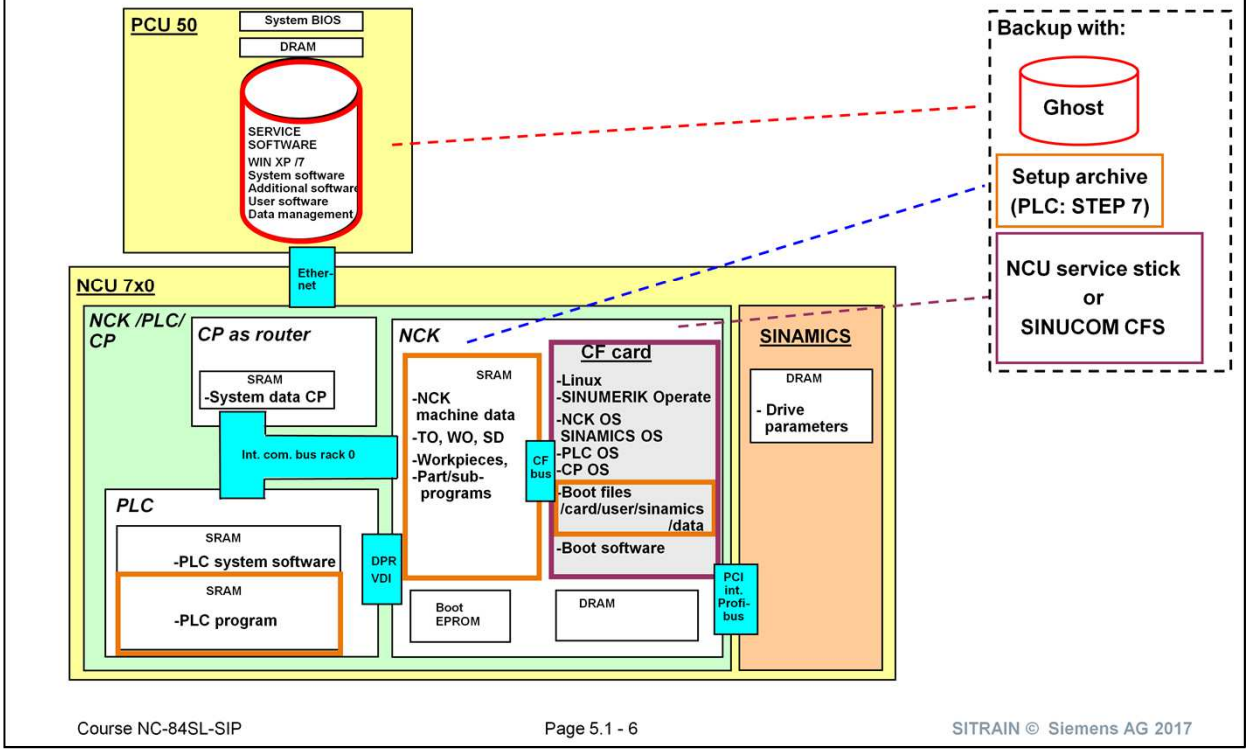
You can set the MCP key-operated switch to 4 positions. These positions are assigned to protection levels 4 to 7. Three different colored keys belong to the key-operated switch. The key positions can be assigned to functions by the machine tool manufacturer or end user. Using display machine data you can set different authorization levels for each user to access programs, data, and functions.

The key-operated switch positions are transferred to the PLC interface (interface signal "Key-operated switch position 0 to 3" (DB10, DBX56.4 to 7) and can be evaluated by the PLC user program. Key-operated switch position 0 has the lowest access rights and position 3 has the highest.

Intervention via the PLC user program

The PLC interface signal "Key-operated switch positions 0 to 3" can be input either directly via the key-operated switch on the machine control panel or from the PLC user program. Only one interface signal must be set. If more than one interface signal is set at the same time, the control sets the key-operated switch to position 3.

Data management Overview



General

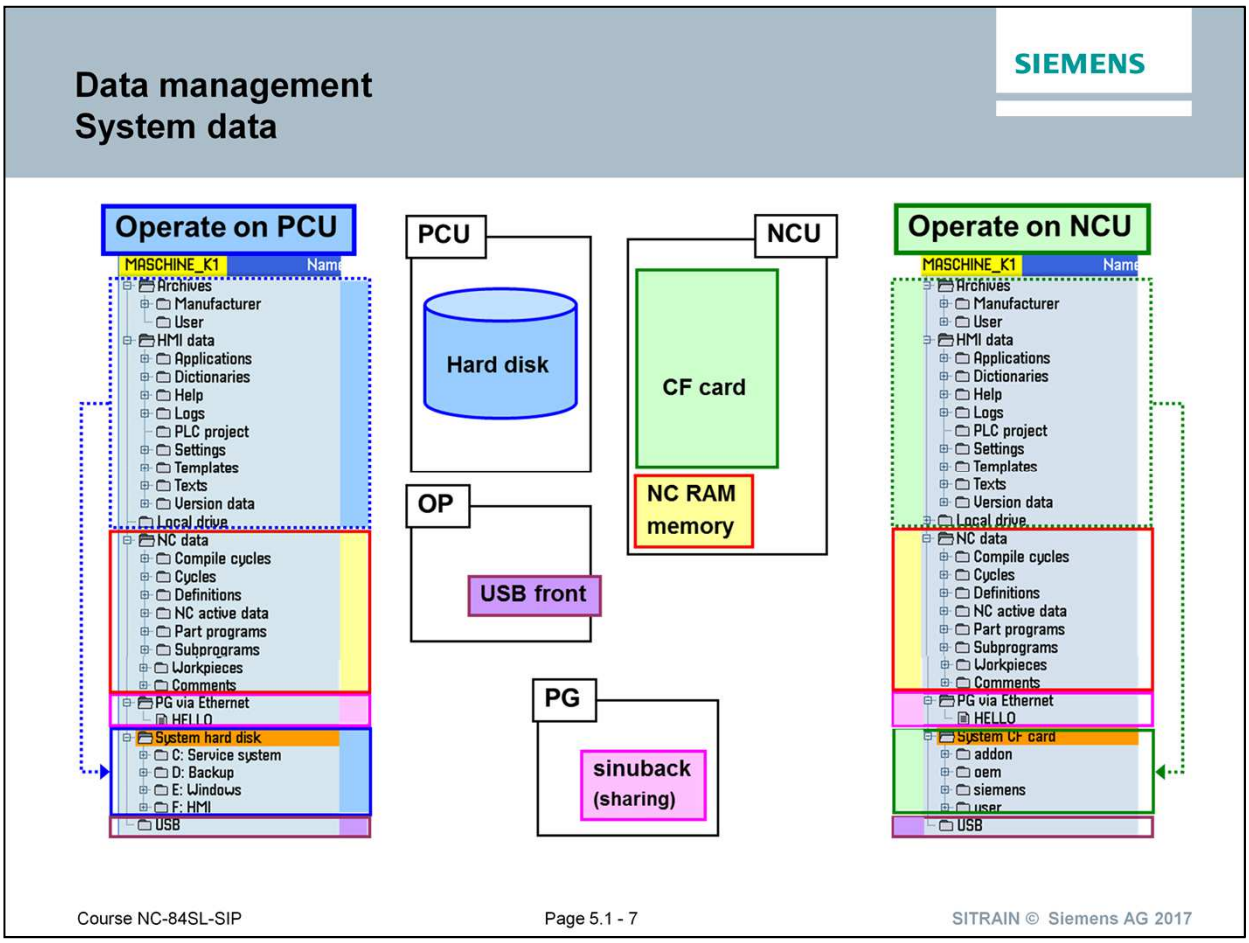
Data backup is necessary

- After commissioning
- After you have changed machine-specific settings
- When performing service tasks (e.g. after replacing hardware, software upgrade)
- During commissioning
- Before changing the memory configuration

- Parts and chapters

The entire data backup for SINUMERIK 840D sl is divided into:

- Data backup for NCK
- Data backup for PLC
- Data backup for Drive
- Data backup of hard disk (PCU only)
- Data backup of the CF card



Location of system data

With SINUMERIK Operate, the system data is located in various storage locations. The details are determined by the HMI variant (PCU/NCU):

- Operate on NCU With SINUMERIK Operate on the NCU, the system data includes:
(HMI designation on the left, storage location on the right):

- **Archive** **CF card (NCU)** (->/card/oem/sinumerik/data/archive)
- **HMI data** **CF card (NCU)** (-> /card/xxx/sinumerik/hmi/ xxx=siemens, oem, user)
- **Local drive** **CF card (NCU)** (->/card/user/sinumerik/data/prog)
- **NC data** **NC-RAM (NCU)**

Other drives can also be displayed, for example:

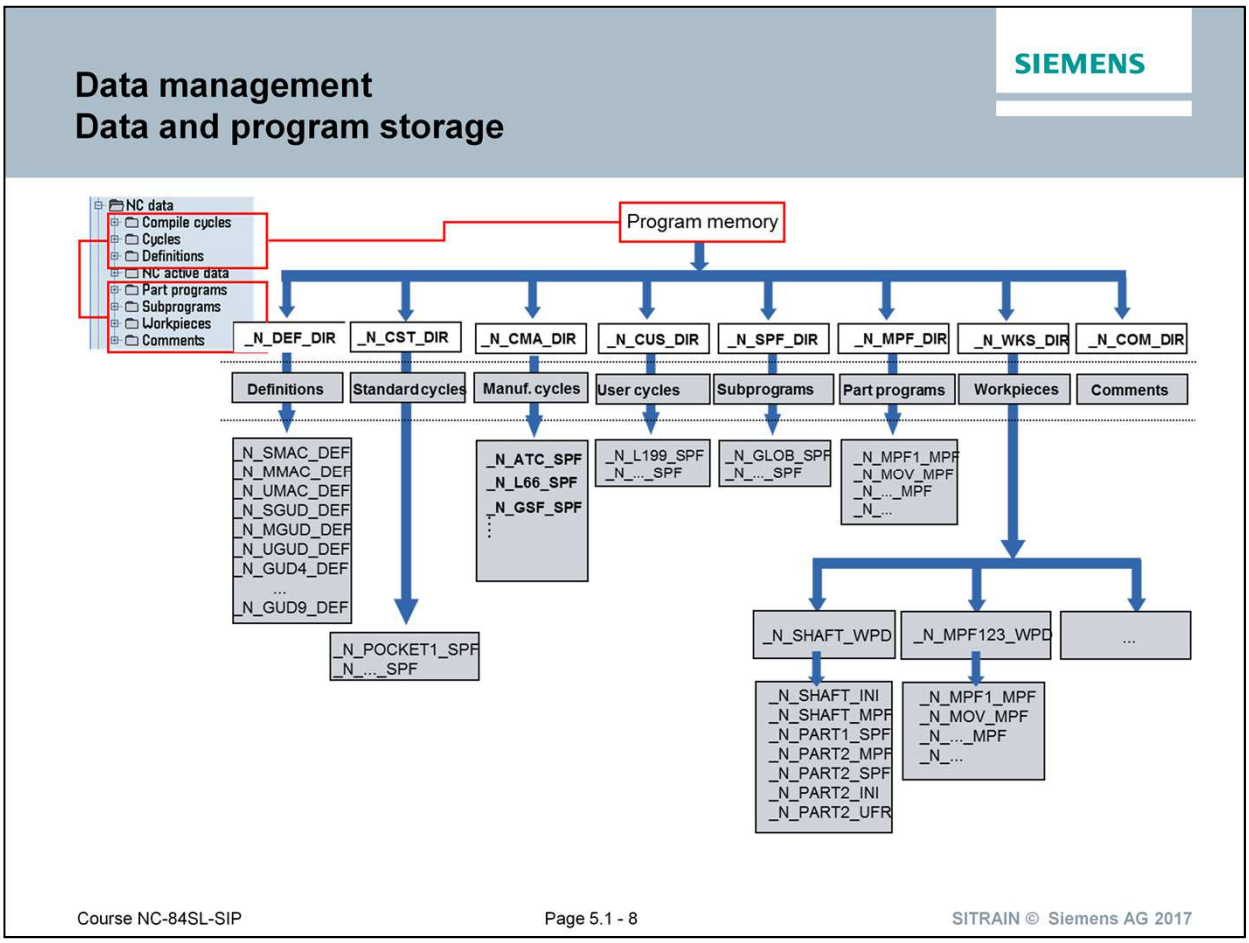
- **PG via Ethernet** **Sharing (sinuback) on PG**

- Operate on PCU With SINUMERIK Operate on the PCU, the system data includes

- **Archives** **Hard disk (PCU)** ->F:\hmisl\oem\sinumerik\data\archive)
- **HMI data** **Hard disk (PCU)** (-> F:\hmisl\xxx\sinumerik\hmi\ xxx=siemens, oem, user)
- **Local drive** **Hard disk (PCU)** (->F:\hmisl\user\sinumerik\data\prog)
- **NC data** **NC-RAM (NCU)**

Other drives can also be displayed, for example:

- **PG via Ethernet** **Sharing (sinuback) on PG**



SINUMERIK Operate

- Program memory

The program memory (softkey: "NC") is located on the NCU.

Part programs and workpieces can be stored in the program memory.

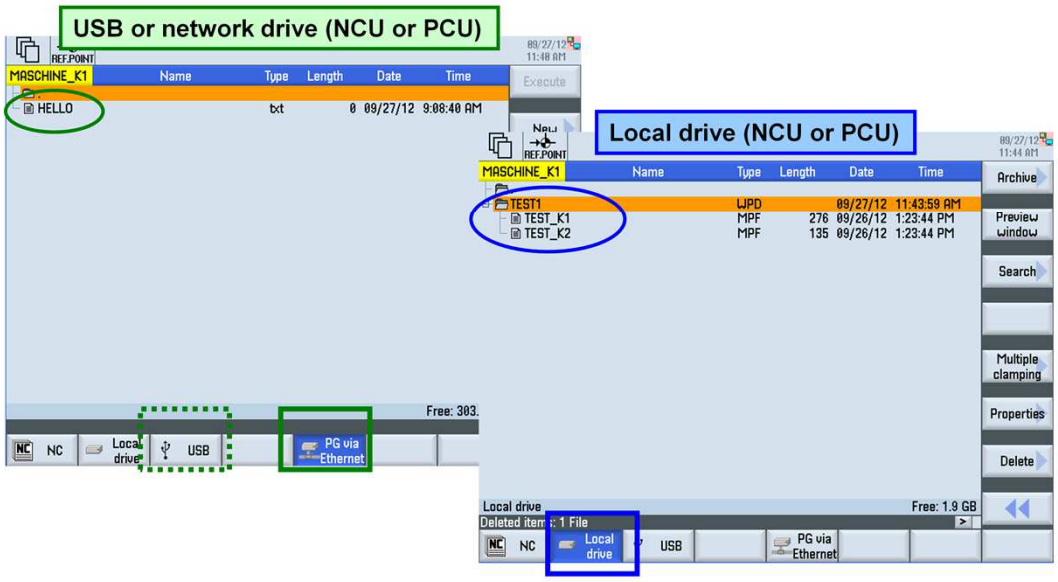
- Definitions

Macros and global user variables are stored in the "**Definitions**" directory. These data are located on the NCU.

Macros can be used to adapt the programming language. You can combine a single program line into a new command in a macro.

The global user variables (GUD) can be used instead of the R-parameters for part programming. GUD must be used for parameters that apply over more than one channel.

Data management
USB, network drive, local drive



 SITRAIN © Siemens AG 2017

Network drive

Part programs and workpieces can be located on network drives.
This is supported by SINUMERIK Operate on NCU or PCU.

The corresponding softkey ("**Network PG**" in the example) must be configured with the "**Logical drives**" function.

USB

Part programs and workpieces can also be located on USB drives.
This is supported by SINUMERIK Operate on NCU or PCU.

Local drive

SINUMERIK Operate on NCU or PCU also supports the use of the "**local drive**" (NCU CF card or PCU hard disk) as a memory for workpieces and part programs. The corresponding path is:

/card/user/sinumerik/data/prog of the NCU, or

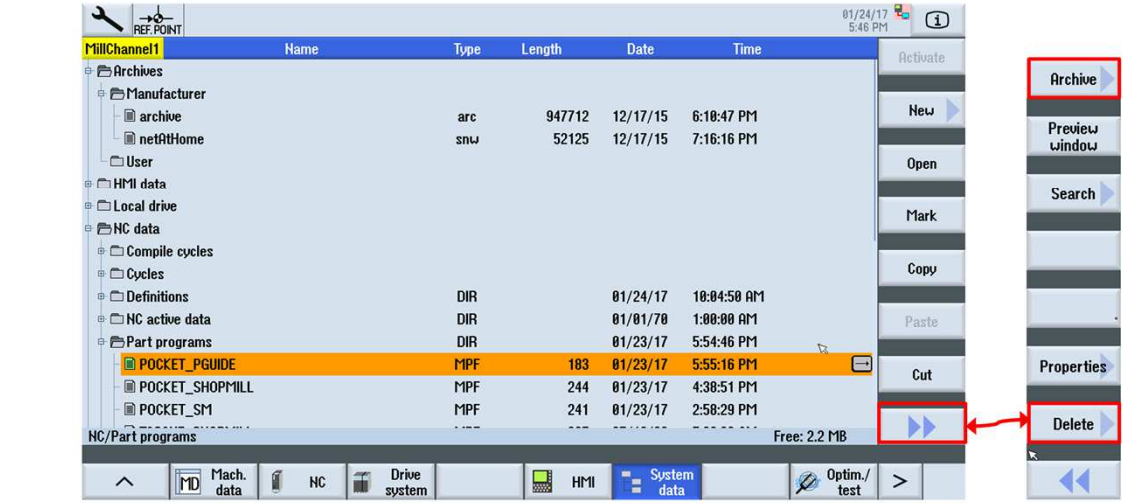
C:\ProgramData\Siemens\MotionControl\user\sinumerik\data\prog of the PCU.

The local drive on the NCU requires option P12 (6FC5800-0A**P12**-0YB0)

Softkey: "Selection"

In all cases, pressing the Select softkey and starting the NC triggers the execution of the corresponding program directly from the network drive/USB drive or local drive.

Archiving specific files



 SITRAIN © Siemens AG 2017

If only certain data are to be backed up, the required files can be directly selected from the data tree and an archive created. That is stored in directory "Archiv". The archive can be stored in binary or punched-tape format.
The content of the selected files (XML, ini, hsp, syf files, programs) can be displayed in a preview.
Information about the file (path, name, creation date, and change date, for example) can be displayed in a Properties window.

"Archive" softkey Data backup of the data areas selected using the "Highlight" softkey in the Archive directory.

Storage of the archives

If the "Archive/Manufacturer" path has been selected, the archive is located as follows:

- operate on NCU: **/card/oem/sinumerik/data/archive** CF card on the NCU

- operate on PCU: **C:\ProgramData\Siemens\MotionControl \oem\sinumerik\data\archive** Hard disk of the PCU

If the "Archive/User" path is selected,
"oem" in the storage paths must be replaced by **"user"**.

USB / Network Alternatively, any USB or network drive can be selected as a path.
This applies for SINUMERIK Operate on NCU and PCU.

Note A password is required for some functions.

Punched-tape format

```
%_N_CONTOUREXT_MPF
;$PATH=/_N_WCS_DIR/_N_EXTERNAL_WPD
N10 G90 G0 G53 X280 Z80 D0 DIAMON
N20 G96 S250 M4 LIMS=4000
N30 T502 D1 M8
N40 MSG("cut off")
N50 Z0
N60 G1 X-3 F.5
...
N490 M30
```

```
%_N_CH1_RPA_INI
CHANDATA(1)
R[0]=10
R[1]=25.36
R[2]=3.65
M17
```

```
%_N_TURNINGTOOLS_TOA
;$PATH=/_N_WCS_DIR/_N_EXTERNAL_WPD
$TC_DP1[1,1]=500
$TC_DP2[1,1] = 0
$TC_DP6[1,1]=0
$TC_DP7[1,1]=0
$TC_DP8[1,1]=0
...
M17
```


SITRAIN © Siemens AG 2017

Punched-tape format Data in punched-tape format can be generated externally or processed with an editor.

Identifier %_N_Filename_Type1_Type2
Path name

File name File names can contain the characters 0...9, A...Z, a...z or _ and must not exceed 24 characters in total.

Type 1 Data type
The Type1 identifier in the file name specifies the type of variable the file consists of (e.g. machine data, frames, user variables).

Type 2 Optional additional identifier for execution
If the Type 2 identifier is "INI", the file is executed when it is read in.

Path names Path names in punched-tape format start with ;$PATH=/<path name>
Examples:

- Main program "MPF100" in the "Main programs" directory

  %_N_MPF100_MPF
  ;$PATH=/_N_MPF_DIR

- Main program "MPF100" in the workpiece "PART 15"

  %_N_MPF100_MPF
  ;$PATH=/_N_WCS_DIR/_N_PART15_WPD

Machine data for data input/output

Generate line checksum	MD 11230, BIT 0 = 1
Generate machine data numbers	**MD 11230, BIT 1 = 1**
Termination behavior on reading MD	MD 11220, < 0 , 1 , 2 >
Only back up MDs deviating from the standard	**MD 11210, < 0 , FF >**
Loading standard MD	MD 11200 < 0 , 1 , 2 >

SITRAIN © Siemens AG 2017

Line checksum

If MD 11230 bit 0 = 1 is set, a line checksum is added to the machine data at data output.
Example: $MC_AXCONF_MACHAX_USED[0]=1 **`2F34**

MD numbers

If MD 11230 bit 1 = 1 is set, then the machine data will be saved with the MD number preceding it.
Example: N20070 $MC_AXCONF_MACHAX_USED[0]=1

Behavior on termination

If errors are detected when reading in machine data files (INI files), the further behavior depends on this MD:

- MD 11220 = 0: Output an alarm, abort upon detection of the 1st error
- MD 11220 = 1 Output of an alarm, continue execution
  Output the number of errors at the end of read-in
- MD 11220 = 2 Execution continues despite possible errors until the end of file is reached. The number of errors is displayed with an alarm at the end.

Backing up changed values

When backing up machine and setting data, you can set MD 11210 UPLOAD_MD_CHANGES_ONLY to select whether to output all data or just the data that was changed and differs from the default setting.

MD11210 = 0	**Save everything**
MD11210 = ff	Only save changes (parameterizable)

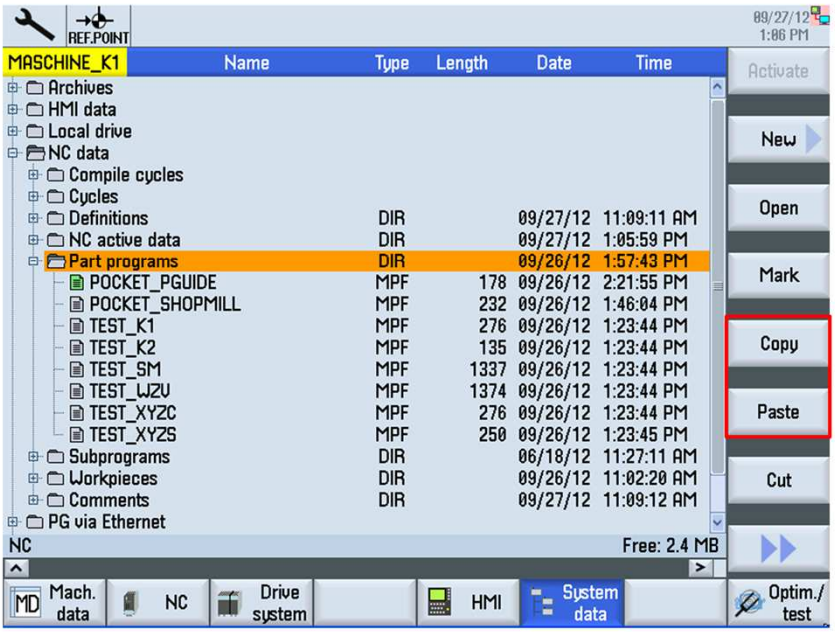
MD11212 influences the data content of selected files if MD 11210 is not equal to 0.

Loading standard data

MD 11200 = 0	Normal run-up
MD 11200 = **1**	The standard MD is loaded
MD 11200 = **2**	Machine data that configure the memory are overwritten with default values.

Copying and pasting files





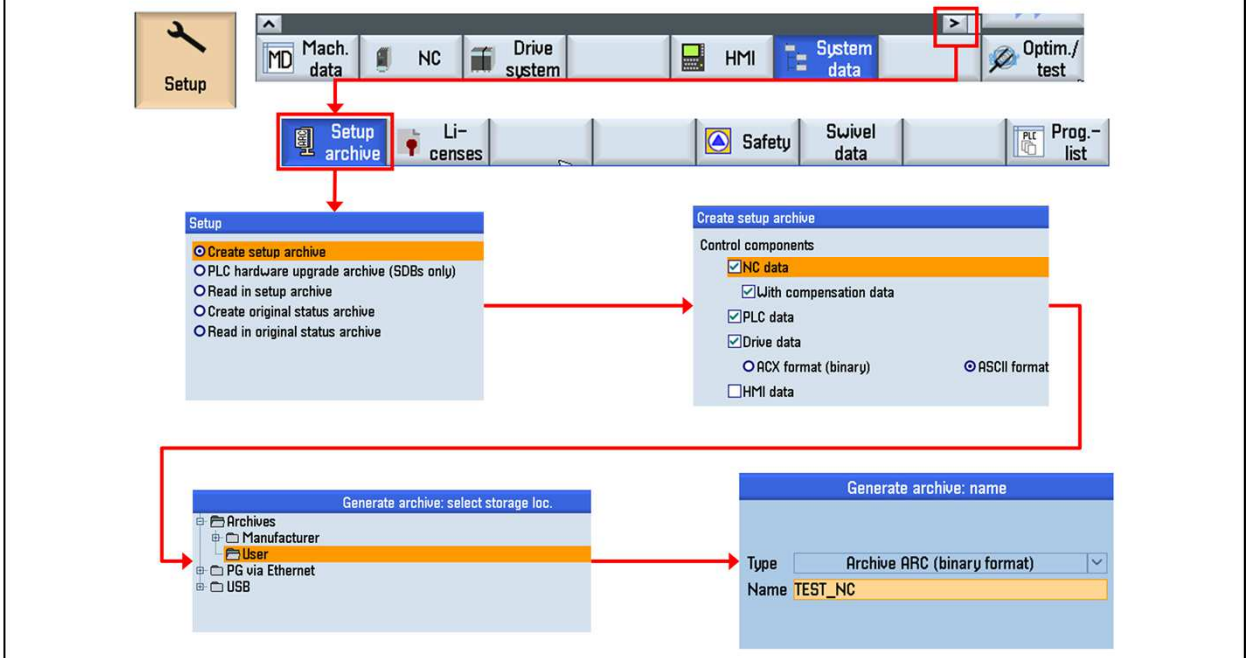
 SITRAIN © Siemens AG 2017

"Copy" and "Paste" softkeys

These softkeys are used to transfer files or directories to a USB flash drive, for example.

The files/directories to be transferred are selected using the Highlight" softkey.

Creating a commissioning archive



 SITRAIN © Siemens AG 2017

Setup archive Is used for simple and complete data backup, as well as for restarting a control after data loss or hardware replacement. These files are saved in binary format.

- NC (optionally with compensation data)
- PLC (PLC program in machine code)
- Drive data (SINAMICS data)

Create You can create setup archives for NC, PLC, drives, and HMI. To do this, select the Setup archives option from the menu extension in the Setup operating area.

Storage locations for archives

When the "**Archive/User**" or "**Archive/Manufacturer**" target is selected, the storage locations are as follows:

Operate NCU
- On the CompactFlash Card under **/card/user/sinumerik/data/archive** or **/card/oem/sinumerik/data/archive**

Operate XP PCU
- On the PCU hard disk under **F:\hmisl\user\sinumerik\data\archive** or **F:\hmisl\oem\sinumerik\data\archive**

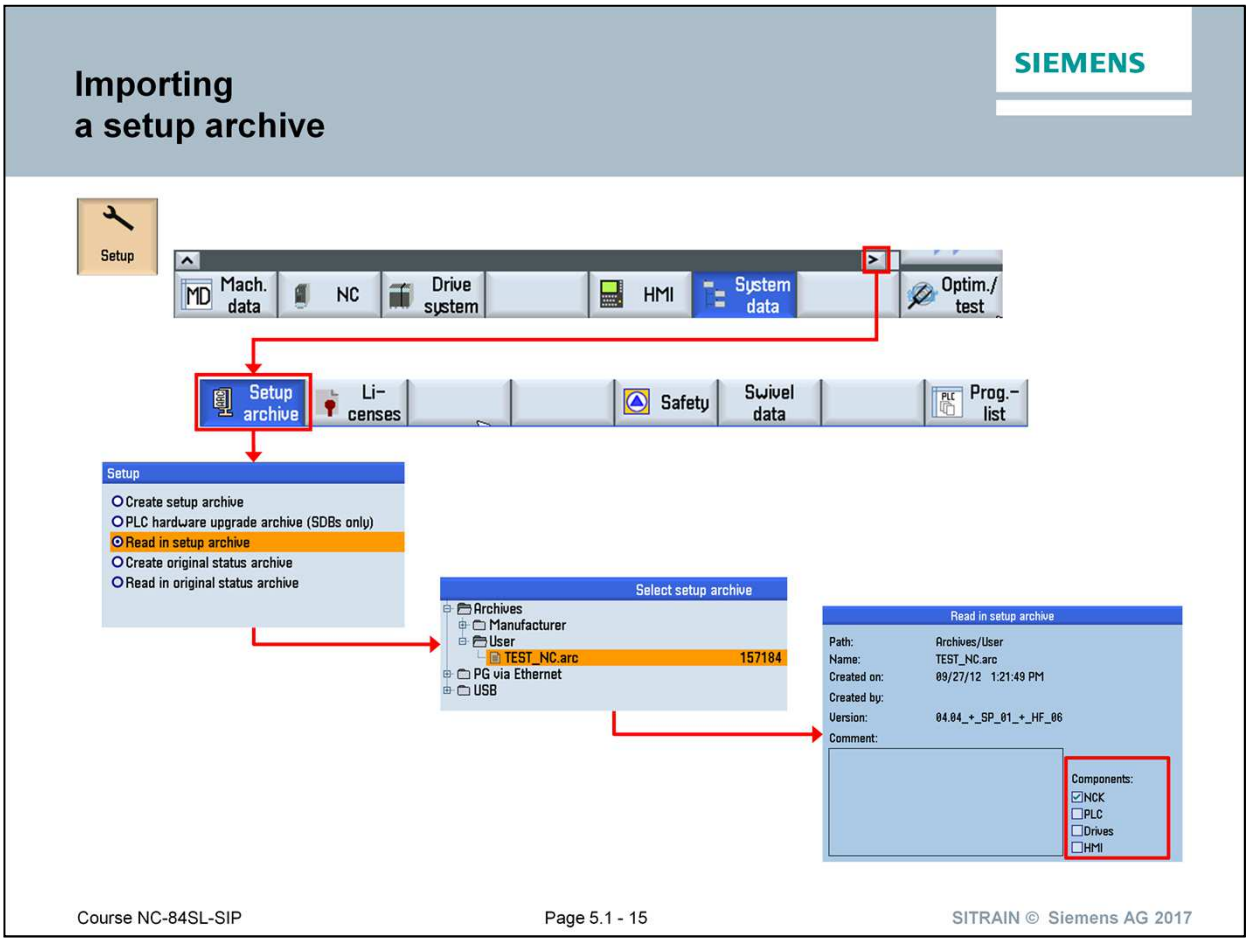
Operate Win7 PCU
- On the PCU hard disk under **C:\ProgramData\Siemens\MotionControl\user\sinumerik\data\archive**

Read in To read in an archive, select "**Read in**" from the **Setup archive** menu.

Notes

- PLC You only save the HW Config with the system data blocks (PLC).

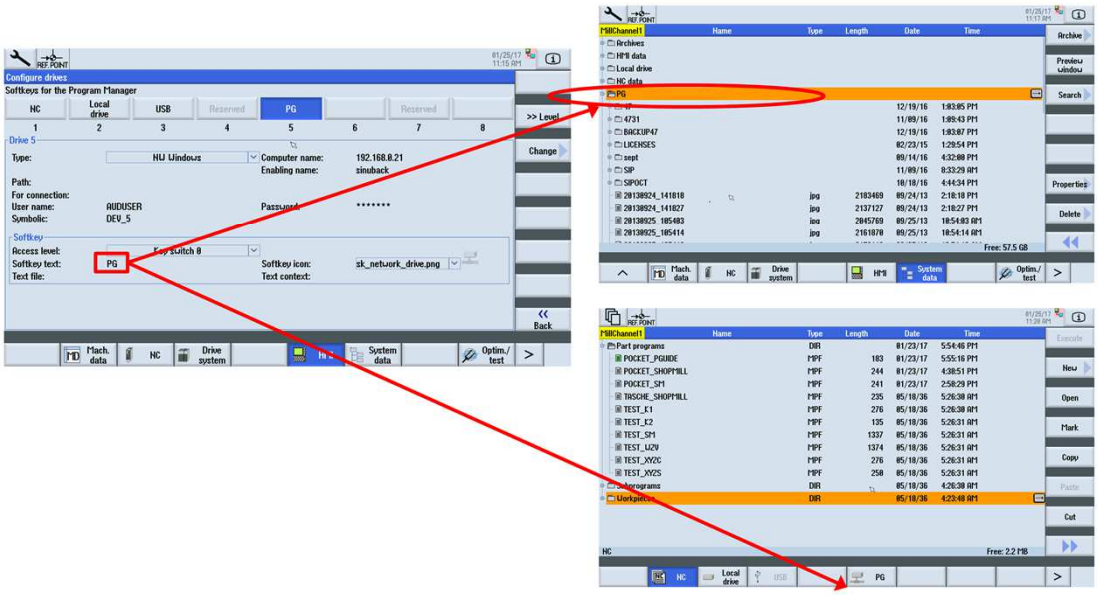
- DP Use ACX format for the drive data.



Selective import

As from SW 4.5, the individual components can also be imported singly from common NC/PLC/drives archives.

HMI: Logical drives



 SITRAIN © Siemens AG 2017

Function

In the Setup operating area,
softkey→ HMI
softkey→ Log. drives
additional USB or network drives can be declared.

These can also be seen under **softkey: System data** and **softkey: Program manager**.

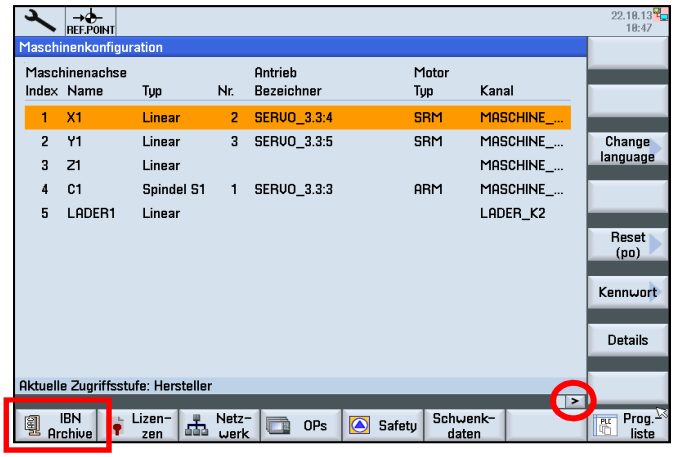
Exercise: Create Setup archives

Objective: To create series setup archives for NC, PLC and drives

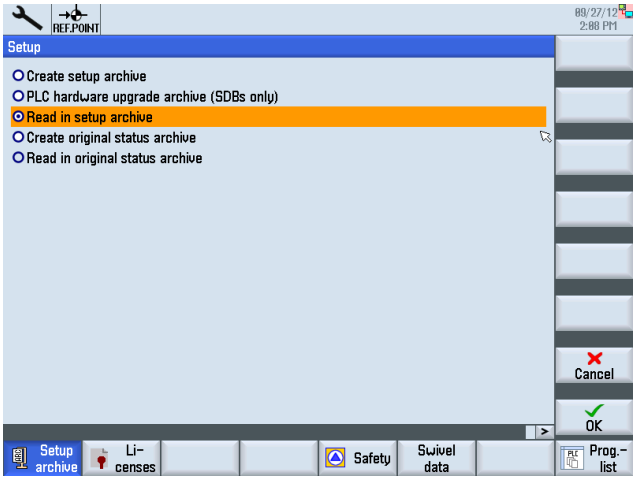
Procedure Switch to the **Setup operating area**.

Extend the horizontal softkey bar
(etc.) to display the "Setup archive" softkey.

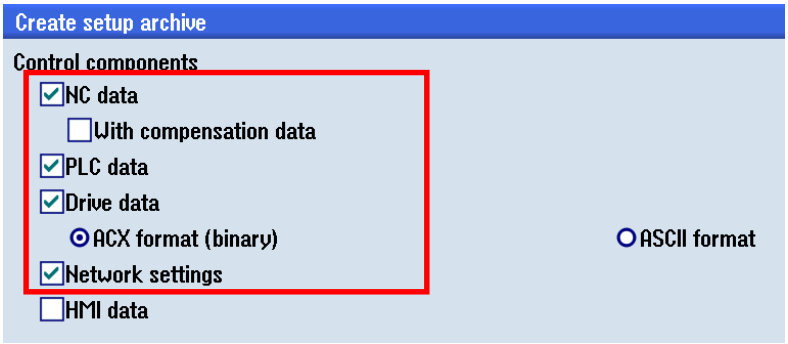




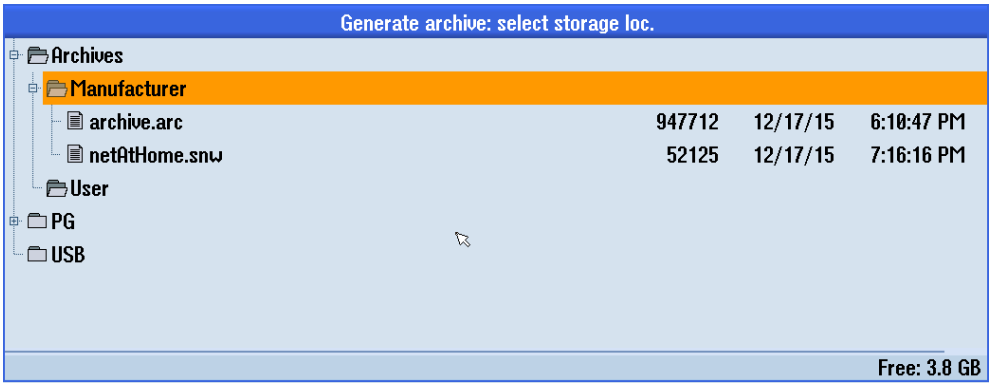
Press the **softkey: Setup archive.** In the dialog box that then opens select "**Create setup archive**" and confirm with **softkey "OK."**



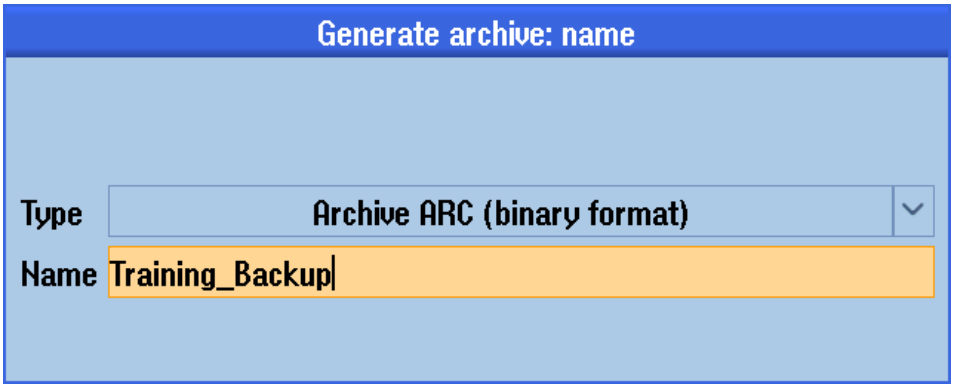
Next, create a **shared** setup archive for the NC, PLC, and drives.



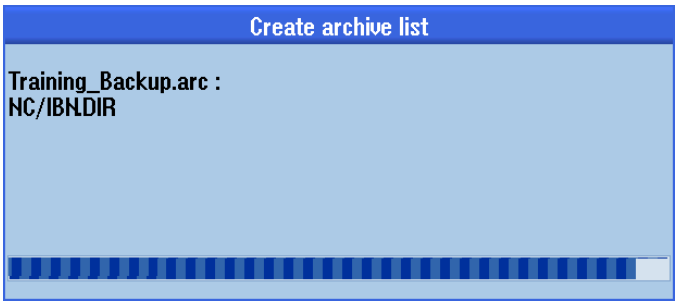
Select the manufacturer directory as the storage location.



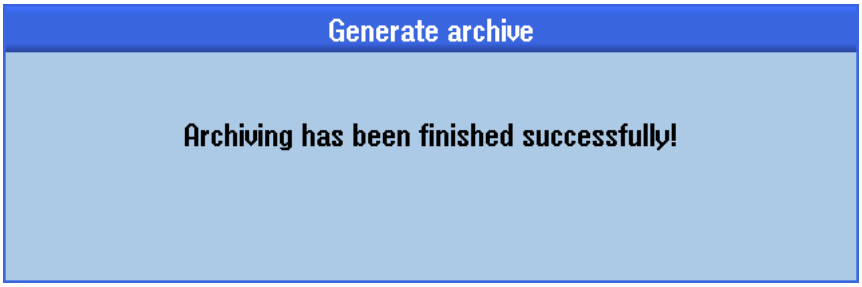
Allocate meaningful names to your archives.



Click OK to create the archives.



On completion of the process, a message is displayed indicating that archiving has been successful.



Exercise: Load SINAMICS Factory Settings

Objective

The purpose of this step is to also restore a known initial state in the SINAMICS. Proceed as follows to set this state:

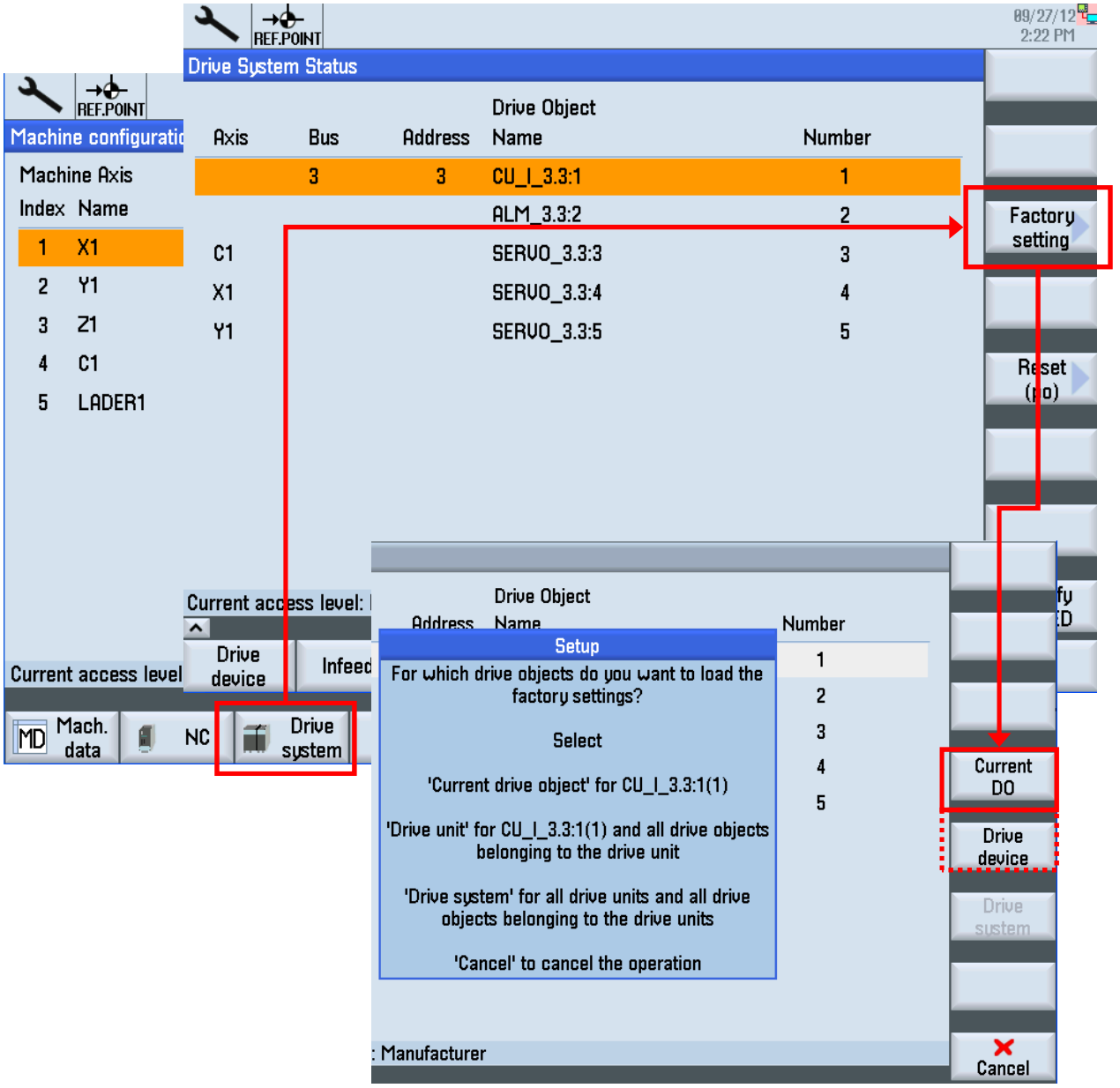
1. Control Unit MD

In the HMI select

-> Setup -> Drive system

-> Factory settings -> Drive system

(SK not active? -> Drive unit)



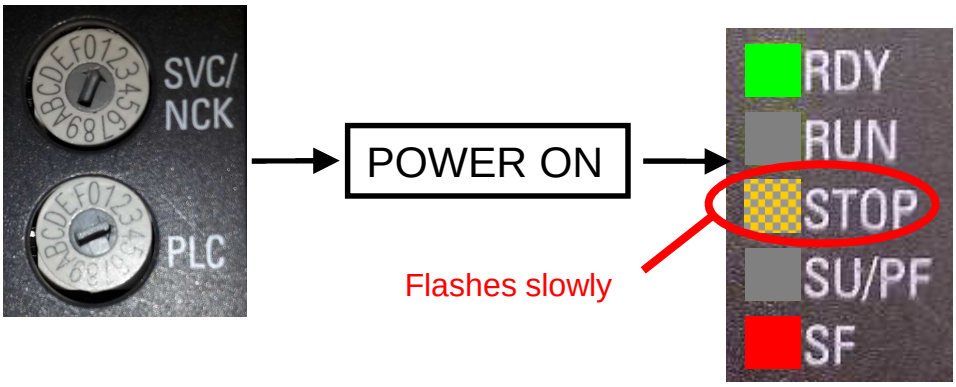
2. Factory settings

Wait for the prompt for automatic device configuration.

General reset of the NC/PLC

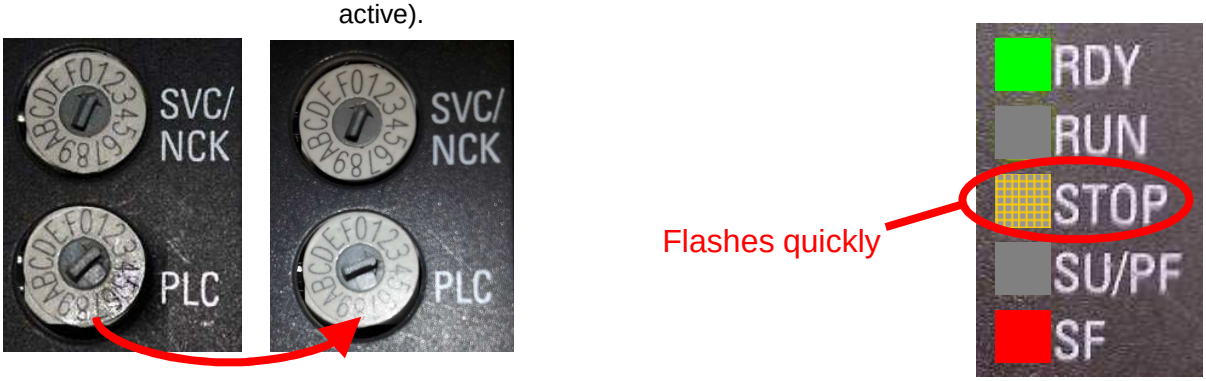
1. Switch off the control.
2. Request a general reset of the NCK and of the PLC
   - Set the "SVC/NCK" rotary switch to position "1".
   - Turn the PLC rotary switch to position "3".
   - POWER ON (switch control off and back on again).

   -> The 'STOP' LED on the PLC flashes slowly (general reset request detected).



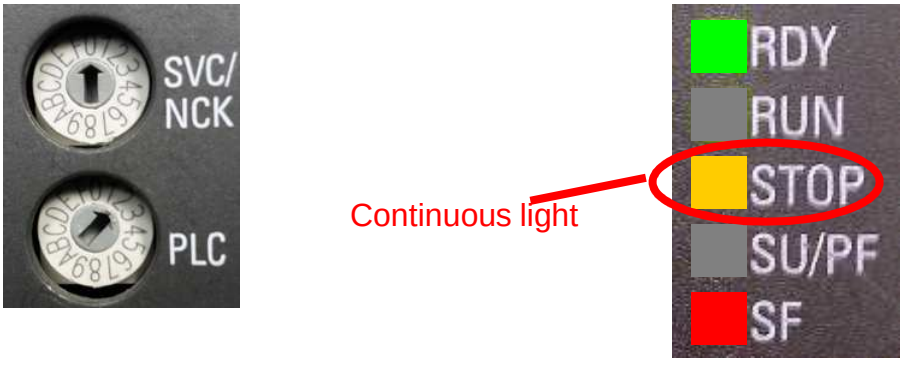
3. Acknowledge the PLC general reset
   - Turn the PLC rotary switch to position "2" and back to position "3".

   -> The 'STOP' LED on the PLC flashes quickly several times (general reset active).



4. At the end, the switches are set as follows:
   - NC switch to "0"

   -> PLC switch in position "2" (STOP). -> The STOP LED is continuously lit.



The NC memory reset is now complete: Data (machine data, R parameters, etc.) are set to default values, all part programs and tool data has been deleted, and the password has been reset.

A general reset of the PLC is performed and it is still in the stop state.

Exercise: Import the setup archive with SINUMERIK Operate

Objective: To read in setup archives of the NC, PLC, and drives

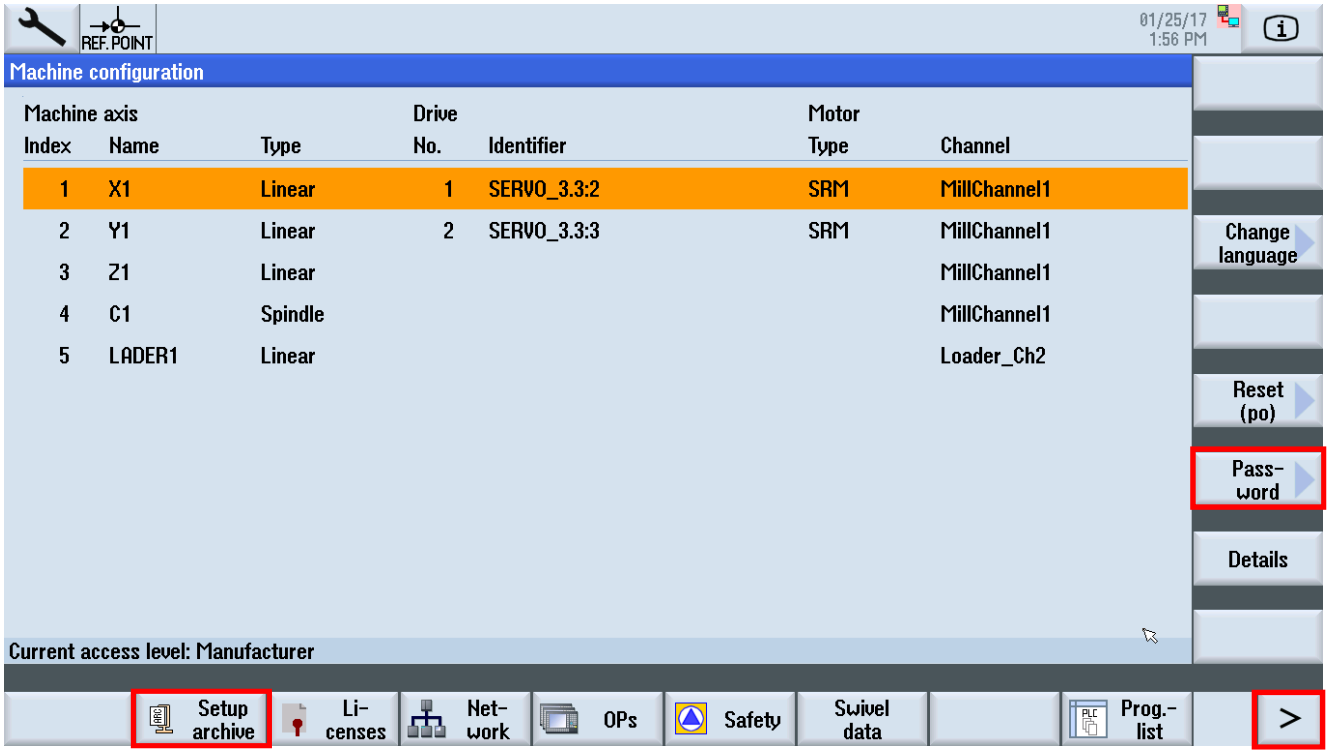
Procedure: Switch to the Setup operating area.

Set the password:

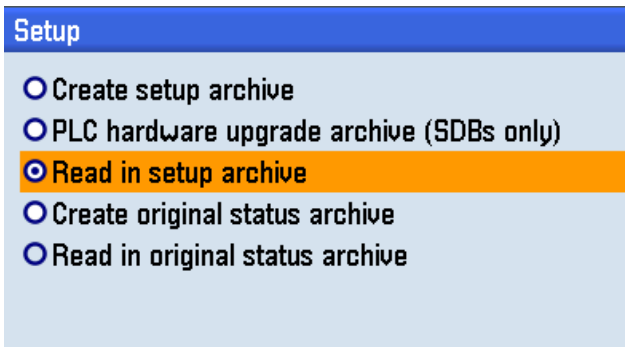
Softkey: Password → Set password → SUNRISE

Extend the horizontal softkey bar
(ETC key) to display the "Setup archive" softkey.



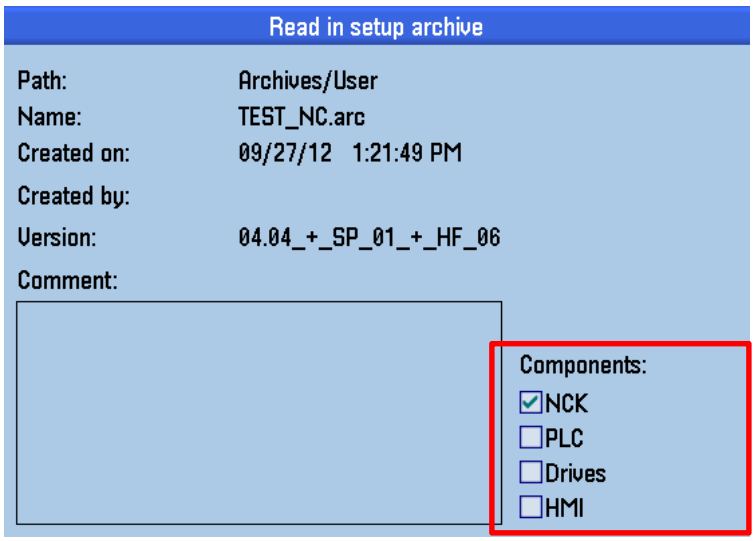


Press the **softkey: Setup archive**, in the dialog box that then opens, select **"Read in setup archive"**.



After you have selected an archive, you can also select whether the complete archive or only parts will be read in. This makes it possible, for example, to restore only the drives from an NC/PLC/drive archive.

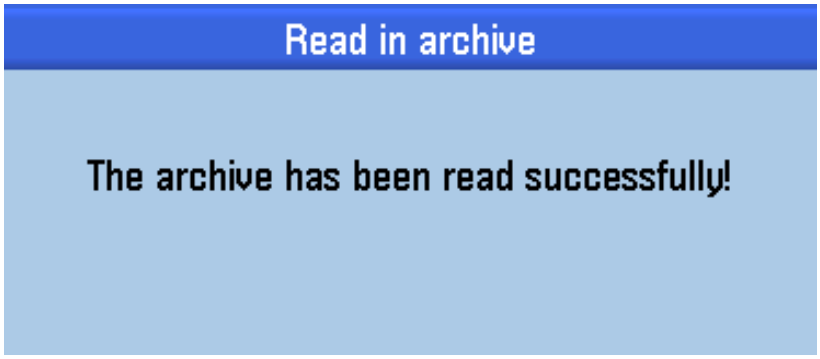
Leave the complete selection (NCK, PLC, drives).



After you have confirmed with the **softkey: OK**, the archive is read in again.

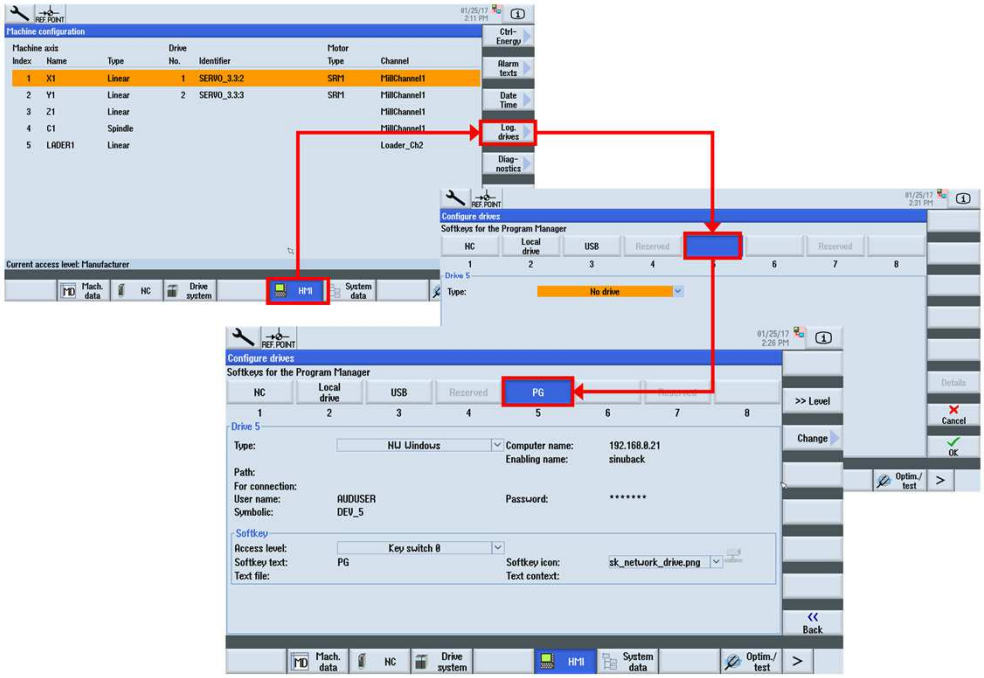


On completion of the process, a message is displayed indicating that the process has been successful.



Exercise: Set up a network drive
Logical drive

SIEMENS



 SITRAIN © Siemens AG 2017

Objective

To make the connected drive available under the "Network drives" softkey.

Logical drives

In SINUMERIK Operate, select

-> **OA Setup -> HMI**

-> **Logical drives**

-> **Change**

-> **Select the fifth softkey with the arrow keys**

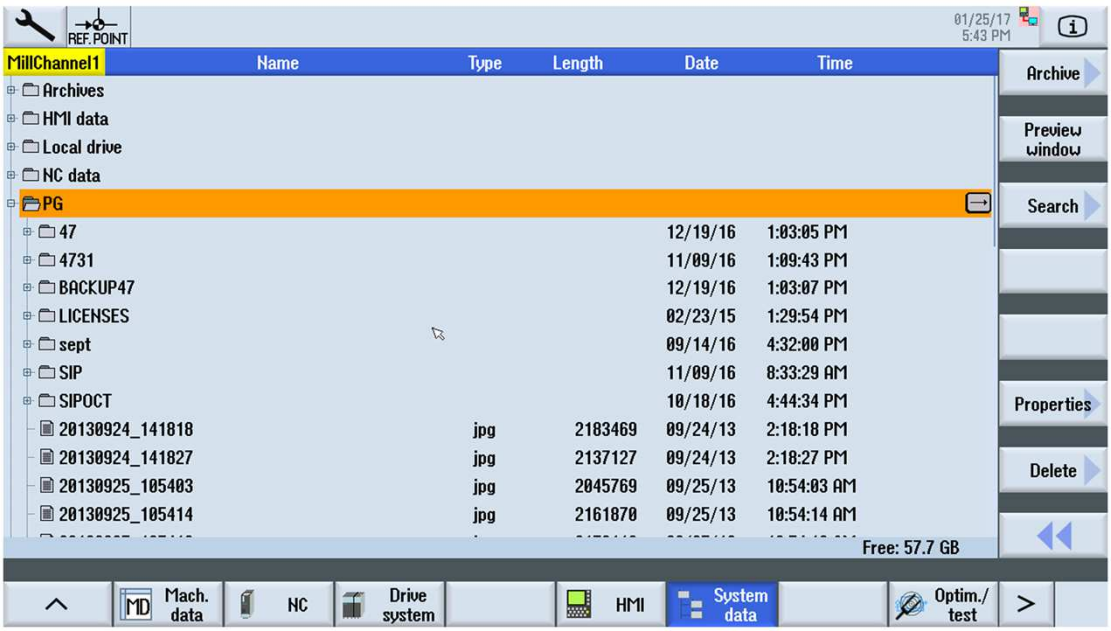
-> **Parameterize what they look like in the display:**

Type:	NW Windows
Computer name:	IP address or name of the target computer
Share name:	Name by which a directory has been shared on the target computer
Path:	Only enter if a subdirectory of the shared directory is to be described.
User name:	User for whom the target directory has been shared.
Password:	The password of this user
Symbolic:	No change required
Softkey text:	Text of the softkey or the directory on the user interface
Softkey icon:	Here, it is possible to choose between different icons that will appear next to the text on the softkey.

Restart

Restart the HMI.

Exercise: Set up a network drive
Checking success



SITRAIN © Siemens AG 2017

Objective To test access to the network

Operation In SINUMERIK Operate, select

-> OA Setup -> System data

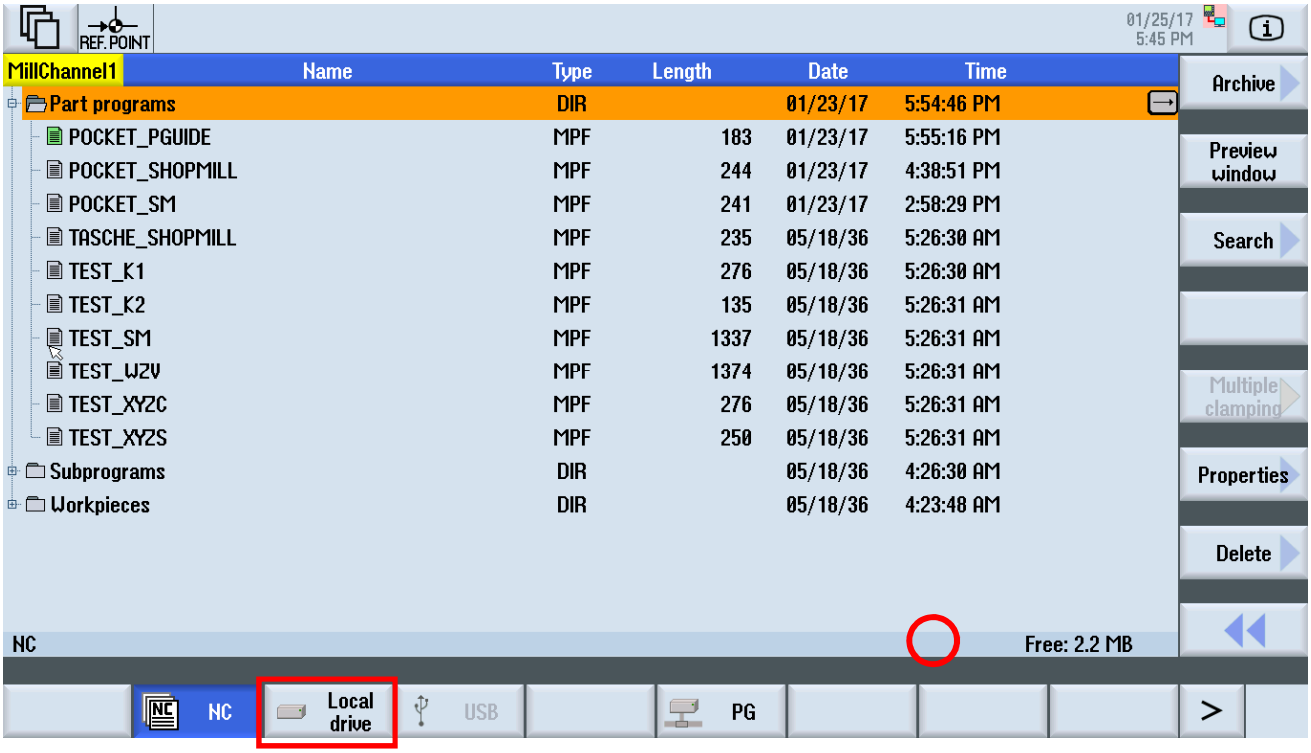
Here, open the drive assigned currently, e.g. "PG via Ethernet".

Exercise: Archive and import part programs on a network drive

Objective: To create an archive of the part program directory on the network drive that is currently connected and import this archive again:

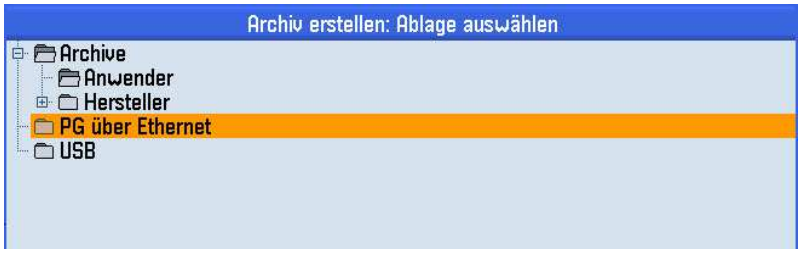
Creating archive Switch to the Setup operating area.

Softkey **System data**



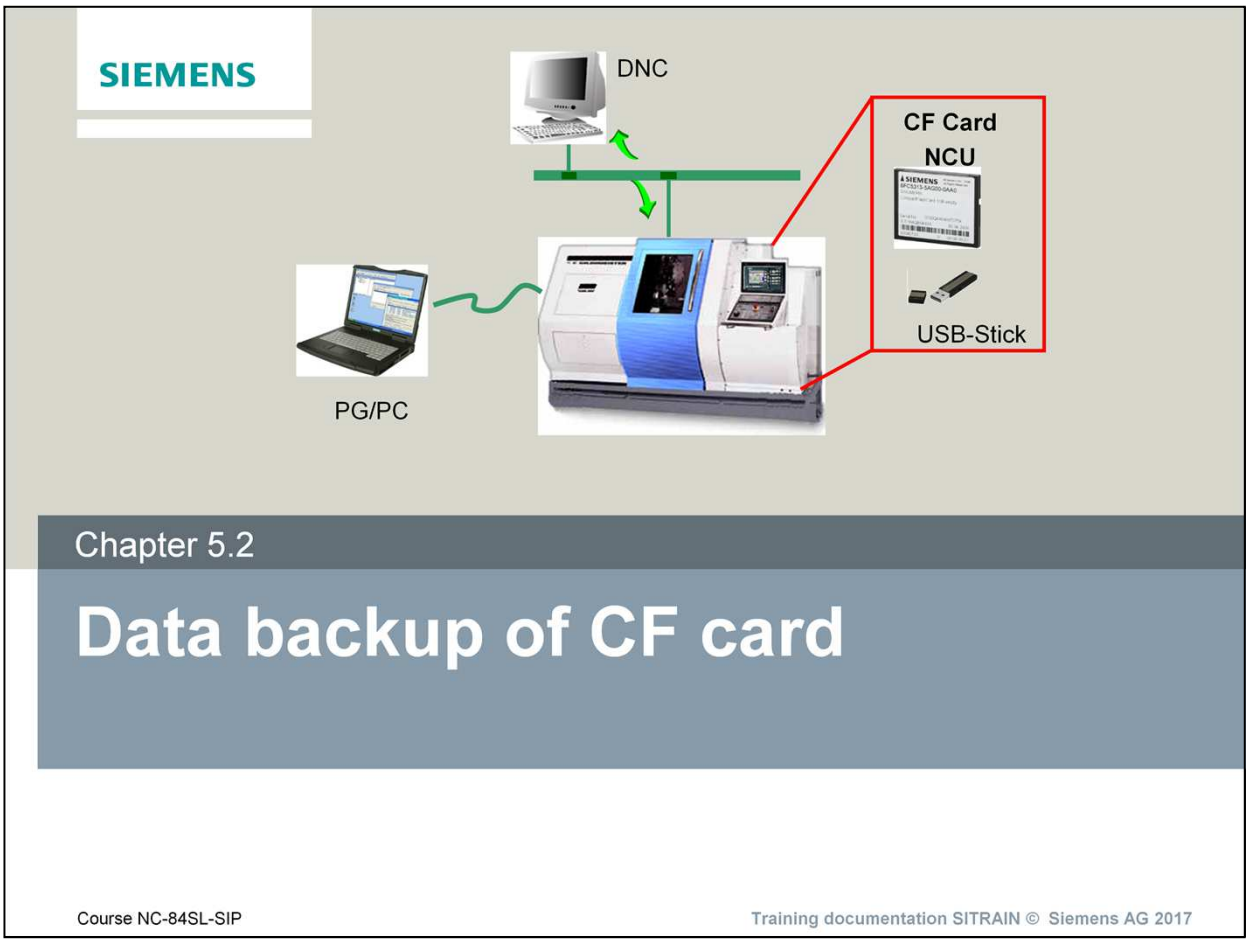
Choose the storage location:





Select a name for the archive and save it.

Importing an archive: Such archives are imported by double-clicking on the generated .arc file in the target directory and confirming the queries.
Such archives cannot be imported with the function "Read in setup archive"



Content **Page**

Objectives 2
IBN-Werkzeuge für 840D sl: CF-Kartensicherung 3
Backup of CF card: Overview 4
Data backup of CF card with NCU service stick 5
Creating the NCU service stick with AMM /P2P 6
Alternative: Creating the NCU service stick 7
Booting the NCU from the service stick 8
Main menu of the NCU service stick 9
Backup user Data to USB-stick 10
Change NCU settings with the service-stick 11
Service Shell 12
Exercise: Back up complete CF card to NCU service stick 13
Exercise: Create a NCU service-stick with AMM /P2P 14
Exercise: Backup the CF card to USB stick 15
Exercise: Upadte/Recover of Software and Data 16

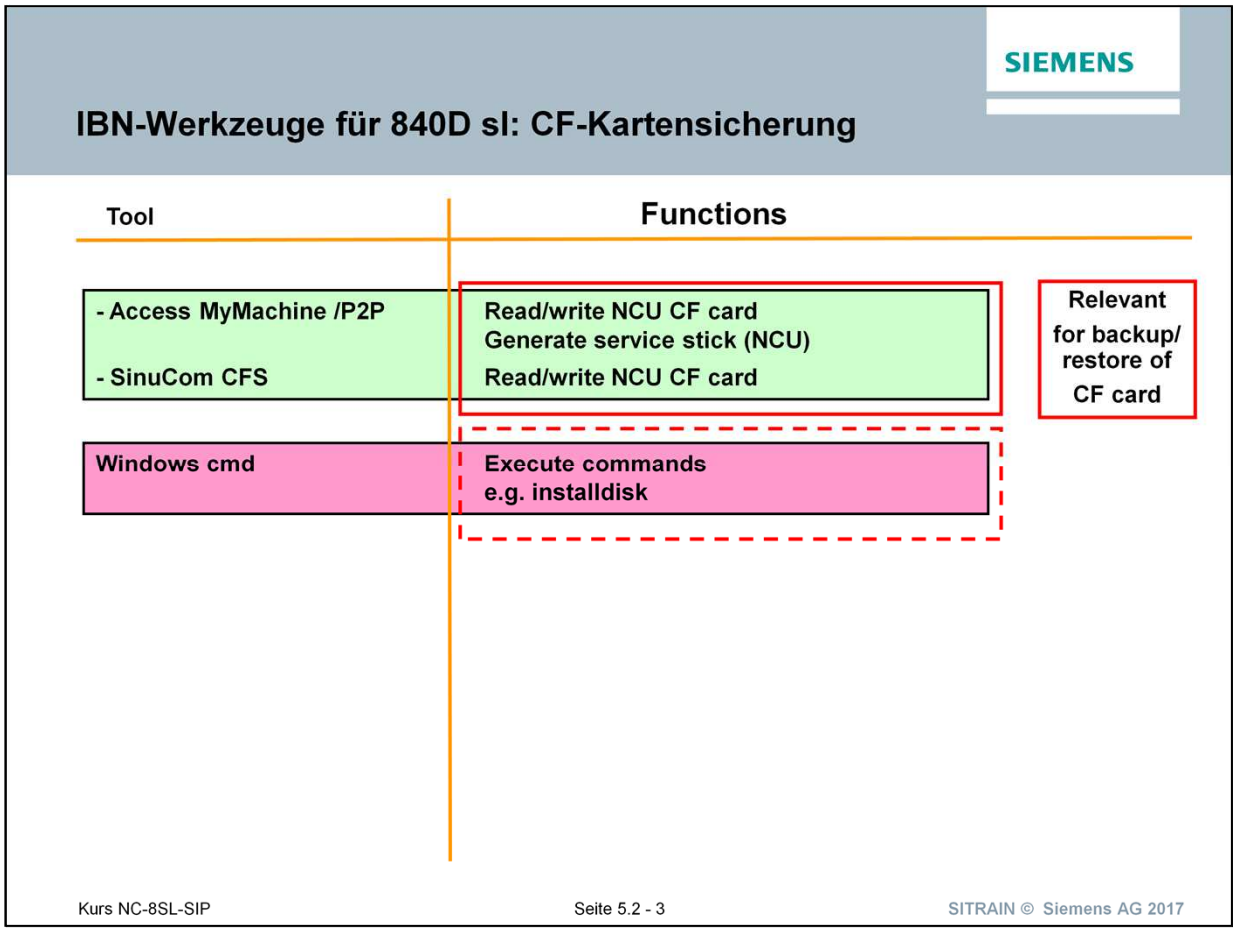
Objectives

You can back up and restore the NCU CF card

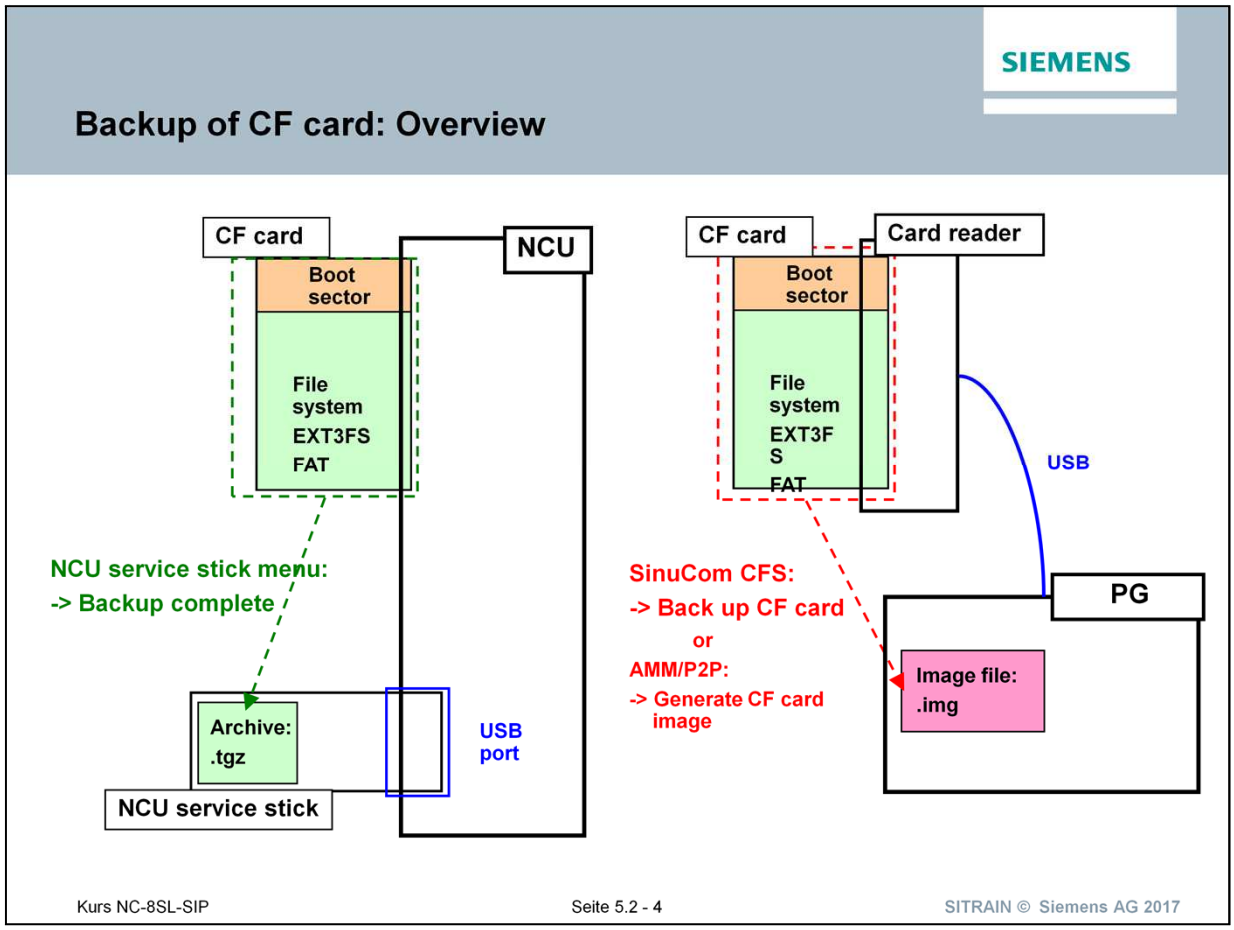


SITRAIN © Siemens AG 2017

Notizen


SIEMENS
IBN-Werkzeuge für 840D sl: CF-Kartensicherung
Tool
Functions
- Access MyMachine /P2P
- SinuCom CFS
Read/write NCU CF card
Generate service stick (NCU)
Read/write NCU CF card
Relevant for backup/ restore of CF card
Windows cmd
Execute commands
e.g. installdisk
Kurs NC-8SL-SIP
Seite 5.2 - 3
SITRAIN © Siemens AG 2017

Notizen



Backup with NCU service stick

The NCU is booted with the **service stick** (plugged into one of the USB ports on the NCU).

The **"Backup complete CF card"** function in the NCU service stick Service menu is used for this purpose.

-> File system With this backup method, the file system and the boot sector of the CF card are saved in a *.tgz file.

Backup with SinuCom CFS (or Access MyMachine)

The CF card is plugged into a card reader on the PG.

In **SinuCom CFS**, the **"Read card"** function is used.
(In AMM /P2P, the function is called "Creating the CF card image").

-> Complete Backs up the complete CF card including the boot sector.

Data backup of CF card with NCU service stick

You need:

1. A NC-bootable USB stick
2. Installdisk.exe software
3. File linuxbase.img
4. An NCU



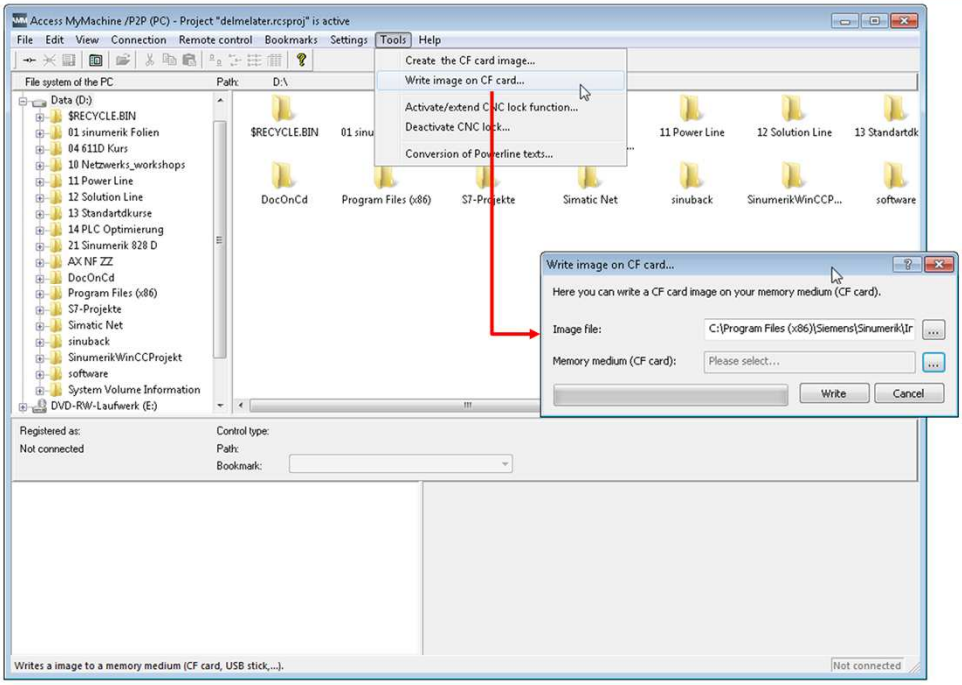


Kurs NC-8SL-SIP Seite 5.2 - 5 SITRAIN © Siemens AG 2017

USB stick

Can be ordered from the SINUMERIK product range
(USB flash drive 8GB, order no. **6ES7648-0DC50-0AA0**).
Other USB sticks may work, although they are not bound to.

Creating the NCU service stick with AMM /P2P



Procedure for creating the NCU service stick with AMM /P2P

Connection of USB boot stick

Connect the USB device to the PG (USB stick, card reader with CF card, etc.)

Check drive

Check the drive letter of the USB device (Explorer).

Write service stick

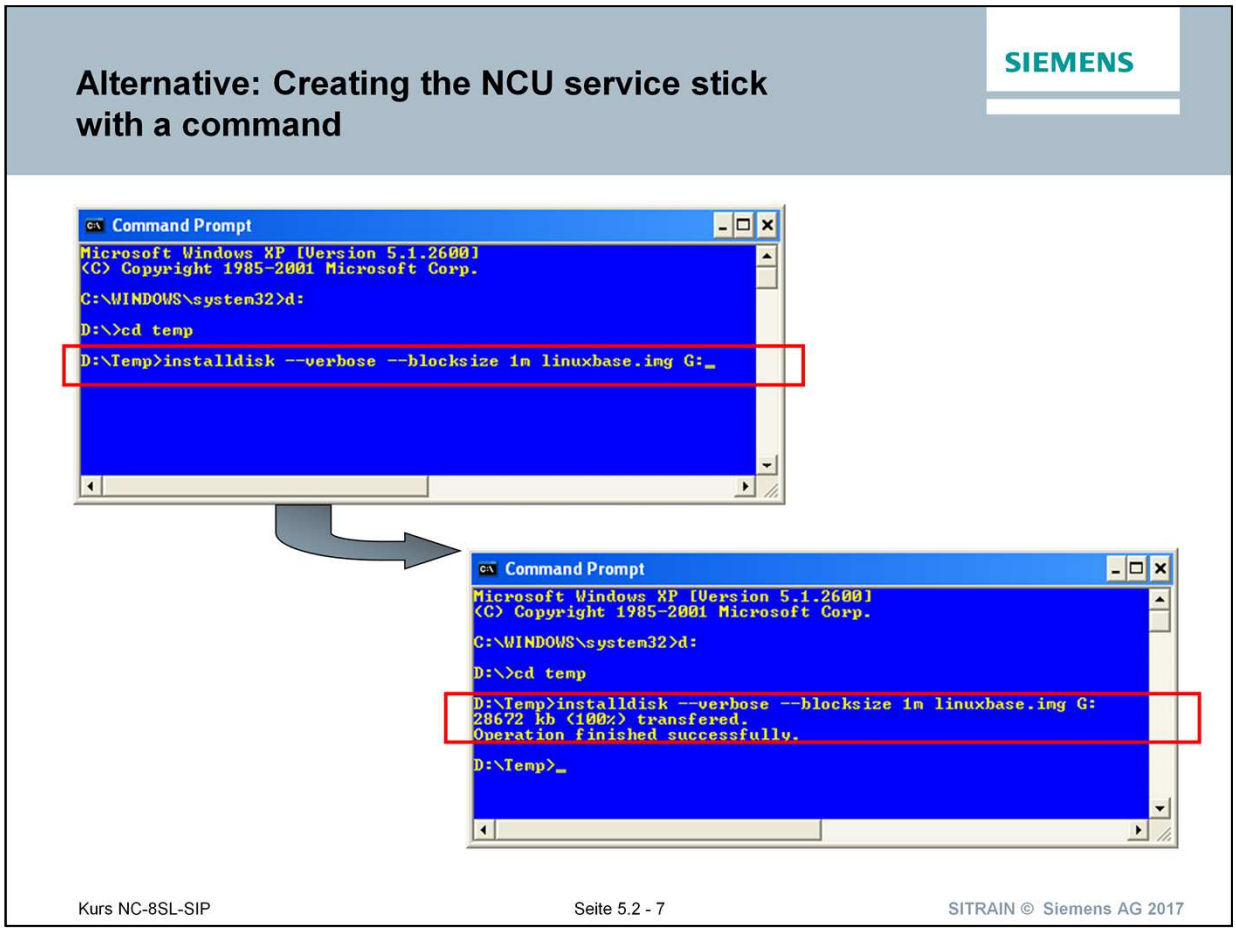
Start AMM /P2P

Write USB stick:

-> Tools -> Writing image to CF card

Select file: **linuxbase.img**

Select USB stick drive **NOTICE: Drive is overwritten!**



Procedure for creating the NCU service stick with the command shell

Connection of USB boot device

Connect the USB device to the PG (USB stick, card reader with CF card, etc., for example).

Check drive

Check drive letter.

Call the command line box via **Start -> Run -> cmd**

Navigate to the directory **D:\temp** containing the files you require:

- installdisk.exe
- linuxbase.img

Find the drive letter for the USB stick that is plugged in.

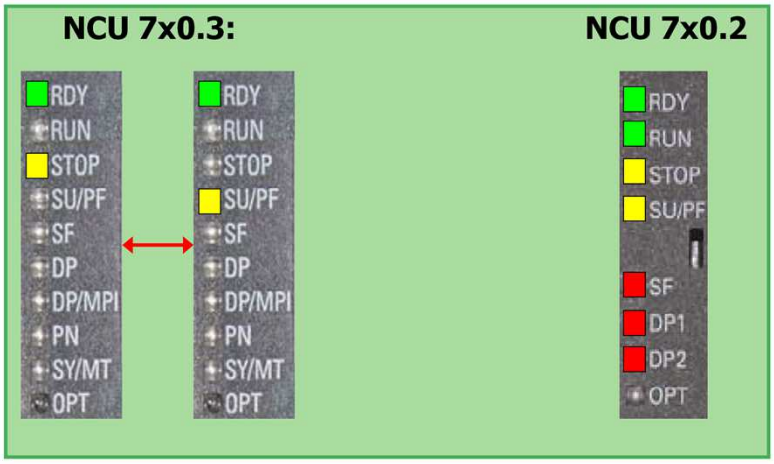
Command

Enter **Installdisk --verbose --blocksize 1m linuxbase.img <drive_letter:>** where **<drive_letter:>** is the drive letter for the USB stick.

AMM /P2P

AMM /P2P can also be used to create a USB service stick.

Booting the NCU from the service stick



Kurs NC-8SL-SIP Seite 5.2 - 8 SITRAIN © Siemens AG 2017

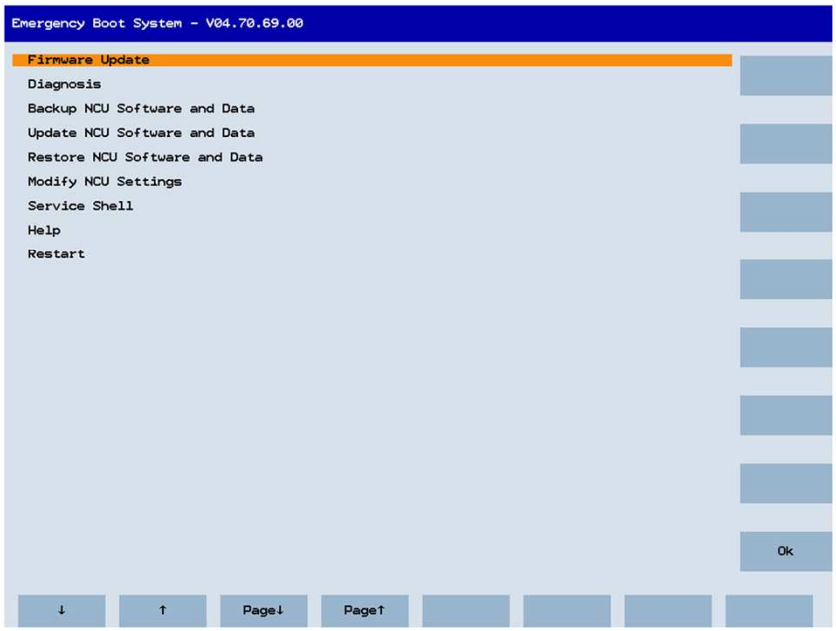
LEDs on the NCU

After inserting the boot stick in the NCU and switching on the control again, the above display is obtained, depending on the particular control type.

Note

The NCU boots up twice before the stick is ready.
The NCU service stick is now ready for use.

Main menu of the NCU service stick



SITRAIN © Siemens AG 2017

Menus

Firmware update

Diagnosis

With this selection it is possible to check the CF card e.g. for a differential diagnosis wether the CF card or the NCU is broken.

Beside reading errors also the filesystem is checked.

Backup NCU Software and Data

Backup the complete content of the CF card.

Update NCU Software and Data

Update of an existing softwar eon CF-card and complete new writeing of CF-card.

Restore NCU Software and Data

Complete rewrite of CF-card.

Modify NCU Settings

Modify different basic NCU Settings for example activate/deactivate the HMI or reinitialisation of NC,PLC and drives.

Service Shell

After a successful login with:

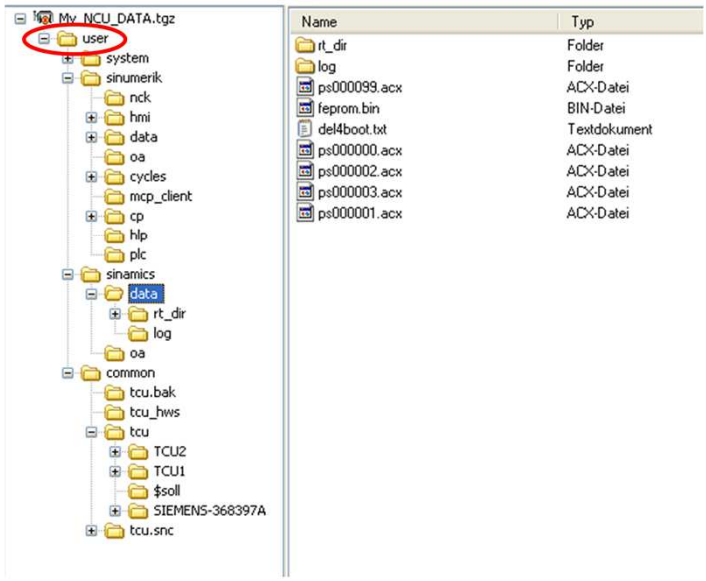
login : manufact
Password : SUNRISE

you can execute service command (sc). See chapter networking.

Restart

to restart the NCU

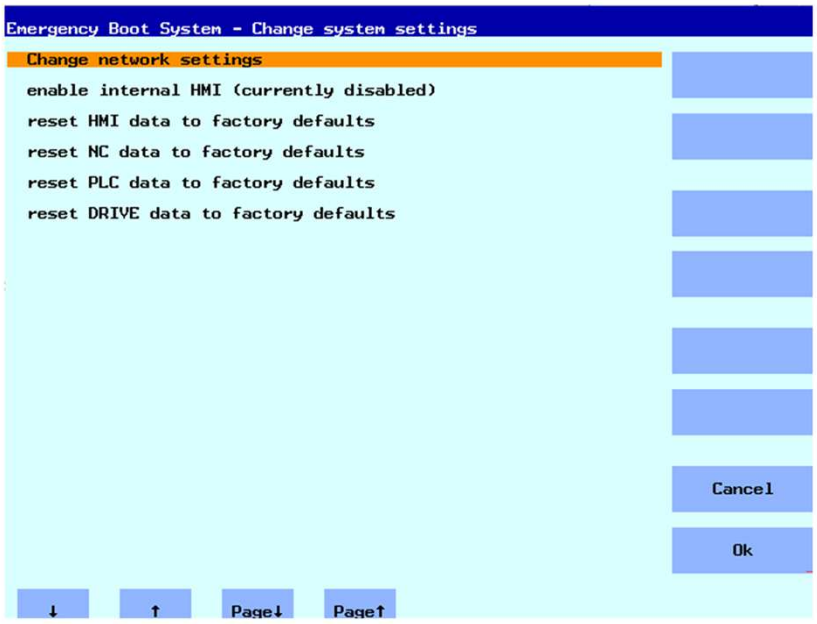
Backup user Data to USB-stick



SITRAIN © Siemens AG 2017

Backup User Data With this menu you can backup the user data path and it‘s complete content to your target drive.

Change NCU settings with the service-stick



SITRAIN © Siemens AG 2017

Modify NCU Settings

Change Network settings

Change DHCP settings of X120 and X130.

Enable internal HMI /Disable internal HMI

Activate Sinumerik Operate on NCU.

reset HMI, NC, DRIVE

Restore factory settings for a specific unit (NC/PLC /Drives/HMI) .

Service Shell



Kurs NC-8SL-SIP Seite 5.2 - 12 SITRAIN © Siemens AG 2017

After successfully logging into the service shell with the following data, for example:

login: manufact

Password: SUNRISE

service commands (sc) can be used.

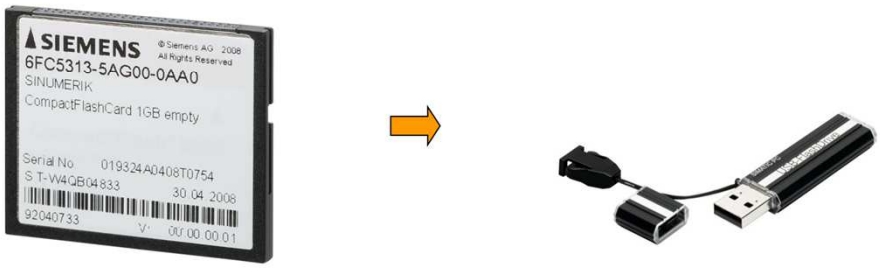
Note

Users log in via the Linux system. As with any UNIX variant, entries are case-sensitive.

This means a separate keyboard is needed when making entries on the TCU.

Or, You use CTRL+SHIFT to switch between lowercase and uppercase letters if this is working on the specific operator panel..

Exercise: Back up complete CF card to NCU service stick



Kurs NC-8SL-SIP Seite 5.2 - 13 SITRAIN © Siemens AG 2017

Objective

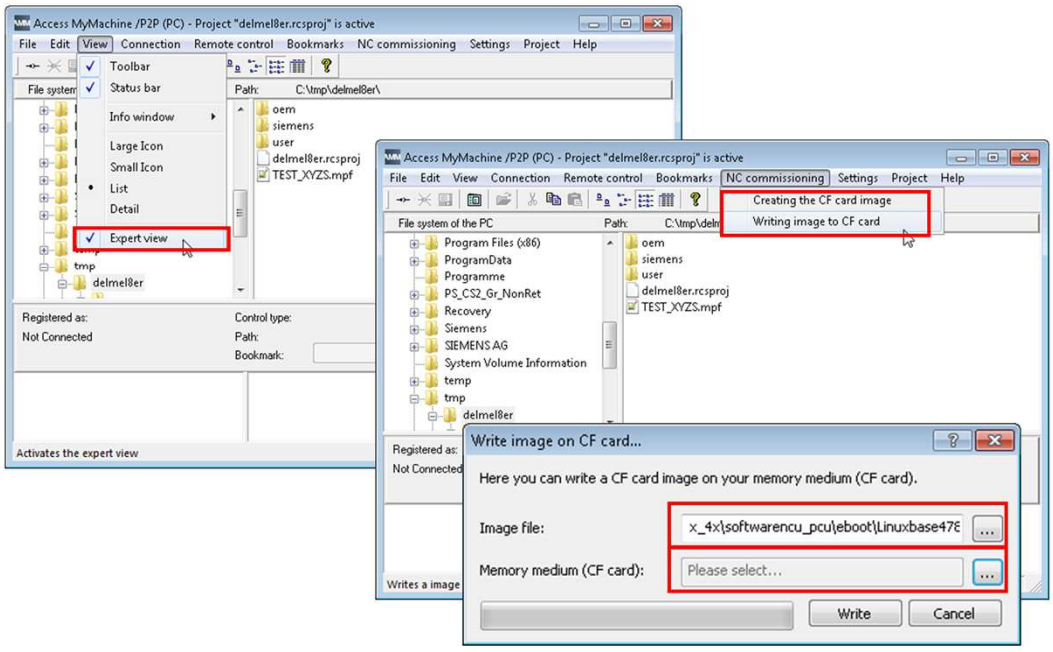
A complete copy of the CF card is to be created using the NCU service stick.

Procedure

Proceed as follows:

1. Create a Linux-bootable NCU service stick.
2. Boot the NCU with this service stick.
3. Work through the service stick menus to backup the CF card
4. Restore the CF card from the service stick.

Exercise: Create a NCU service-stick with AMM /P2P



Kurs NC-8SL-SIP Seite 5.2 - 14 SITRAIN © Siemens AG 2017

Creation of the NCU service-stick with Access MyMachine /P2P

1. Connect USB-stick to the PG/PC
2. Check which drive letter is assigned to your USB drive

Creating the stick

Start AMM/P2P
Write USB Stick :

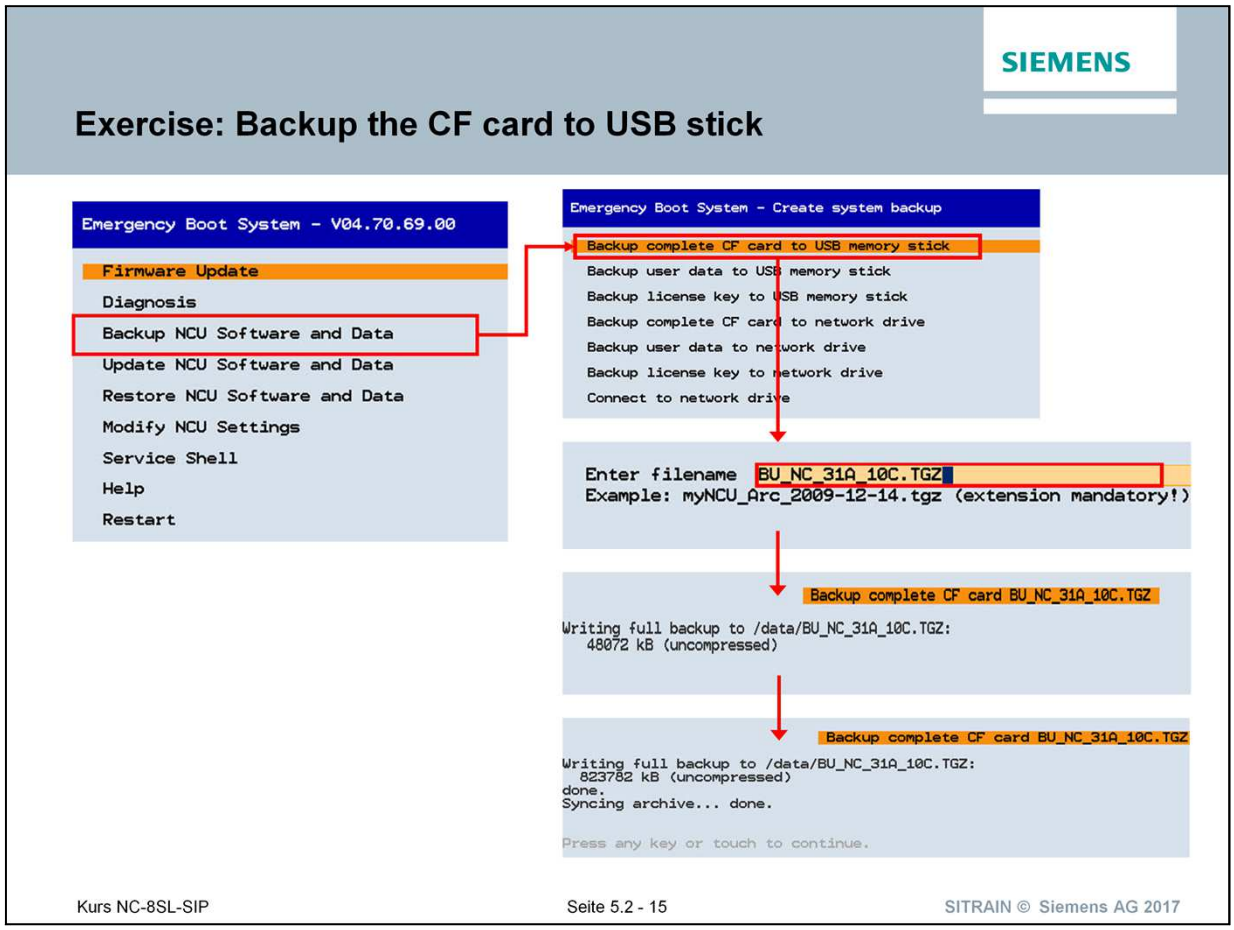
NC-Commisioning -> write Image to CF-card

Select the file: **linuxbase.img**

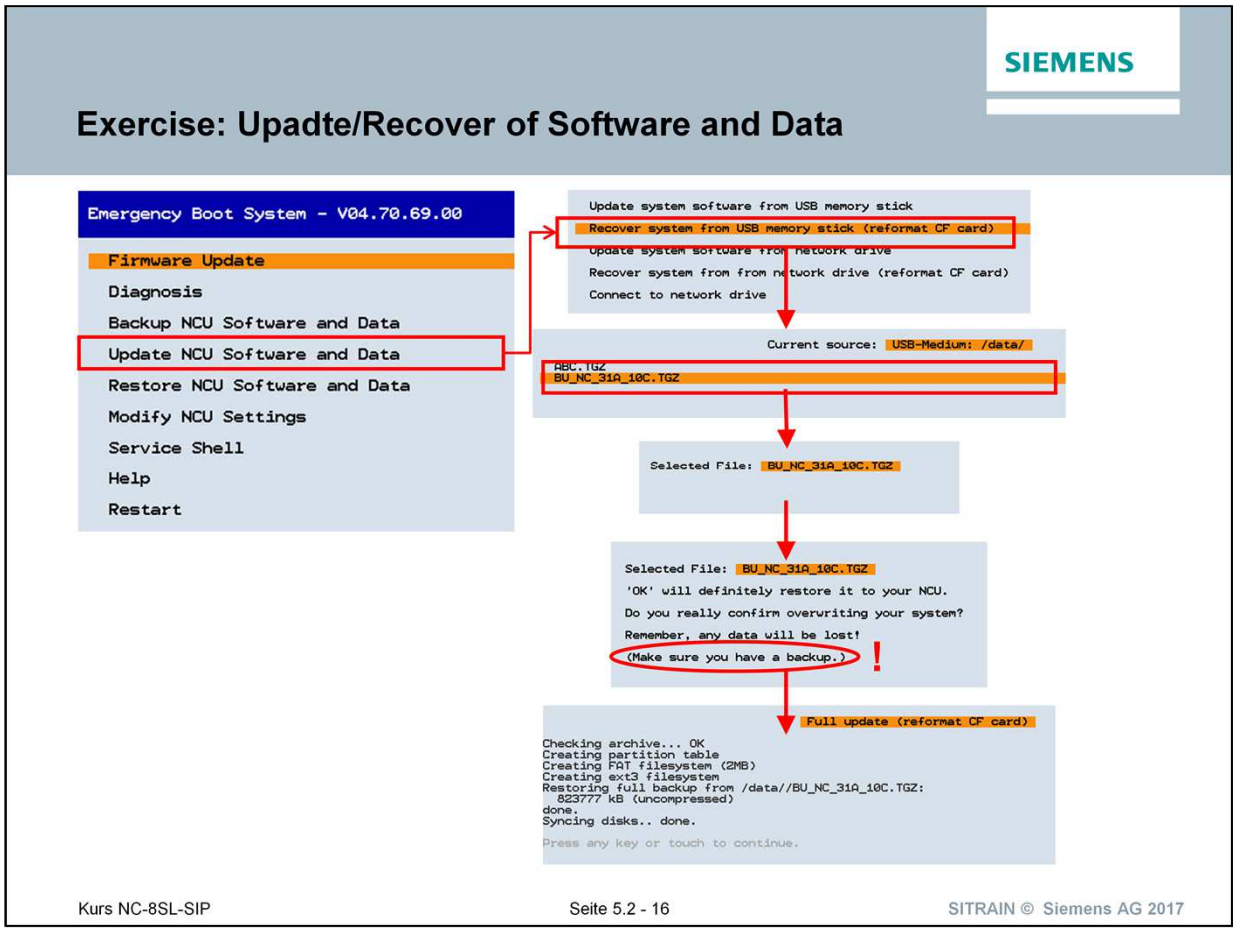
Select the right USB drive letter.

Path on PG

D:\840Dsl_SW_2x_4x\software\ncu_pcu_\eboot\eboot_ncu\eboot_xxxxxx

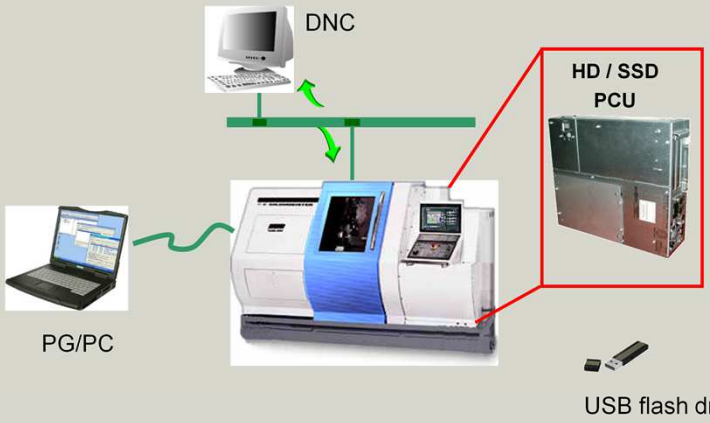


Backup	Using the Menu „Backup NCU Software und Data“ you should create a data backup. The target path for the backup could be your thumb drive or a network. It is possible to backup the complete CF-card or just parts of it. Following selections are available:
Function	Backup complete CF Card to USB memory stick
Hint	When entering the file name the extension .tgz is mandanory. The operation of the menus can be made with the cursor keys.


SIEMENS
Exercise: Upadte/Recover of Software and Data
Emergency Boot System - V04.70.69.00
Firmware Update
Diagnosis
Backup NCU Software and Data
Update NCU Software and Data
Restore NCU Software and Data
Modify NCU Settings
Service Shell
Help
Restart
Update system software from USB memory stick
Recover system from USB memory stick (reformat CF card)
Update system software from network drive
Recover system from from network drive (reformat CF card)
Connect to network drive
Current source: USB-Medium: /data/
ABC.TGZ
BU_NC_31A_10C.TGZ
Selected File: BU_NC_31A_10C.TGZ
Selected File: BU_NC_31A_10C.TGZ
'OK' will definitely restore it to your NCU.
Do you really confirm overwriting your system?
Remember, any data will be lost!
(Make sure you have a backup.)
!
Full update (reformat CF card)
Checking archive... OK
Creating partition table
Creating FAT filesystem (2MB)
Creating ext3 filesystem
Restoring full backup from /data//BU_NC_31A_10C.TGZ:
823777 kB (uncompressed)
done.
Syncing disks.. done.
Press any key or touch to continue.
Kurs NC-8SL-SIP
Seite 5.2 - 16
SITRAIN © Siemens AG 2017

Notes

SIEMENS



Chapter 5.3

Data backup of PCU50 with Ghost

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Hard disk backup/restore with Ghost 3
Partitions in PCU Windows 7 4
Windows 7 Ghost backup 5
Boot from USB and backup to an external hard disk 11
Booting from USB 12
Exercise: Windows 7 Ghost backup 13
Exercise: Create EBOOT flash drive for PCU 17
Appendix: Windows XP Ghost backup 22
Partitions in PCU Windows XP 23
Backup/restore of the PCU 50 to/from network drive 24
Rollback and emergency image 26
Backup to an external USB hard disk 27
Backup with ServiceCenter of the PCU to USB drive 28
Restore with ServiceCenter of the EBOOT flash drive Image from USB hard disk 29
Restore with ServiceCenter of the EBOOT flash drive Image from EBOOT flash drive 30
Instructions: Booting from USB 31
Exercise: Start Windows XP on PCU 32
Exercise: Backup with ServiceCenter of the PCU 50 to network drive 33
Exercise: PCU backup – Complete and check 36
Exercise: Restore with ServiceCenter of the PCU 50 from network drive 37
Exercise: Restore PCU – Complete and check 39

Learning objectives

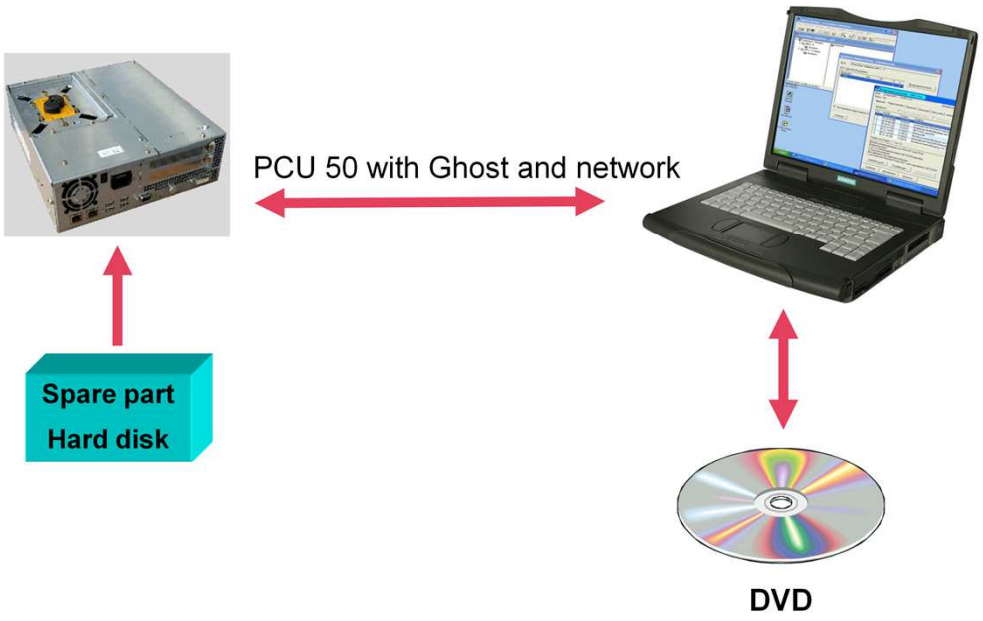
- You will be able to perform a backup / restore with Ghost
- You will be familiar with data backup using a USB medium
- You will be able to create an emergency boot flash drive.



SITRAIN © Siemens AG 2017

Notes

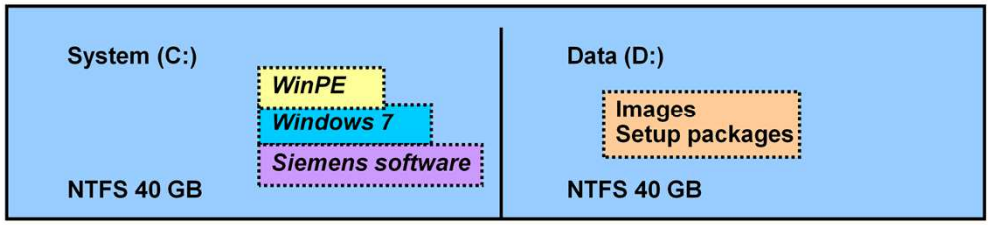
Hard disk backup/restore with Ghost



Course NC-84SL-SIP Page 5.3 - 3 SITRAIN © Siemens AG 2017

Ghost	The backup program Norton Ghost is installed on every PCU 50.
Functions	Ghost provides user-friendly means of backing up/restoring PCU 50 hard disks. Operating system software, add-on software, and user-specific settings are saved in their entirety. Transmission for the PCU 50 is implemented via Ethernet or USB. The hard disk image (saved as file) can be saved to an external data medium (e.g. CD) for long-term storage.
Restore	Restore is the loading of image files to an unformatted hard disk. The new target disk can be larger or smaller than the original (as long as there is enough space for the data, of course). When you copy hard disks with multiple partitions, partition sizes may be changed.
Ghost Explorer	The Norton Ghost Explorer is a tool for handling image files. It allows you to display the file structure. Individual backup files can also be extracted. For more information, see http://www.ghost.com.
Note	With a more recent version of Ghost Explorer, old images can also be opened, but not vice versa.

Partitions in PCU Windows 7



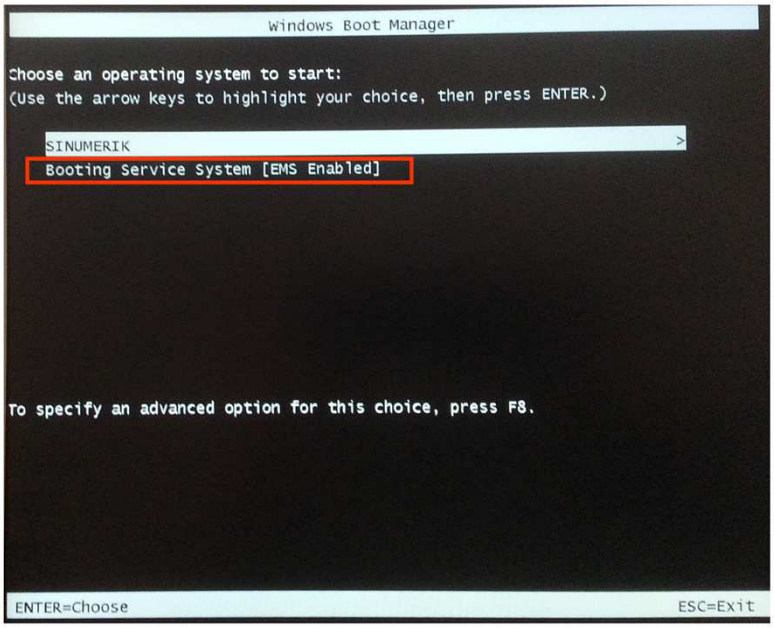
Course NC-84SL-SIP Page 5.3 - 4 SITRAIN © Siemens AG 2017

Partitioning the hard disk

- The hard disk is divided into 2 NTFS partitions.
- The operating software, Windows XP and Win PE 2005 run on the same partition.

Windows 7 Ghost backup
Starting the Service Center – Boot menu

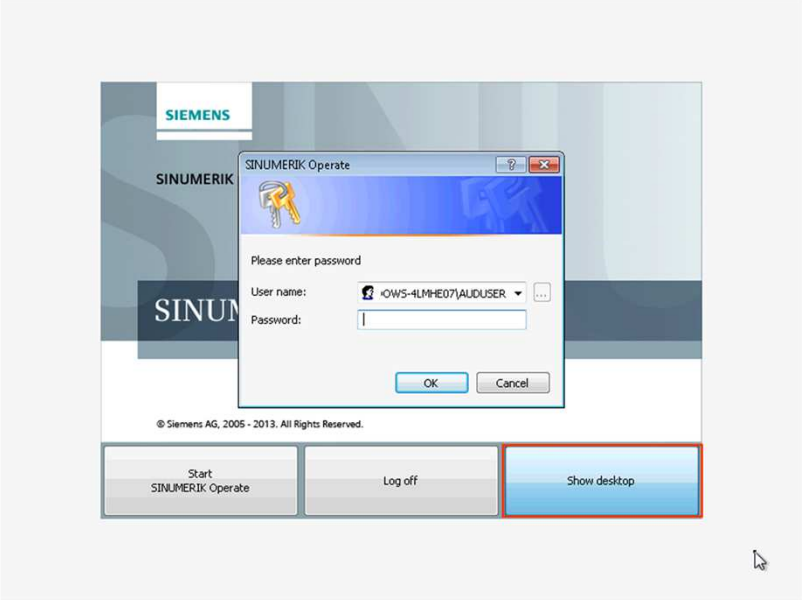
SIEMENS



SITRAIN © Siemens AG 2017

If your PCU50.5 / Windows7 operating computer is mounted directly on an operator panel, you can select the menu time "Booting Service System..." when the system starts up, which starts the Service Center.

Windows 7 Ghost backup
Desktop logon screen Windows 7 PCU



Course NC-84SL-SIP Page 5.3 - 6 SITRAIN © Siemens AG 2017

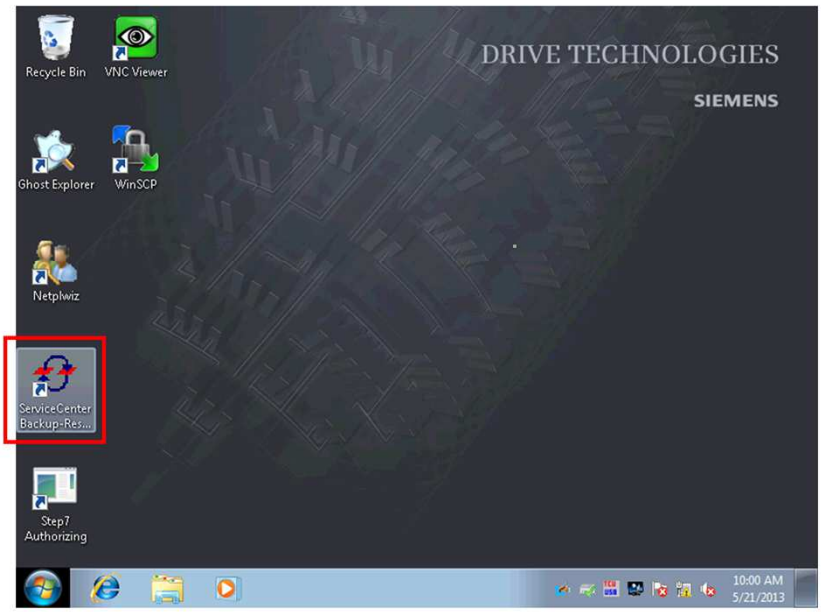
Windows Desktop

If you start the Service Center from the Windows desktop, press key 3 while the Windows/Sinumerik Logo is shown during startup or put the service switch (rotary switch under the 7-segment display) into position "3" before you switch on.
Then select "Show Desktop". You then enter the correct password to start the Windows desktop, from which you call the Service Center.

Password

The default password is "SUNRISE".

Windows 7 Ghost backup
Service Center

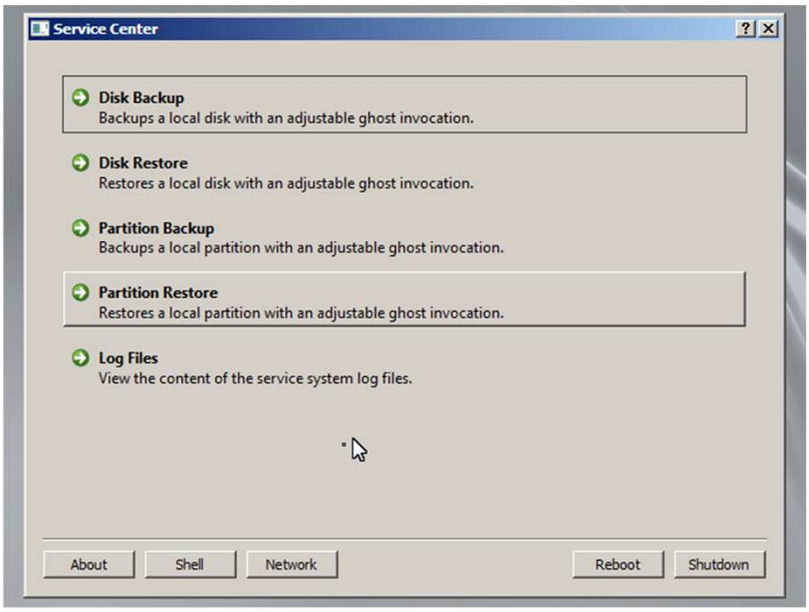


SITRAIN © Siemens AG 2017

You have arrived at the Windows 7 desktop and you can start the Service Center by double-clicking the icon.
The PCU50 is restarted and boots the Service Center.

Windows 7 Ghost backup
Service Center Backup/Restore

SIEMENS



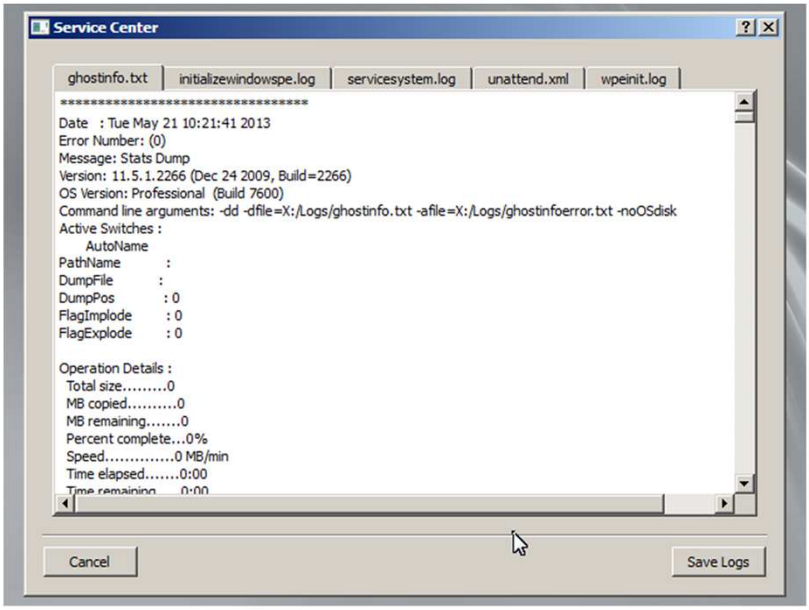
 SITRAIN © Siemens AG 2017

When you click the Service Center icon on the desktop, the PCU 50 immediately restarts. (Agree when Windows 7 ask you to confirm the change.)
The screen shown above appears.
5 menu items can be chosen in this screen.

Disk Backup	Backs up the entire hard disk.
Disk Restore	Loads a Ghost image file back onto the built-in hard disk.
Partitions Backup	Backs up a selected partition onto a target drive.
Partitions Restore	Loads a Ghost image file of the partition backup back onto a partition of the built-in hard disk.
Log Files	Contains the display of different LOG files.

Windows 7 Ghost backup
Log files

SIEMENS

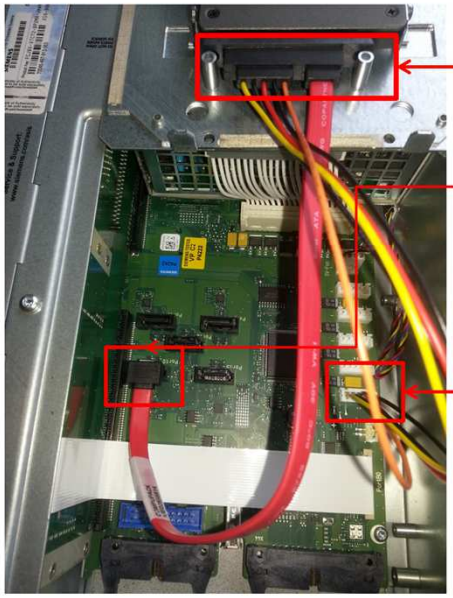


SITRAIN © Siemens AG 2017

The "Logfiles" button takes you to a number of log files that can be displayed here.
They contain information, e.g. about initializing and starting Windows PE, but also, for example, about the version of Ghost used.
The most interesting thing relating to the Ghost data backup is certainly the servicesystem.log in which errors that occurred during data backup are logged. A reference to the log file may appear.

Windows 7 Ghost backup
SSD Disk in PCU 50.5

SIEMENS



Connections of the hard disk

SATA connection for the hard disk

Supply voltage of the hard disk

View of the hard disk connection in PCU50.5. The SATA connector and power supply are connected as in the delivered condition here.

Partition C:	Partition D:
Windows system User interface	Installation folder EBoot directory

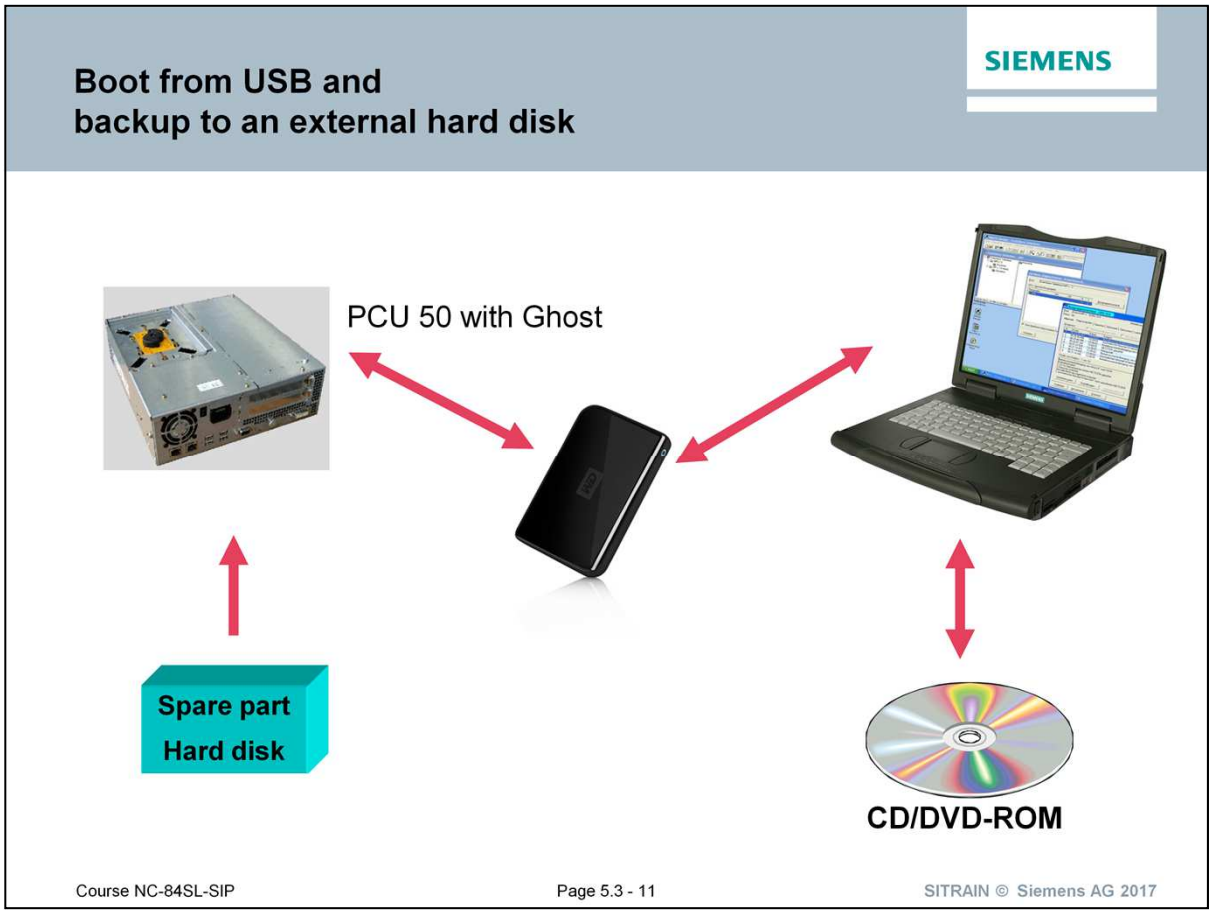
Course NC-84SL-SIP Page 5.3 - 10 SITRAIN © Siemens AG 2017

Only the PCU50.5 for Windows 7 is permissible. The SSD has a size of 80 GB; the RAM has 8 GB.

The hard disk (SSD) of the PCU50.5 is divided into 2 partitions for Windows 7. (A further partition Win7 is usually set up as "System reserved" – DO NOT REMOVE THIS!! Among other things, it contains the boot manager.)

Replacement

If you replace the hard disk of the PCU 50.5, make sure that you use the same hard disk connection as the one to which the removed hard disk was connected. Make sure that the SATA connection engages correctly. Otherwise, it may be shaken loose by vibrations.



Note

A SIMATIC-USB flash drive can be made bootable for the PCU. This is done by installing the image located on the PCU under D:\ebbot\eboot.gho with the program ghost32.exe on the flash drive. If a USB boot medium is connected, it is used to boot the system.

If the USB medium is an eboot drive of sufficient size (e.g. an external hard disk), a data backup directly to it can be performed.

Boot from USB



③ **Service switch**
Position E: Boot from USB

Course NC-84SL-SIP Page 5.3 - 12 SITRAIN © Siemens AG 2017

Boot from USB Set the service switch on the PCU 50.5 to position '**E**'.

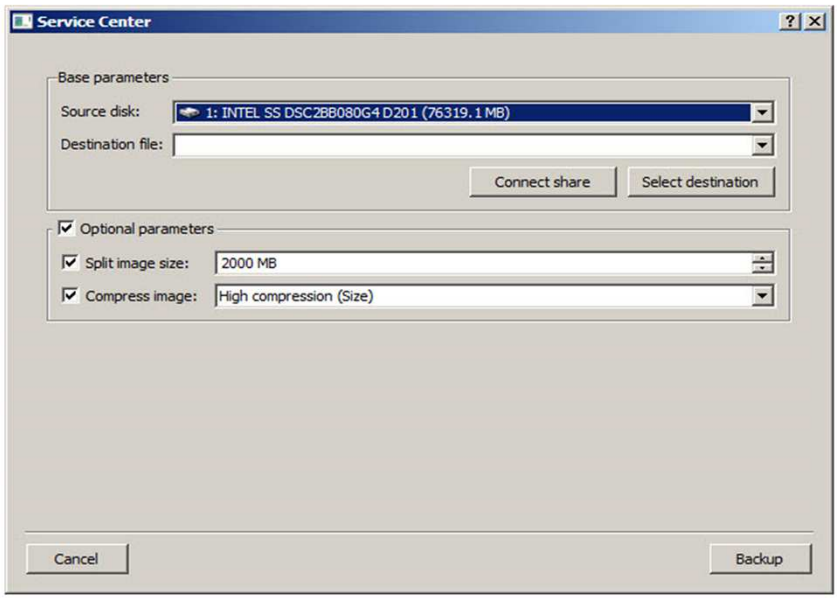
Boot from SSD Set the service switch on the PCU 50.5 to position '**0**'.

Boot from USB application

You must proceed as follows for a Ghost restore, for example:

1. Switch off the PCU, set the service switch -> **E**.
2. Connect the USB boot flash drive (to any port).
3. Switch on the PCU and wait for the Service menu to appear.
4. Perform the Ghost restore.
5. Switch off the PCU (do not reboot).
6. Service switch -> **0**
7. Remove the USB boot medium.
8. Switch on PCU

Exercise: Windows 7 Ghost backup

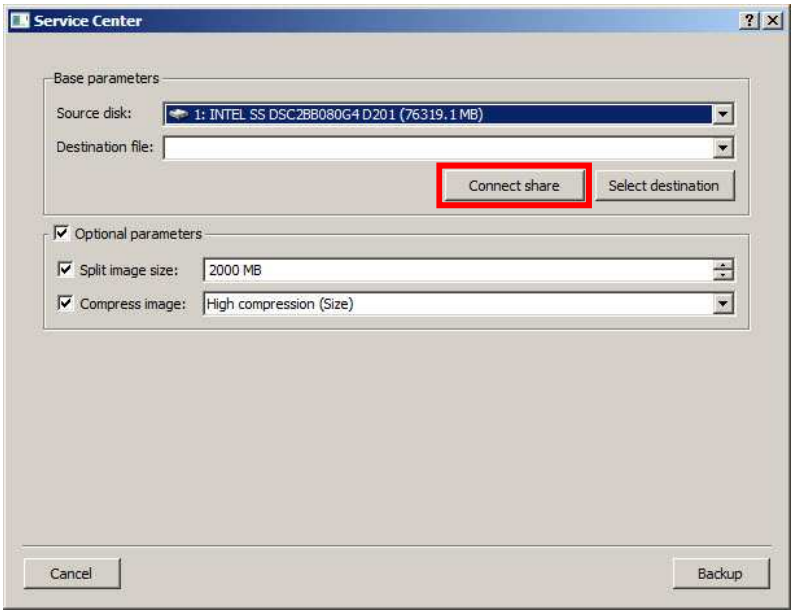


 SITRAIN © Siemens AG 2017

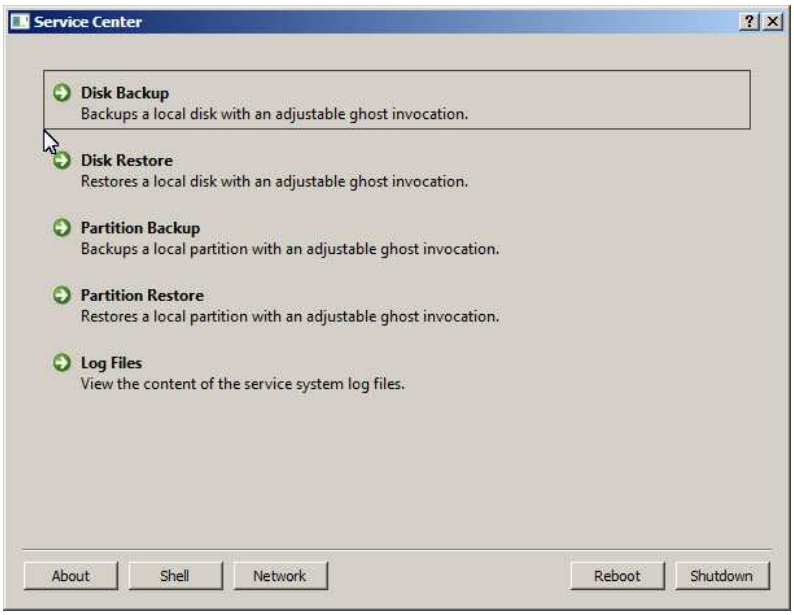
Procedure

1. Open Windows Desktop
2. Open Backup/Restore Service Center
3. Connect network drive
4. Name files
5. Start backup

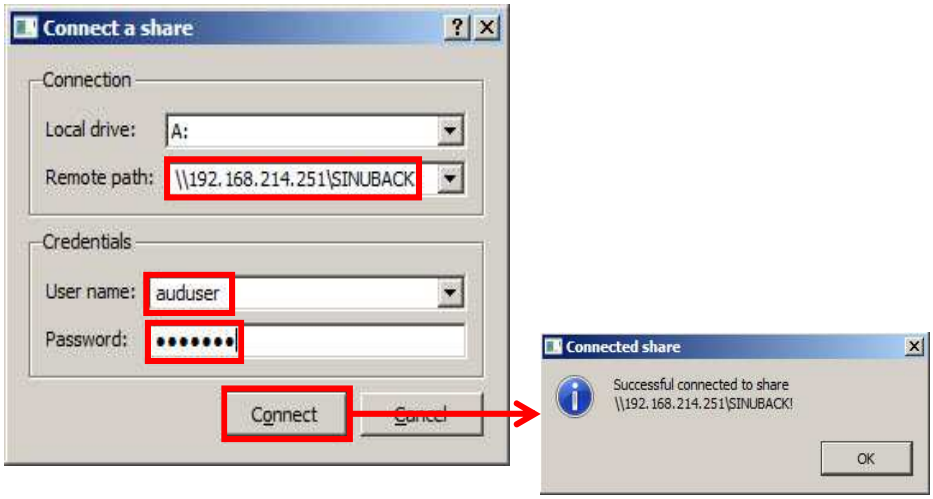
1. Select the Disk Backup menu item



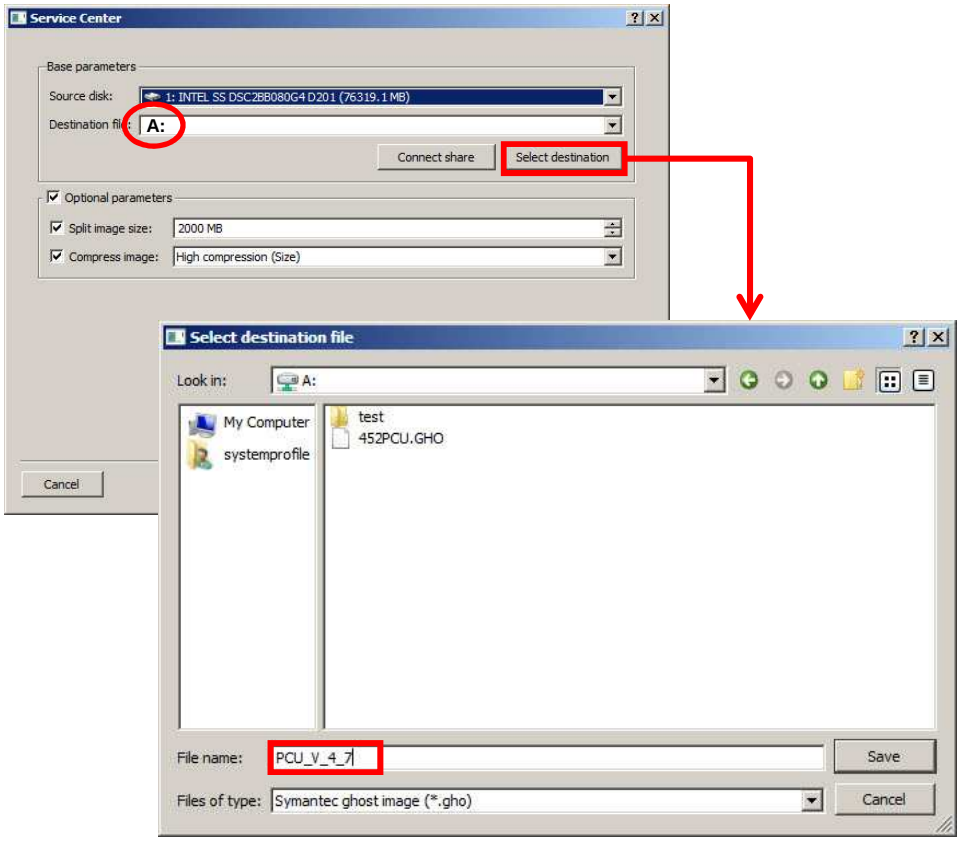
2. The source hard disk is displayed, therefore first connect a network drive



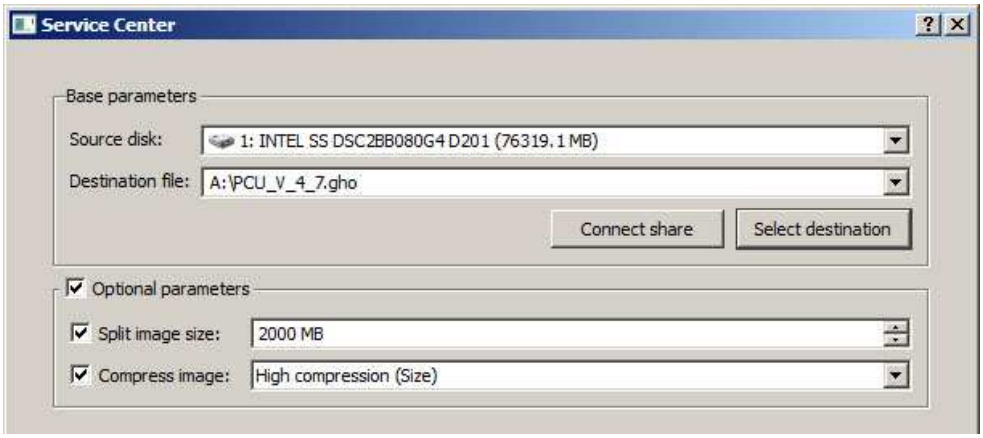
3. Enter the computer name or the IP address of the target computer and the share names of the target drive in the "Remote path" as shown in the screenshot.
   The user name and the password of the user of the write privileges must also be entered on the target drive.
   With "Connect", the drive is connected.

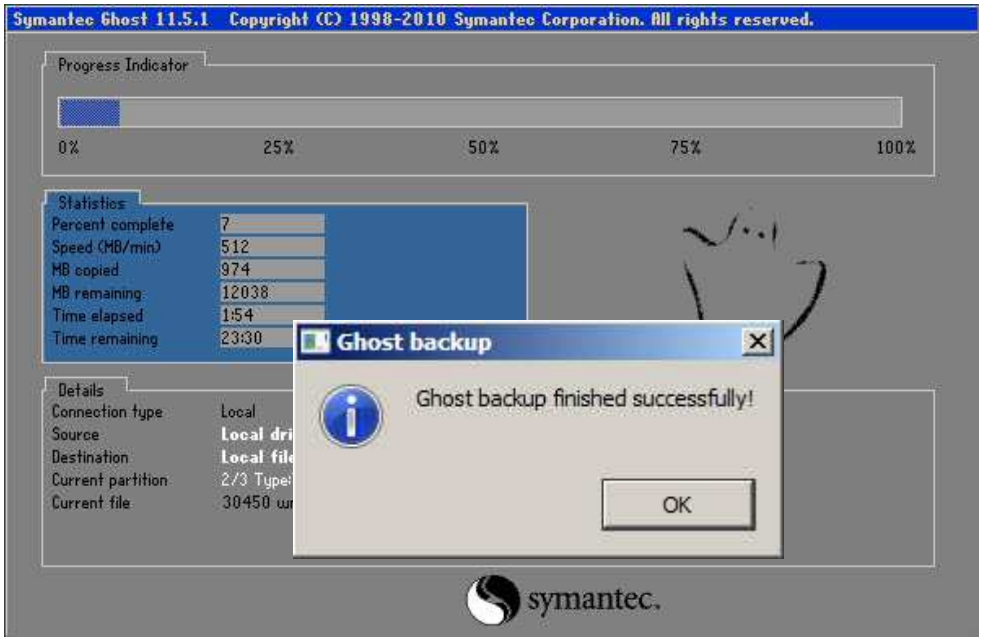


4. The connected directory will be shown as the drive (here "A:")
   With "Select Destination", a file name can be entered and confirmed with "Save".

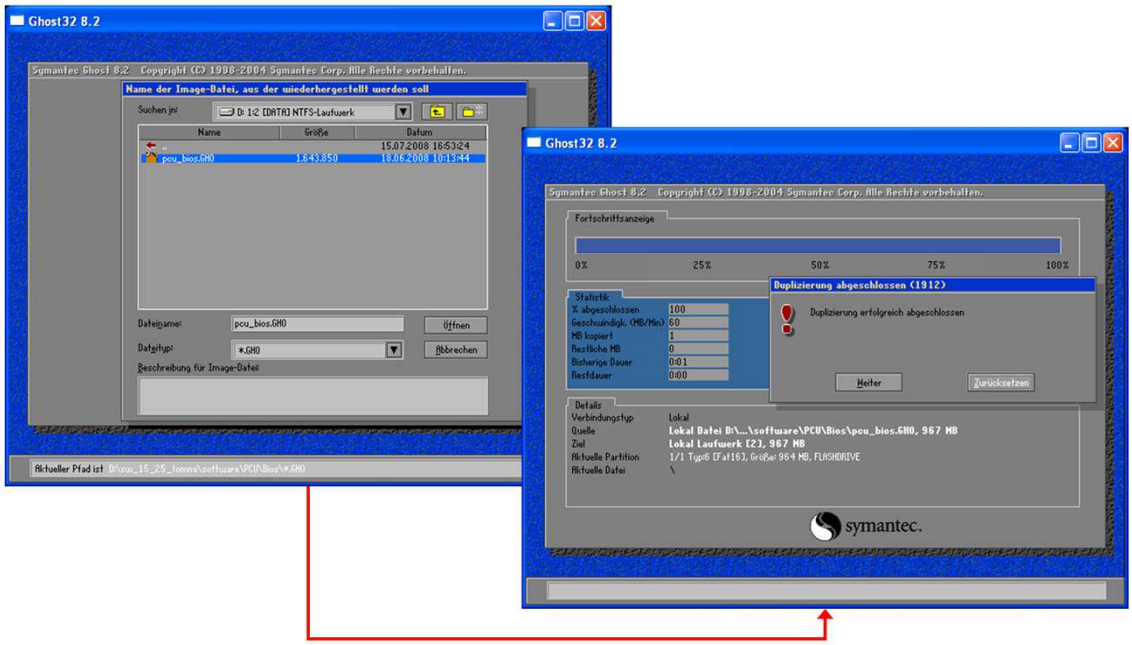


5. The file name now appears as the target file.
"Backup" starts backing up the hard disk.
With "Split image size", the backup file can be split into smaller subfiles, e.g. to be able to spread it across multiple data media.
With "Compress image", the file can be compressed and the standard setting "high compression" can be retained.





Exercise: Create EBOOT flash drive for PCU



Objective

To create a USB flash drive from a Ghost Image for an emergency PCU boot (=EBOOT flash drive).

Preconditions

Ghost32.exe and eboot.gho are available.

The image is located on the PCU:

Windows XP PCU **E:\eboot\eboot.gho**

Windows XP PCU **E:\tools\ghost32.exe**

Windows 7 PCU **D:\eboot\eboot.gho**

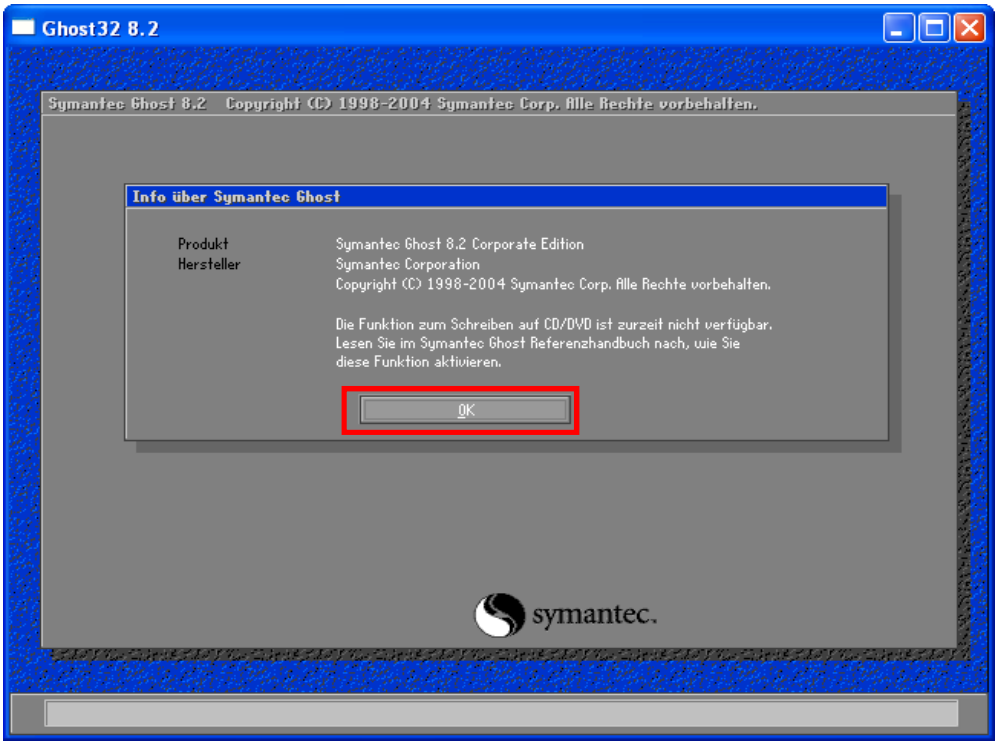
Windows 7 PCU **Ghost32.exe on C:\programme x86\siemens\MotionControl\siemens\system\etc\ghost32.exe**

Step

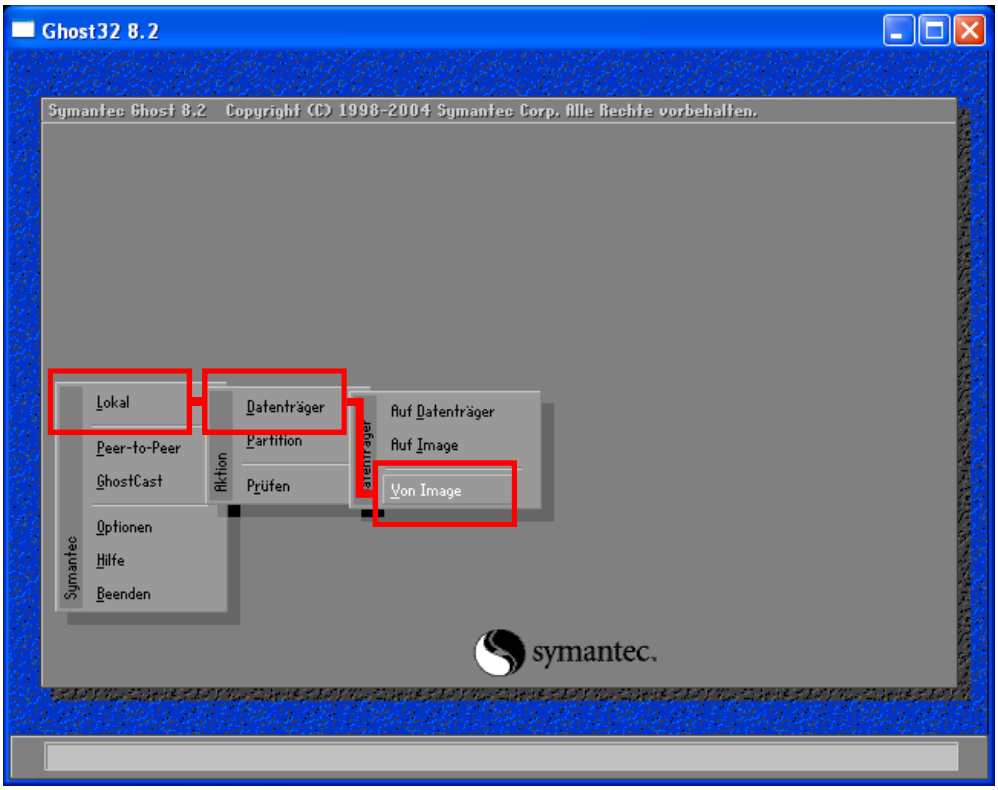
1. Start Ghost32.exe
2. Choose Restore: **Data medium -> From image**
3. Choose image: **eboot.gho**
4. Select target drive
5. Confirm details about the target drive (partitioning)
6. Confirm overwrite
7. Confirm the completion message
8. Exit Ghost

Procedure:

1. **Start Ghost32** -> Ghost32 -> OK

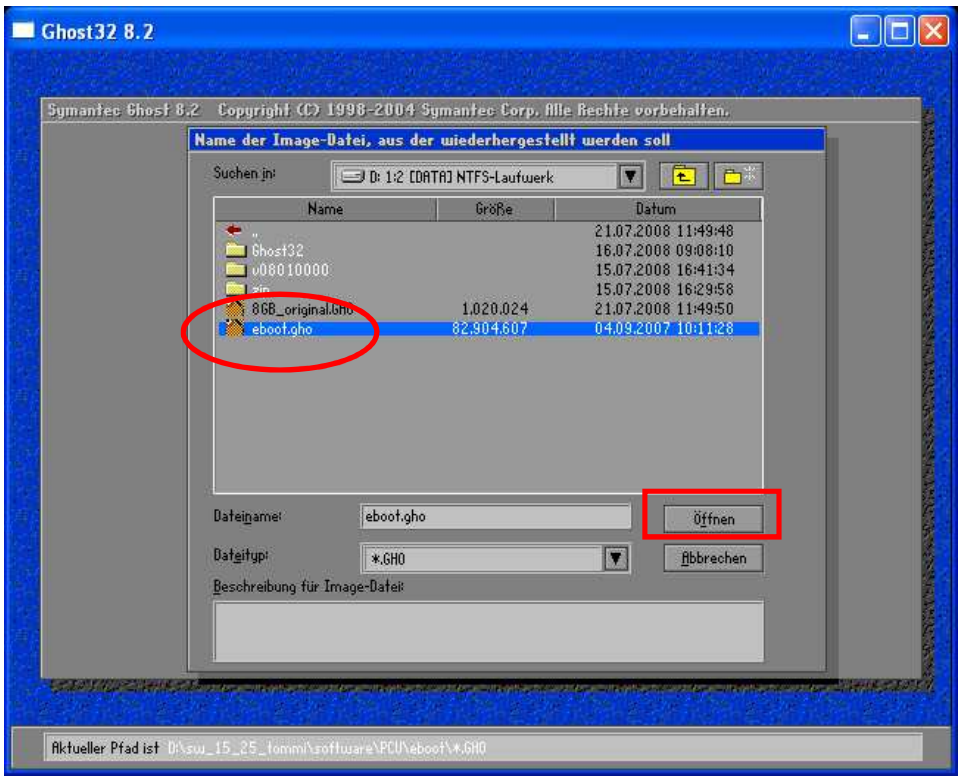


2. Choose Restore -> Local -> Data medium -> From image



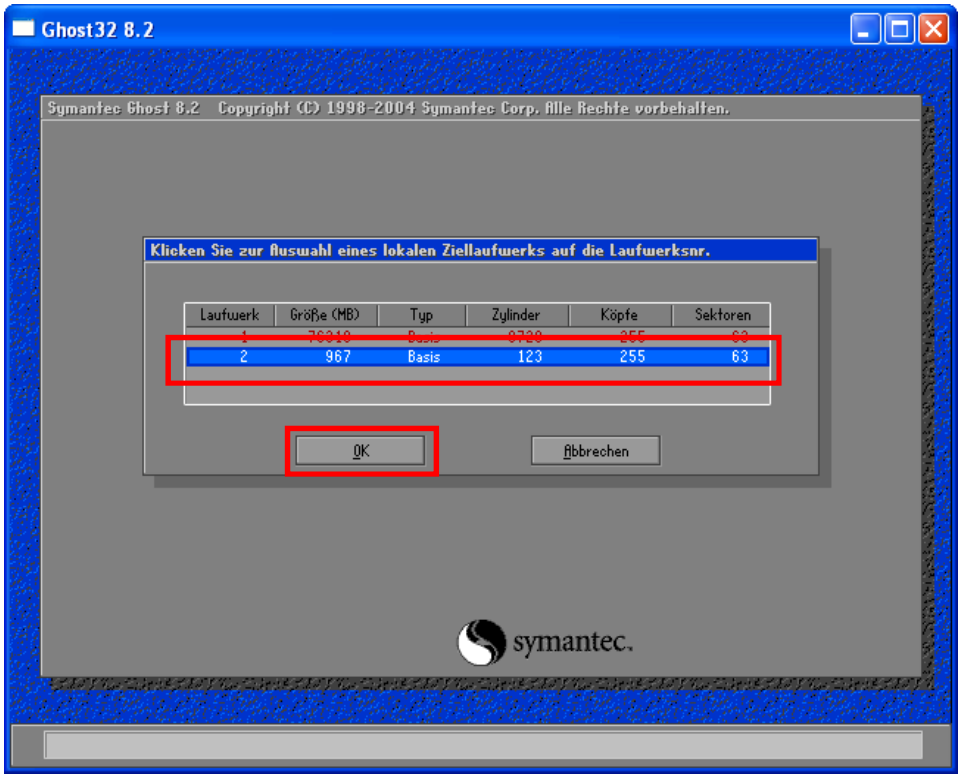
3. Select image Select the file:

eboot.gho



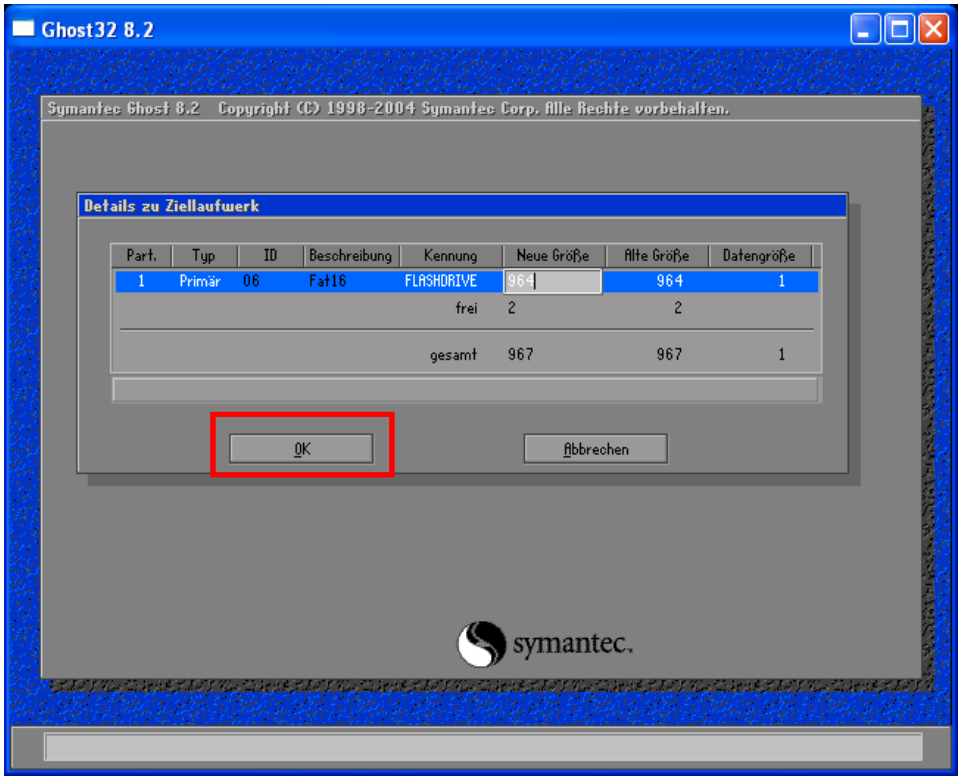
4. Select target drive Select the target drive:

Drive 2 (USB flash drive)



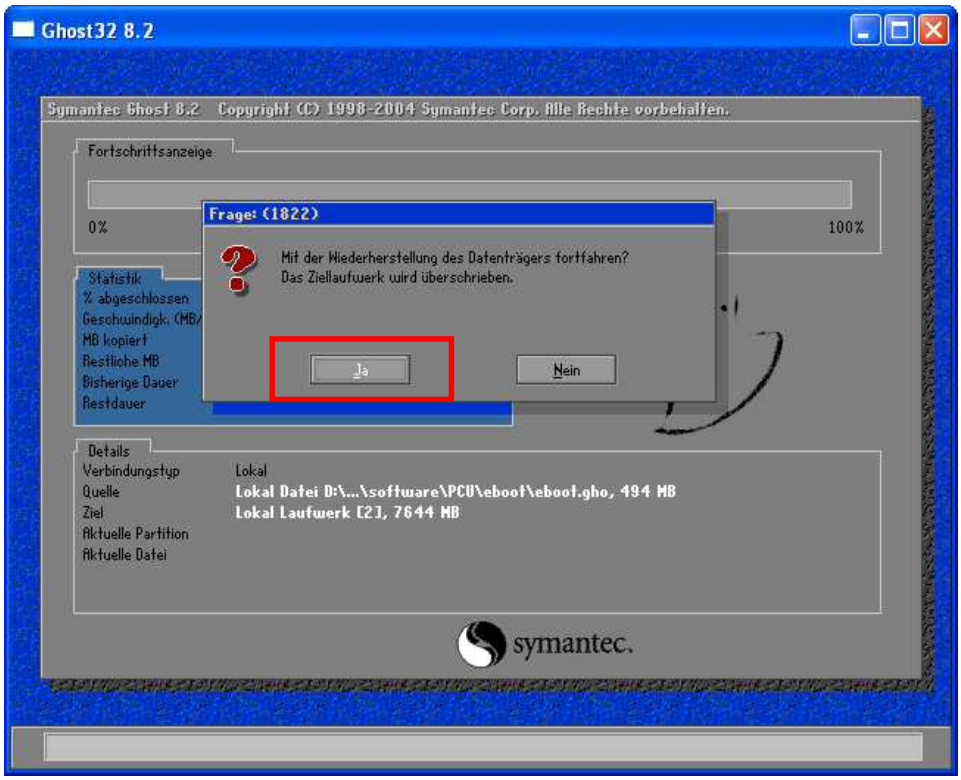
5. Confirm details about the target drive

Confirm destination drive details



6. Confirm overwrite

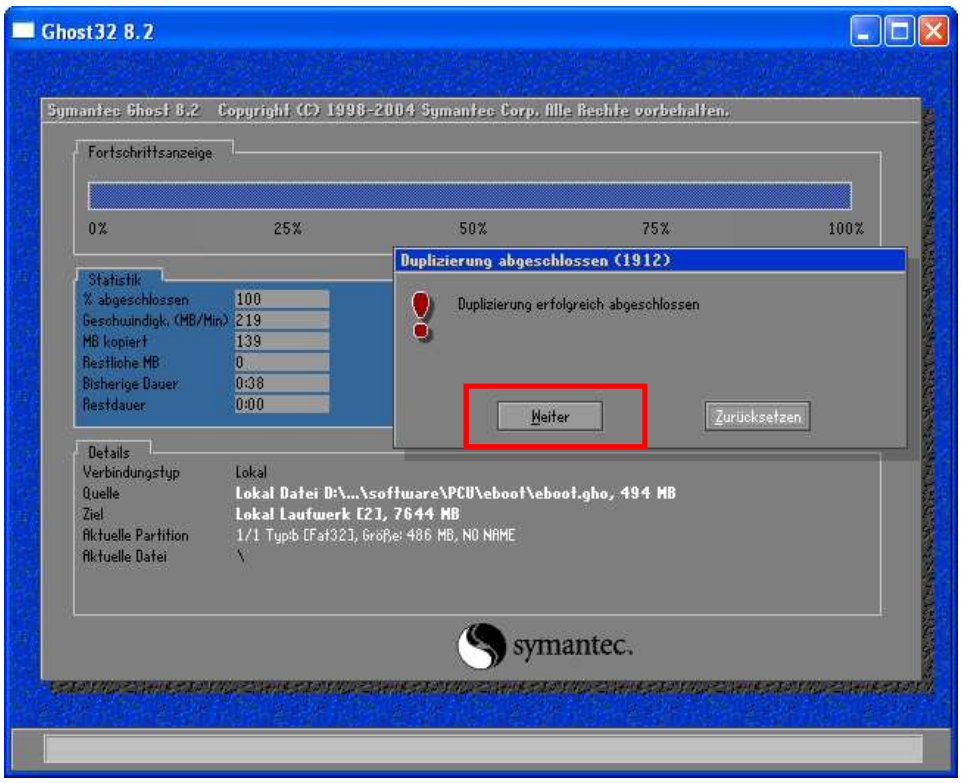
Confirm that you want the destination drive to be overwritten:



7. Confirm completion

Confirm the final message by selecting

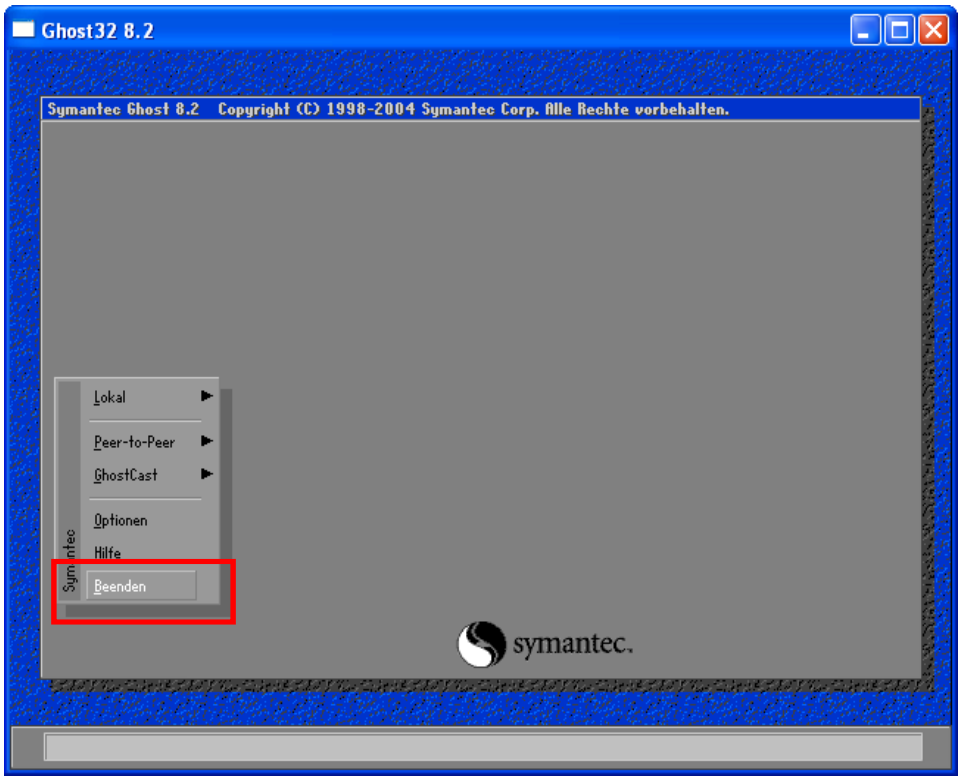
-> Next



8. End Ghost

Exit the Ghost program by selecting

-> Exit



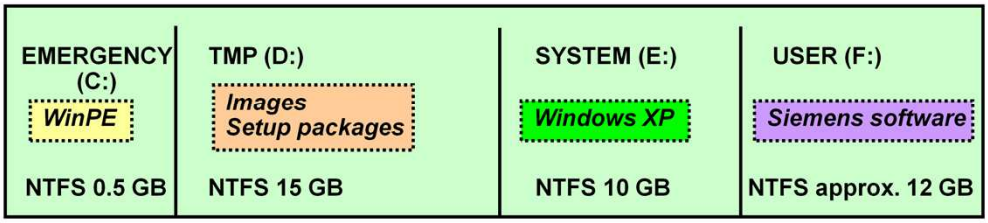
SIEMENS

Appendix: Windows XP Ghost backup



SITRAIN © Siemens AG 2017

Partitions in PCU Windows XP



 SITRAIN © Siemens AG 2017

Partitioning the hard disk

The partitioning is different depending on the operating system:

- Windows XP
- The hard disk is divided into 4 NTFS partitions.
- The operating software, Windows XP and Win PE 2005 run on different partitions.

- Windows 7
- The hard disk is divided into 2 NTFS partitions.
- The operating software, Windows XP and Win PE 2005 run on the same partition.

Windows

After booting Windows, the version of the HMI base software is displayed in the bottom right-hand corner, e.g. 'V 01.03.00.00'

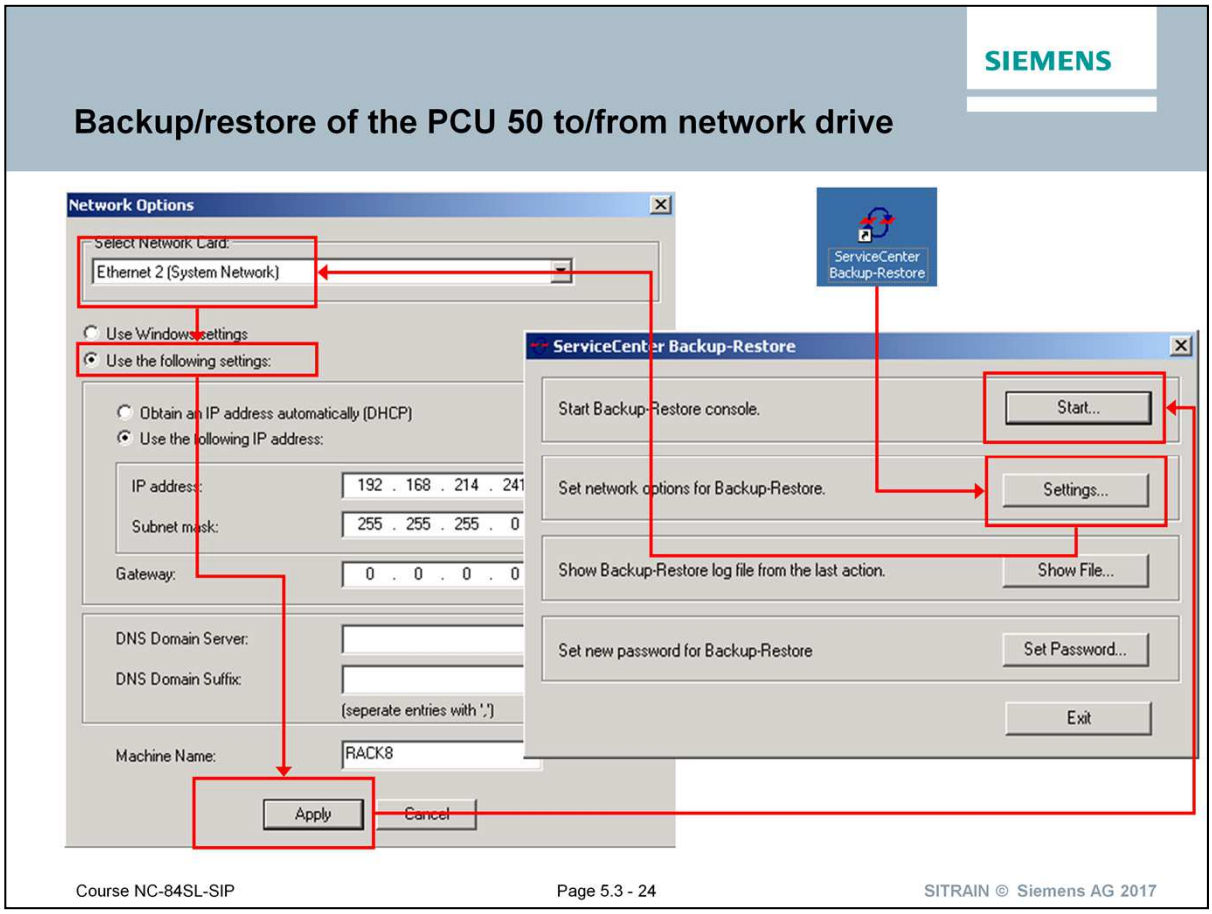
At this moment you can enter the number
-> 3

with password **-> SUNRISE**
and button -> **SINUMERIK Desktop**

to switch to the standard Windows interface.

Note

When the system is starting (after the BIOS messages) you can select either "SINUMERIK" or " " for booting the service system.



Booting the PCU **from C: of the PCU hard disk**

Target drive: **Network drive**

ServiceCenter Settings

On the SINUMERIK desktop, you will find the link to the 'ServiceCenter" program. Double-click the link to open the ServiceCenter dialog box.

Click the button labeled:

-> Settings

to access the 'Network Options' dialog box.

Network Options

Now select the interface:

- **Ethernet 2**

The easiest selection for the IP address settings is

- **Use Windows settings**

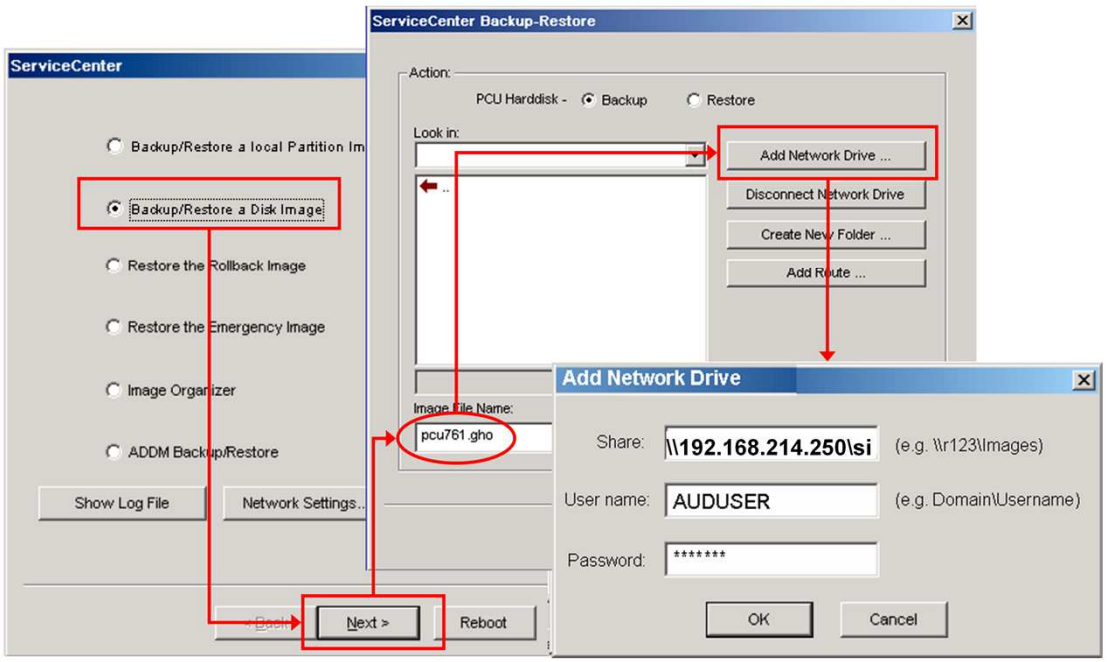
Then there is no need to specify the IP address and computer name.

- As an alternative, you could specify
  - For Ethernet 2 the IP address **192.168. 214. 241**.
  - The 'Subnet mask" is usually **255.255.255.0.**
  - Enter the computer name of the PCU under 'Machine Name'.

Start

The **-> Start** button in the higher-level dialog starts the 'Backup/restore console' via a reboot.
There, a Windows PE 2005 runs on a RAM disk. You must first wait until all drives have been loaded and user programs have started.

Backup/restore of the PCU 50 to/from network drive



ServiceCenter

In the ServiceCenter under Windows PE, you can choose between

- **Backup/Restore a local Partition Image**
  Back up/restore local partition image
- **Backup/Restore a Disk Image**
  Back up/restore whole disk
- **Restore the Rollback Image**
  Restore latest images
- **Restore the Emergency Image**
  Restore image of the Windows system partition
- **Image Organizer**
  Manage images

Backup/Restore a Disk Image

In this case, the dialog box shown above is displayed when you press the -> **Next** button. There you must choose
-> Backup or **-> Restore** .
The filename of the image file must also be specified under

- 'Image File Name'. (extension: **.gho** !)

Add Network Drive

The other entry fields are only displayed once the Add Network Drive dialog box has been selected. They mean:

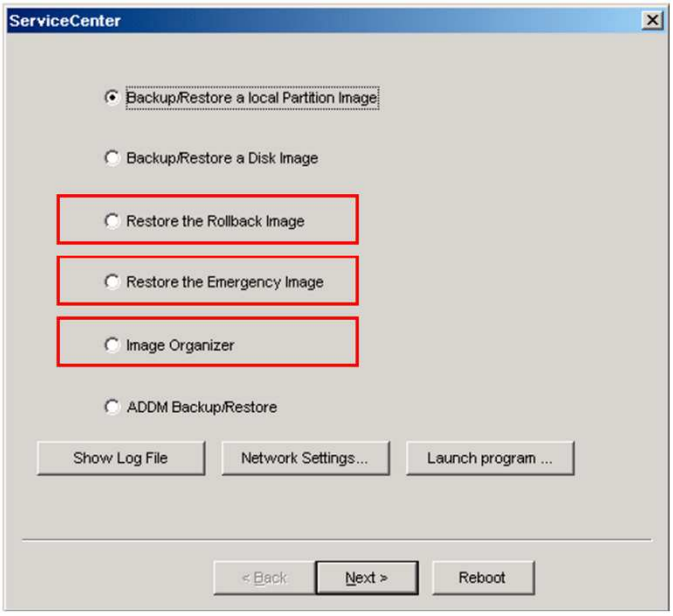
- 'Share' external computer (=PG) and image path with the form
  \\Computer_name\Share_name or
  \\IP_address\Share_name
- **'User name'** a valid logon name on the external computer (=PG)
- '**Password**' a suitable password.

Options

When you click the **-> Next** button, an overview of the selected settings is displayed.
Go to **-> Options**, activate the 'Splitting' function, and set the size of the individual files under 'Splitsize'.

Clicking -> **Next** will start the backup/restore operation.

Rollback and emergency image

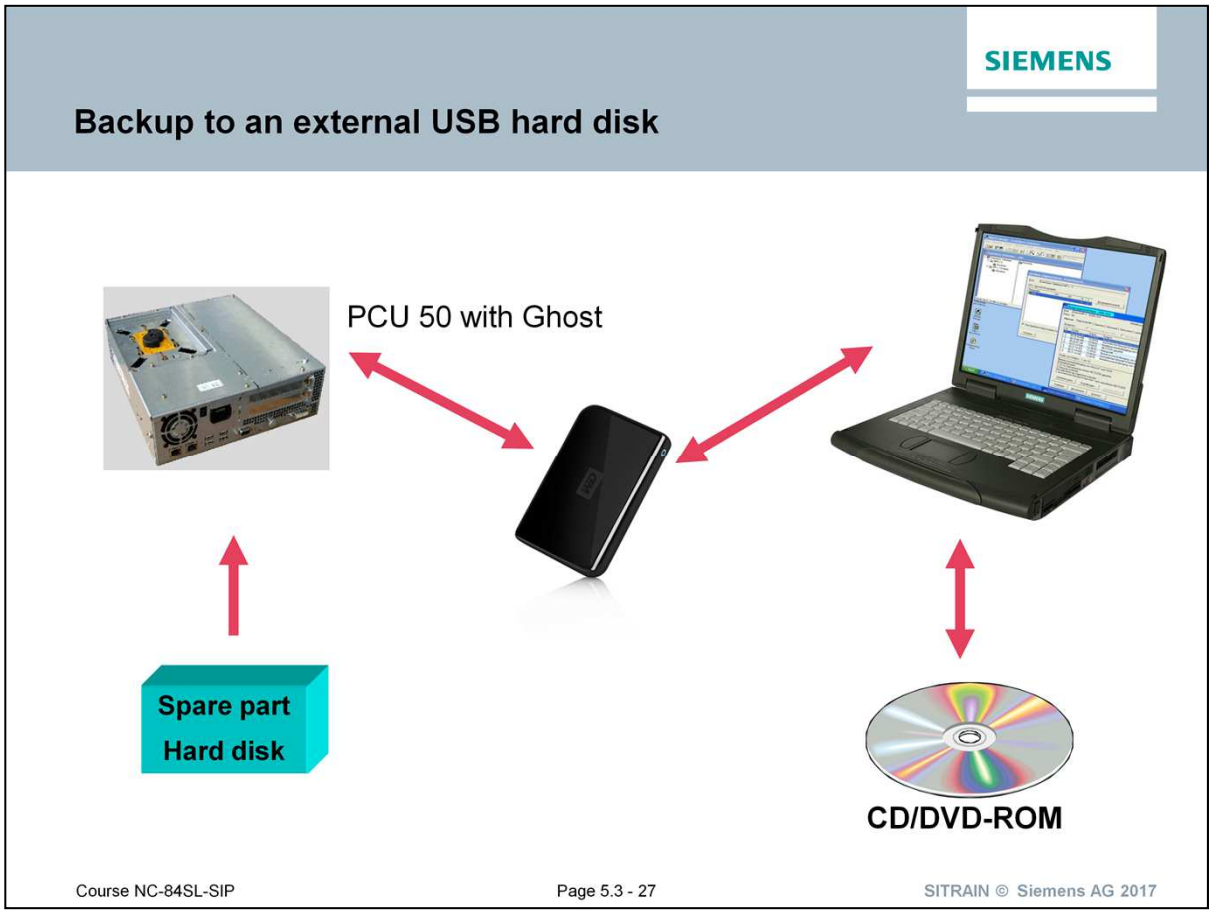


SITRAIN © Siemens AG 2017

Rollback Image If a number of local images have been created, the one created most recently is the rollback image.

Emergency Image The emergency image is a local image that has been defined as the emergency image with the Image Organizer.

Image Organizer The Image Organizer is used to organize local images (to delete an image or define an image as the emergency image, for example).



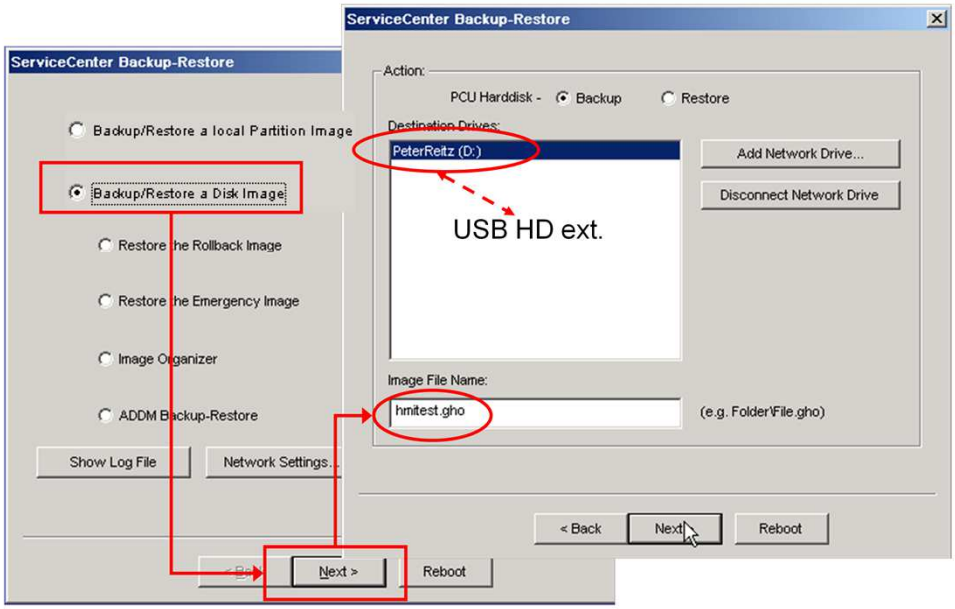
Note

With PCU 50.3 BIOS version 5.1.11 and higher, if the hard disk is locked, the BIOS searches for a connected USB boot medium automatically.

If a USB boot medium is connected, it is used to boot the system.

If the USB medium is an eboot drive of sufficient size (e.g. an external hard disk), a data backup directly to it can be performed.

Backup with ServiceCenter of the PCU to USB drive



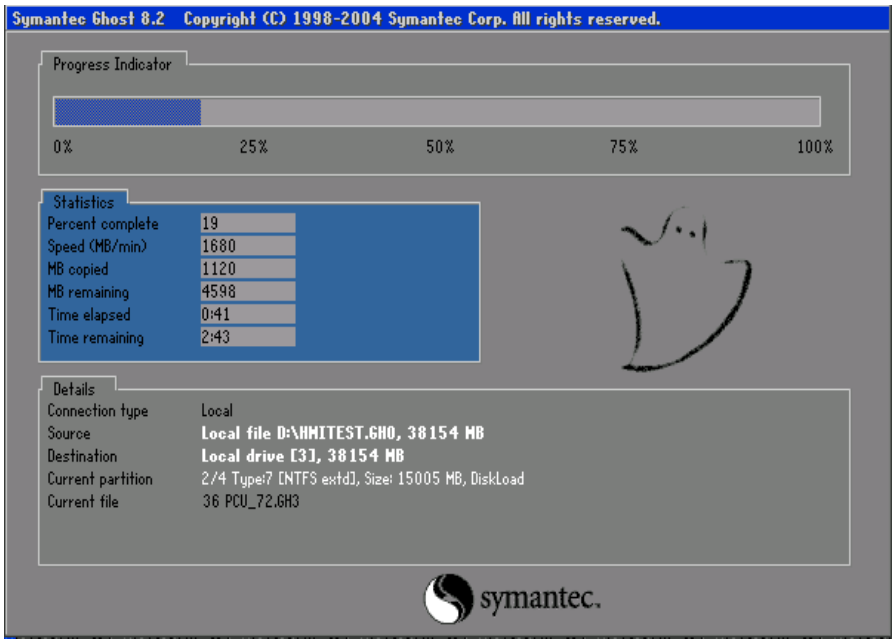
Course NC-84SL-SIP Page 5.3 - 28 SITRAIN © Siemens AG 2017

Booting the PCU **from C: of the PCU hard disk**

Target drive: **External hard disk**

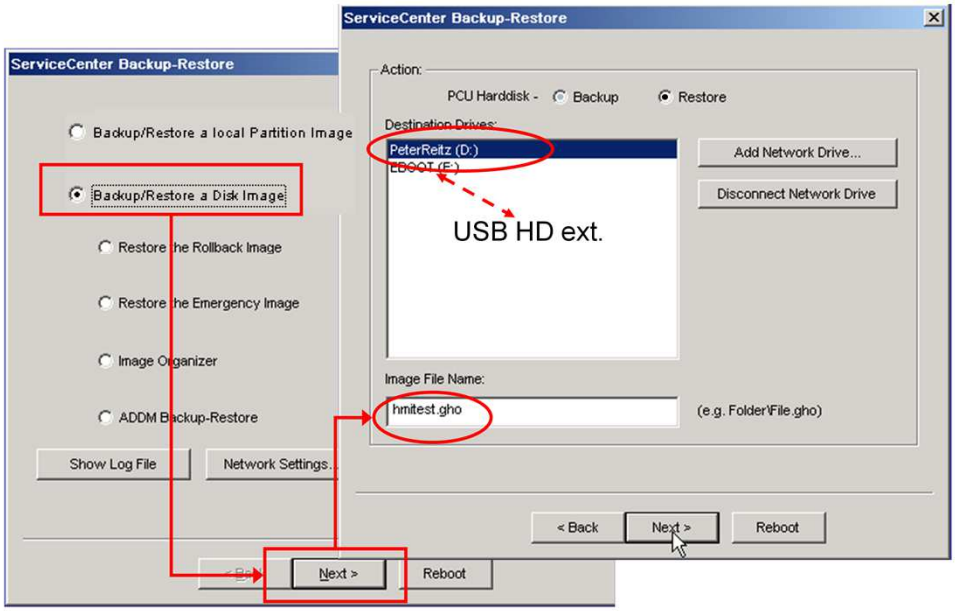
When the ServiceCenter on the PCU starts up, an external USB hard disk should be connected to the PCU. This is then used to create a copy of the hard disk of the PCU, as shown above. The remaining steps follow the standard procedure.

Booting the control The control is started in the Windows XP ServiceCenter in the usual way. Then the USB medium is booted.



Restore with ServiceCenter of the EBOOT flash drive Image from USB hard disk

SIEMENS

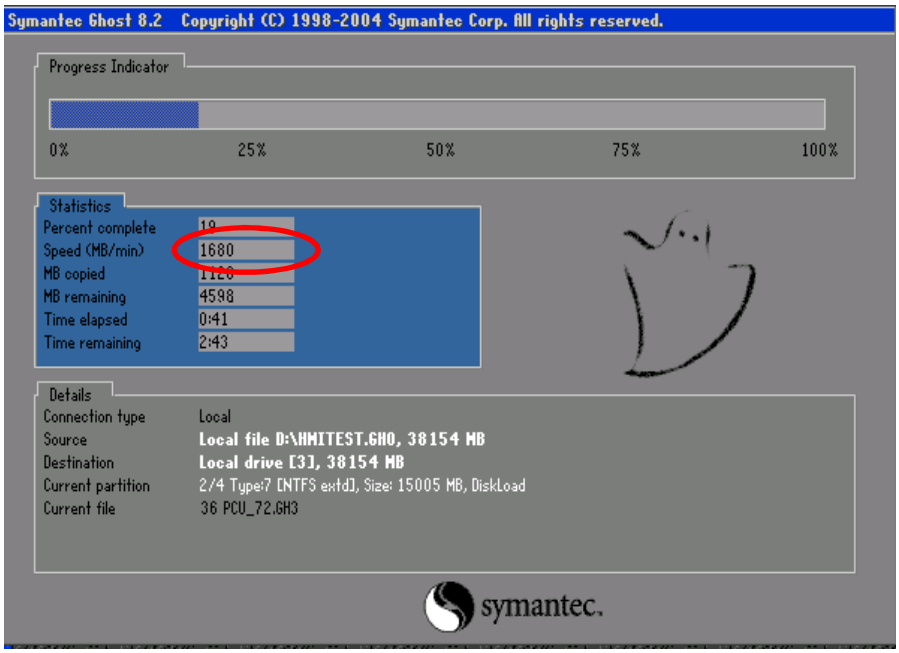


 SITRAIN © Siemens AG 2017

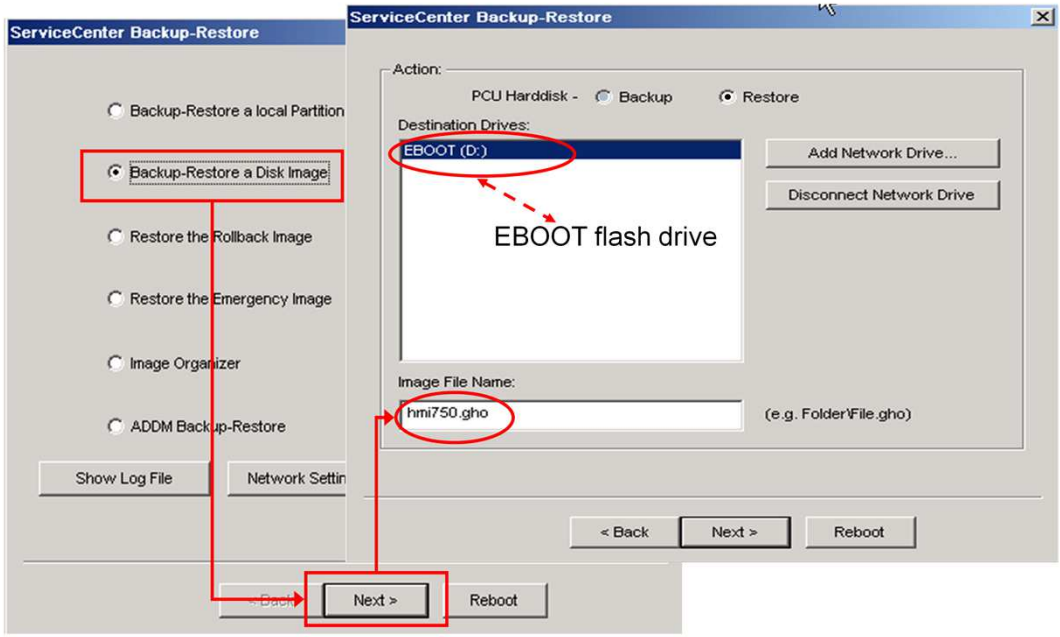
Boot flash drive and USB HDD

When the ServiceCenter is started with the PCU emergency boot flash drive, for example, no other USB drives must be connected. (If they were, they have higher priority than the flash drive and the BIOS would not permit booting from the USB flash drive.)

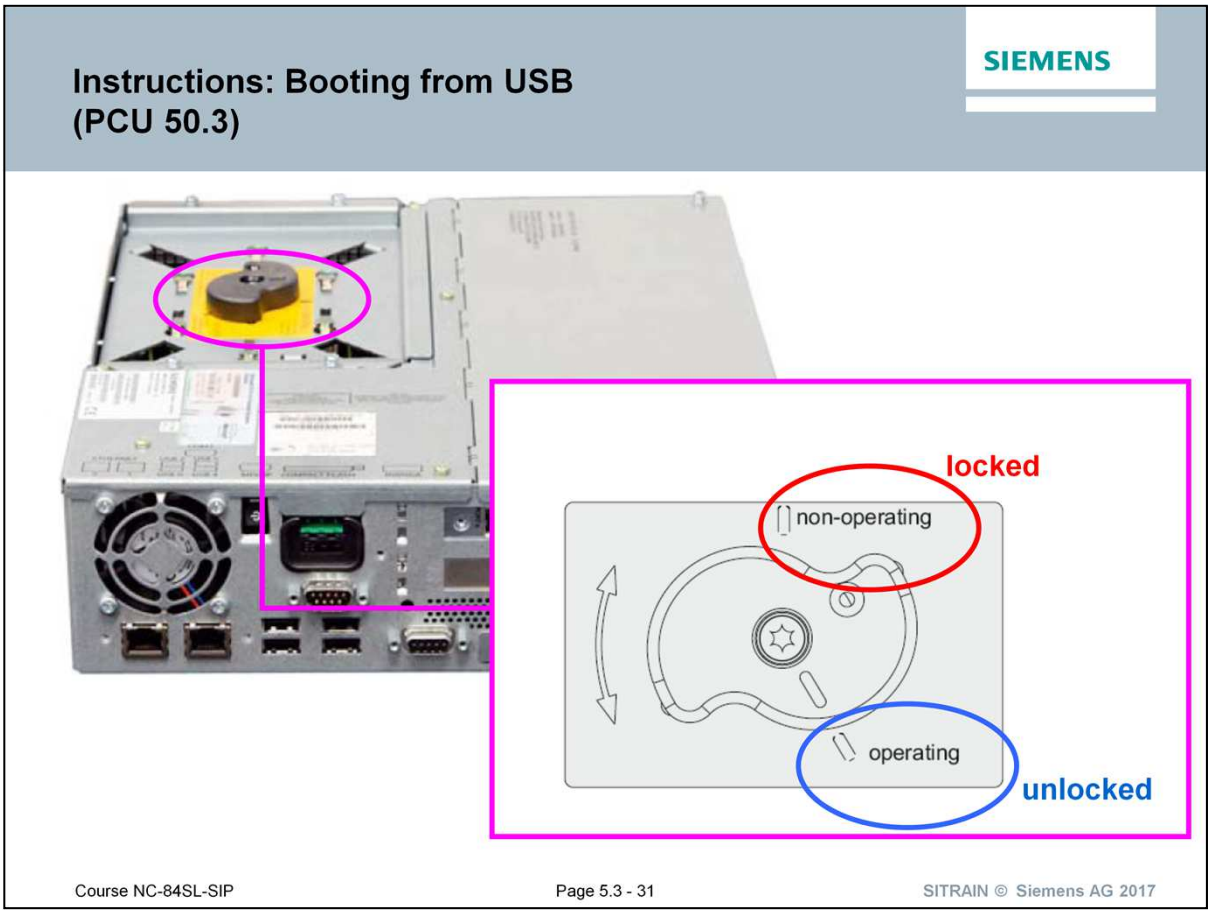
If the service center is booted from the flash drive, the external hard disk is also plugged in. The Ghost image can then be selected from the USB disk and the restore operation can commence.



Restore with ServiceCenter of the EBOOT flash drive image from EBOOT flash drive



Emergency	Insert the EBOOT stick into one of the 4 USB ports on the PCU.
- PCU 50.3	Reboot the PCU with the hard disk locked (as of Bios Version 5.1.11).
- PCU 50.5	Reboot the PCU with the service switch set to **E**.
	The service system will run under Windows PE after booting.
	The system will also operate with TCU and distributed PCU!
ServiceCenter	In the ServiceCenter under Windows PE, select: • **Backup/Restore a Disk Image** Back up/restore whole disk After you have pressed the **-> Next** button, the 'Backup/Restore' dialog box will appear.
Backup/Restore	Select or enter **->Restore** • Image File Name: File name of image file, e.g.: **PCU_761.gho**
Options	When you click the **-> Next** button, an overview of the selected settings is displayed. **-> Next** starts the backup / restore
Notice	Unlock the hard disk when the data backup is complete!



Locking Set the interlocking switch on the PCU hard disk to **'non operating'**.

Unlocking Set the interlocking switch on the PCU hard disk to **'operating'**.

Boot from USB application

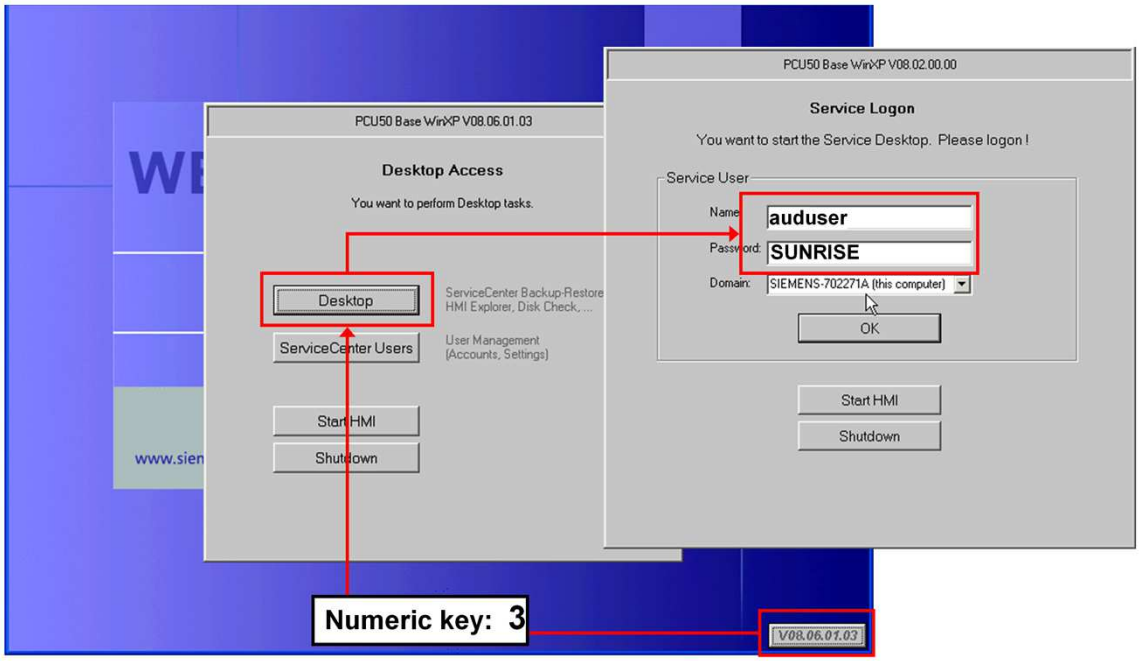
In **BIOS version 5.1.11** and higher, setting the switch to '**non-operating**' forces booting from a USB flash drive (EBOOT flash drive).

(The hard disk always continues to function as only the spring suspension is locked mechanically. The power supply to the disk is not interrupted!)

You must proceed as follows for a Ghost restore, for example:

1. Switch off the PCU, lock the hard disk (switch position: '**non operating**').
2. Connect the USB boot flash drive (to any port).
3. Switch on the PCU and wait for the Service menu to appear.
4. Perform the Ghost restore.
5 Switch off the PCU (do not reboot).
6. Unlock the hard disk (switch position: '**operating**').
7. Remove the USB boot medium.
8. Switch on PCU

Exercise: Start Windows XP on PCU



Course NC-84SL-SIP Page 5.3 - 32 SITRAIN © Siemens AG 2017

Objective

To start PCU with Windows XP only.
This is the only way you access the ServiceCenter for backup/restore.

ServiceCenter

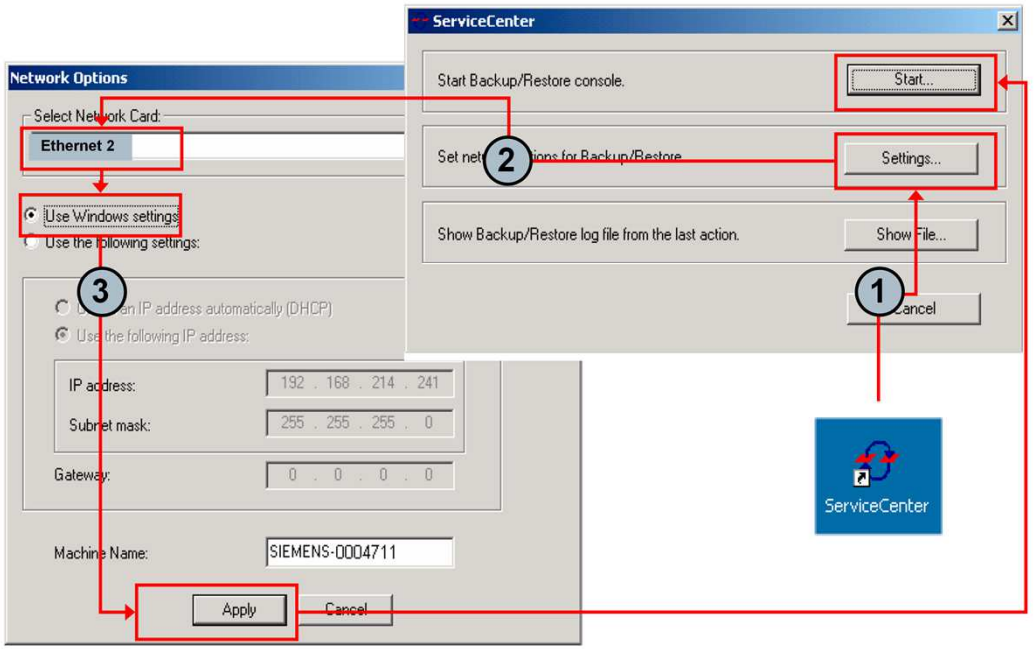
The data backup is performed via the 'ServiceCenter'.

Windows XP

Boot the PCU.
After Windows XP has booted, the version of the PCU Base Software,
e.g. V08.06.01.03 (PCU 50.3)
or V01.02.00.00 (PCU 50.5)
will appear bottom right. At this precise moment, press the number key **-> 3 (on the operator panel).**
Then press the INPUT key to select the desktop
Enter the password:
-> SUNRISE
and press the
-> **OK**
button. No HMI is now running on the PCU and you are on the Windows operator interface.

Exercise: Backup with ServiceCenter of the PCU 50 to network drive

SIEMENS



 SITRAIN © Siemens AG 2017

Objective

To make settings in the ServiceCenter under Windows XP. These settings should match the backup of the hard disk.

ServiceCenter

On the SINUMERIK desktop, you will find the link to the 'ServiceCenter' program. Double-click the link to open the ServiceCenter dialog box.

Settings

Click the
-> Settings
button to open the 'Network Options' dialog box.

Network Options

Select the CPU interface:
• **Ethernet 2**

For the IP address settings select
• **Use Windows settings** and **Apply**

Then you will not need to complete any of the remaining fields.

Start

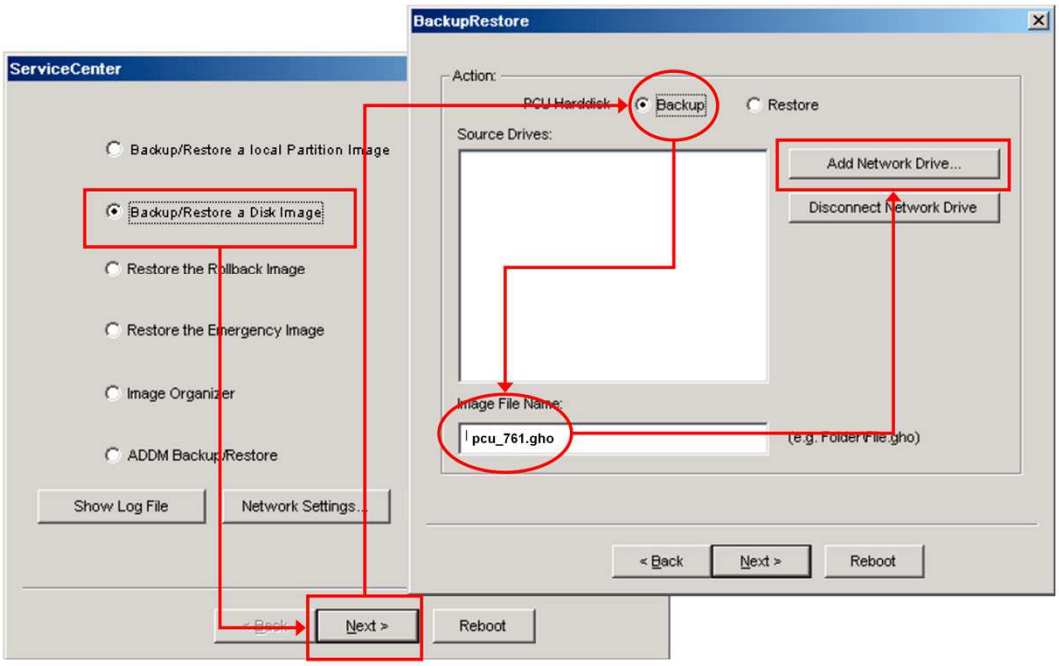
Press the **-> Start** button in the higher-level dialog box
and confirm the subsequent message '... do you wish to continue' with **-> Yes**
After reboot the 'Backup/Restore Console' starts.
A Windows PE 2005 runs on a RAM disk there. First you have to wait until the items of the 'Factory' menu have been executed.

• Installation hardware devices

• Setting display resolution

• Starting networking

Exercise: Backup with ServiceCenter of the PCU 50 to network drive



 SITRAIN © Siemens AG 2017

Objective

To make settings for the network connection.
These settings prepare for subsequent data transfer with Ghost.

ServiceCenter

In the ServiceCenter under Windows PE, select:

- **Backup/Restore a Disk Image** (back up/restore whole disk)

After you have pressed the
-> Next button, the 'Backup/Restore' dialog box will appear.

Backup/Restore

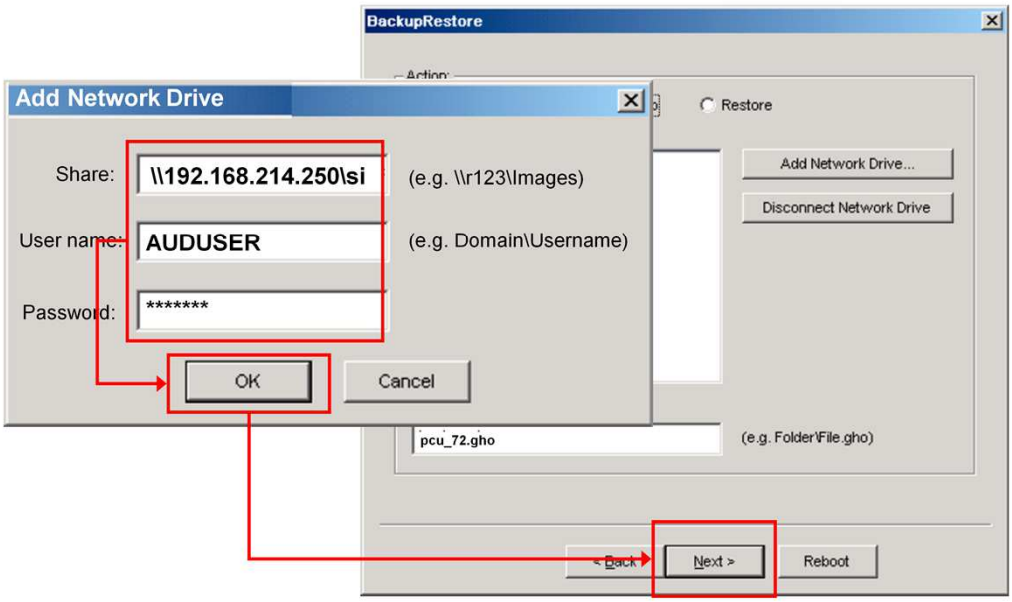
Select or enter

-> Backup

- 'Image File Name' file name of the image file, e.g.: **PCU_761.gho**
  Select the dialog box with this name with the
  -> Add Network Drive
  button.

Exercise: Backup with ServiceCenter of the PCU 50 to network drive

SIEMENS



Course NC-84SL-SIP Page 5.3 - 35 SITRAIN © Siemens AG 2017

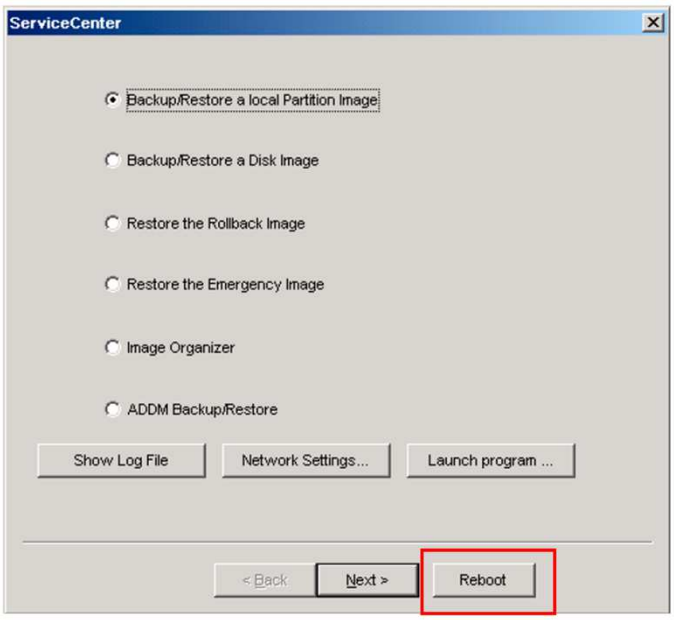
Objective

To connect a network drive.
This prepares for the following data transmission with Ghost.
Add Network Drive In this dialog box, you specify:
• 'Share' external computer (=PG) and image path with the form
\\Computer_name\Share_name or
\\IP_address\Share_name
e.g.: \\192.168.214.250\SINUBACK
• 'User name' , e.g. **AUDUSER**
• 'Password' , e.g.: **SUNRISE**
After **OK,** the connection is established and the network drive can be seen in the backup / restore window.

Options

Press the
-> Next
button. An overview of the selected settings is displayed. Check the overview. Then start backing up with the
-> Finish button.

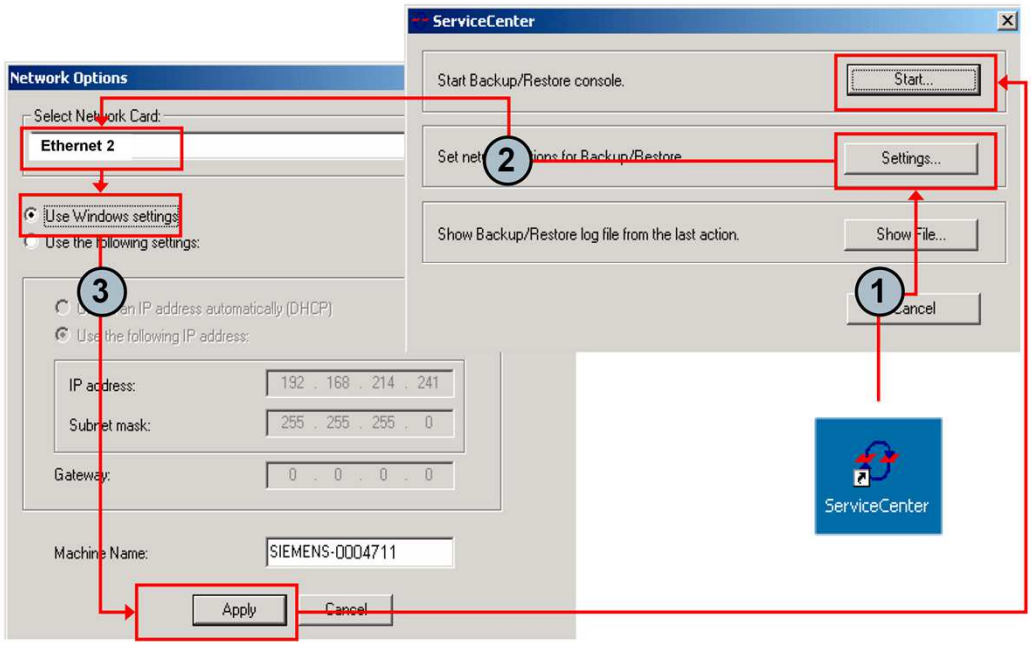
Exercise: PCU backup – Complete and check



Course NC-84SL-SIP Page 5.3 - 36 SITRAIN © Siemens AG 2017

Objective	To exit service mode and check the data.
Exit ServiceCenter	To exit the ServiceCenter, click the following button: **-> Exit**
Check	Check whether the corresponding backup file exists on the hard disk of your PG/PC. The data backup is now complete.

Exercise: Restore with ServiceCenter of the PCU 50 from network drive



Objective

To make settings in the ServiceCenter under Windows XP. They should match the restore of the hard disk.

ServiceCenter Settings

On the SINUMERIK desktop, you will find the link to the 'ServiceCenter' program. After you have double-clicked on it, the dialog box with the same name opens.
Click the
-> Settings
button to open the 'Network Options' dialog box.

Network Options

Select the interface of the PCU:
- **Ethernet 2**

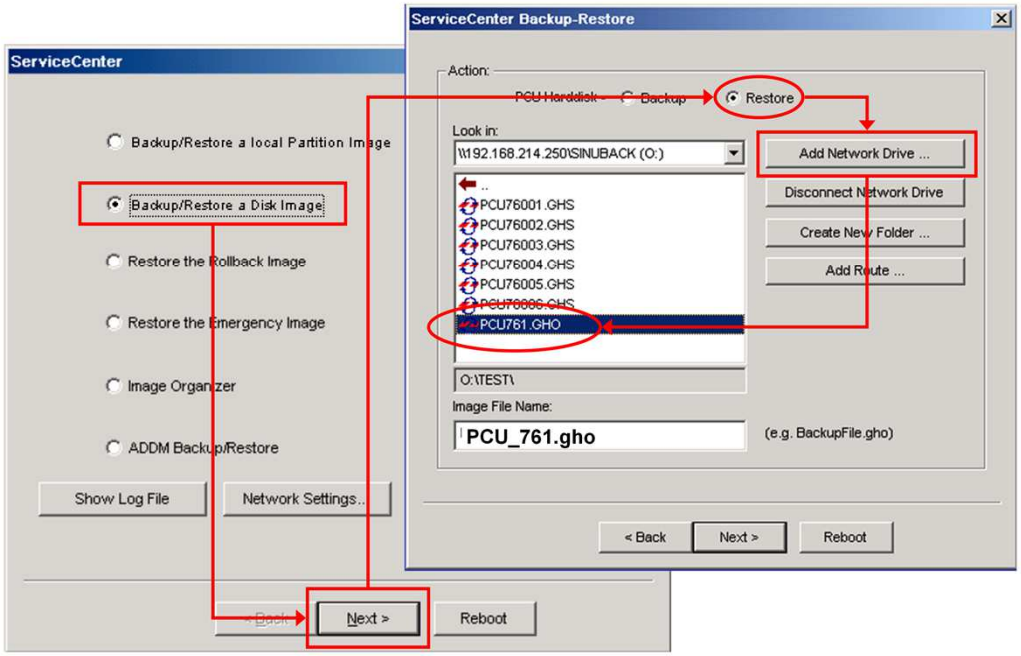
For the IP address settings, select
- **Use Windows settings**

The other input fields are then not required.

Start

Press the **-> Start** button in the higher-level dialog box
and confirm the subsequent message
'... do you wish to continue' with **-> Yes**
After reboot the 'Backup/Restore Console' starts.
A Windows PE 2005 runs on a RAM disk there. First you have to wait until the items of the 'Factory' menu have been executed.
• Installation hardware devices
• Setting display resolution
• Starting networking

Exercise: Restore with ServiceCenter of the PCU 50 from network drive



 SITRAIN © Siemens AG 2017

Objective

To make settings for the network connection.
These settings prepare for subsequent data transfer with Ghost.

ServiceCenter

In the ServiceCenter under Windows PE, select:
• **Backup/Restore a Disk Image** (complete disk backup / restore)

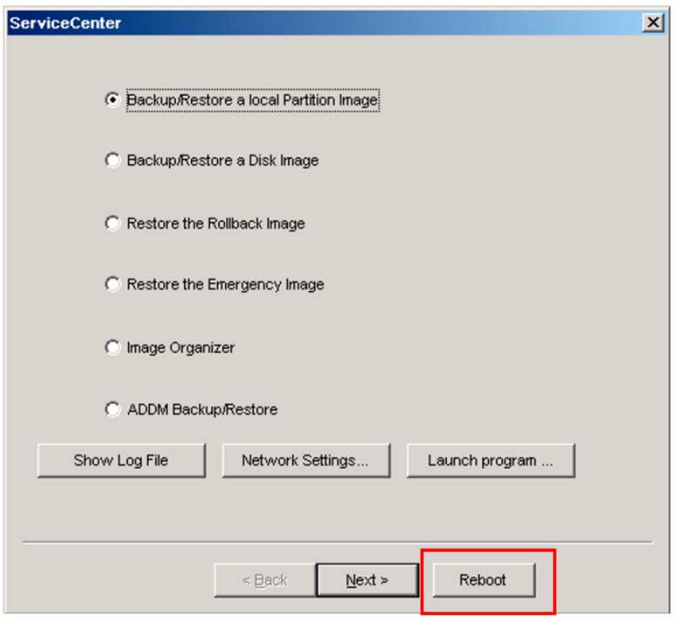
Click the
-> Next button. The 'Backup/Restore' dialog box is displayed.

Backup/Restore

Select or enter
-> Restore
-> Add Network Drive

Once you have selected the network drive (in the same way as in the backup with Ghost exercise),
select the image file, e.g.:
• **PCU_761.gho**
Press the
-> Next
button and you receive an overview of the chosen settings. Check the overview. Then start restoring with the
-> Finish button.

Exercise: Restore PCU – Complete and check

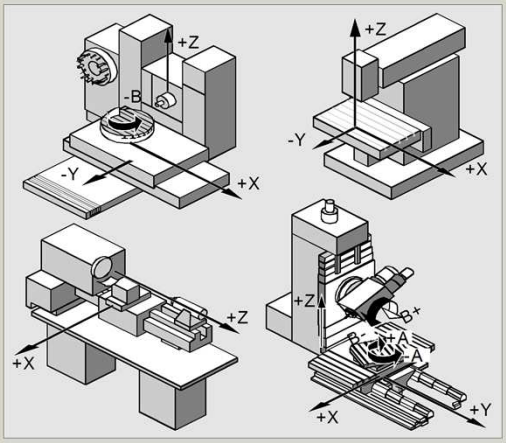


Objective To exit service mode and check the data.

Exit ServiceCenter To exit the ServiceCenter, click the following button:
-> Reboot

Check Check that the PCU now boots normally (as when creating the backup).

SIEMENS



Chapter 6

Commissioning Basics

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Content **Page**

Learning objectives 2
Structure of the NCU 3
Machine and setting data ranges 4
MD – SINAMICS 5
Activating machine data 7
Switching over the MD display to plain text 8
Searching and display options 9
Help function for MD 10
Documentation for machine data 11
Using the Select key to enter MD bits 12
BICO 13
Overview: Inputs/outputs after commissioning 18
Managing data 20
Data comparison 21
Drive objects (DOs) and components 23
Commissioning requirements 24
DRIVE-CLiQ rules 25
Commissioning overview 26
Information: Software on training PG 27
Overview of the training rack 28
Interconnection of ON/OFF 1 and line contactor 29
Exercise: DRIVE-CLiQ wiring on device with ALM 30

Learning objectives

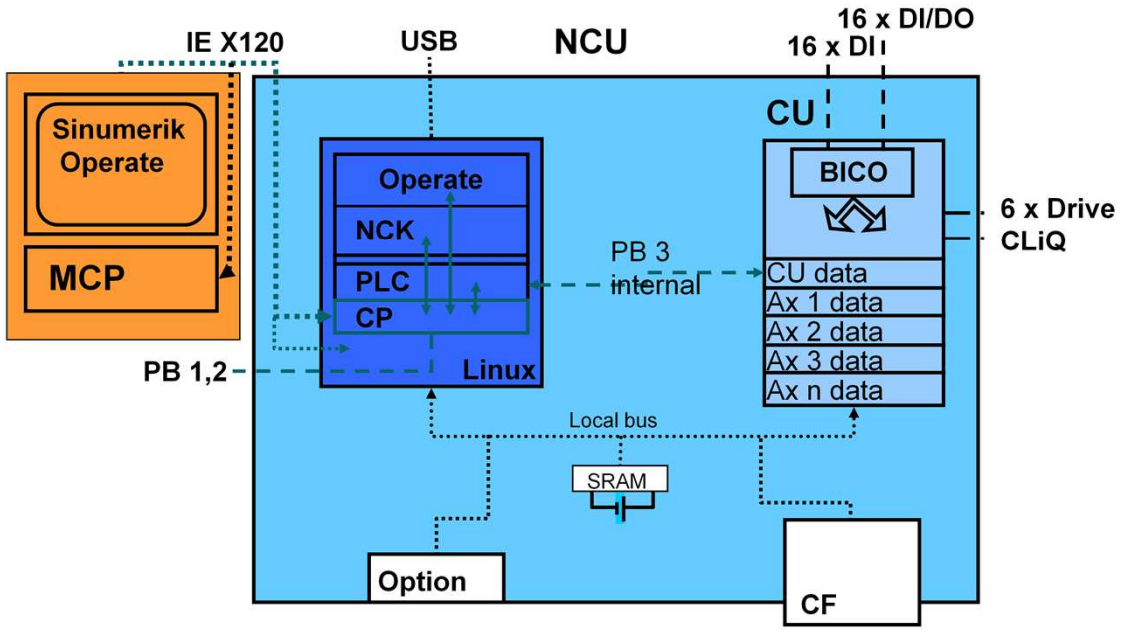
- You will be familiar with using machine data
- You will be familiar with manipulating machine data
- You will be familiar with basic settings
- You will be familiar with the requirements for commissioning
- You will be familiar with the commissioning sequence



SITRAIN © Siemens AG 2017

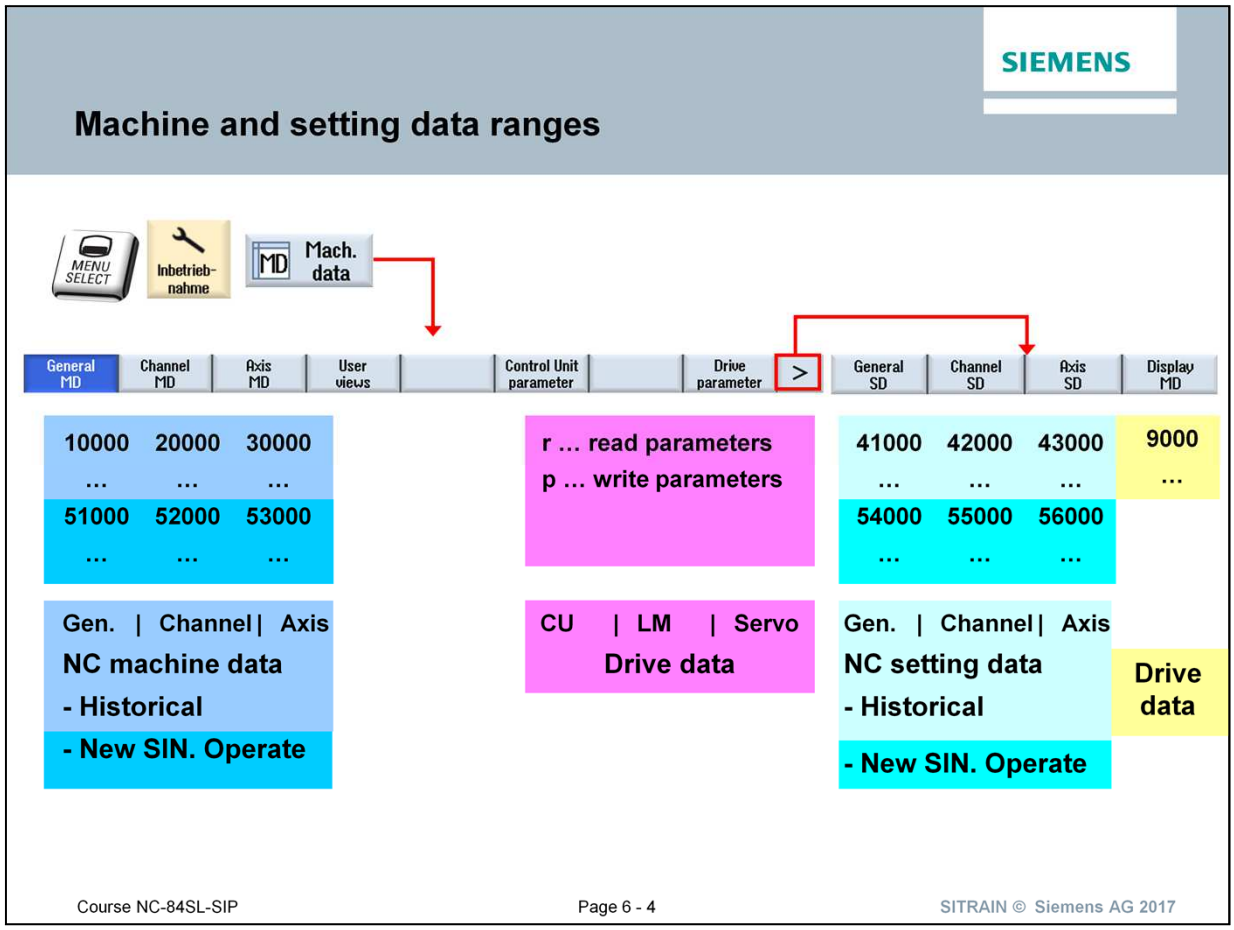
Notes

Structure of the NCU



SITRAIN © Siemens AG 2017

Notes



General machine data

General machine data are those that contain basic settings for the entire NC. Example: Names of the machine axes

Channel-specific machine data

Channel-specific machine data are used for settings relating to program execution in a channel. They include, for example, the number of R parameters in the channel

Axis-specific machine data

These machine data only apply to one axis and depend on the drive and mechanical system used on the machine. They include settings for the main spindle. Example: maximum axis velocity

Drive "machine data"

The SINAMICS S120 is parameterized in special drive parameters. The data are used, for example, to optimize the drives.
The data are divided into the categories Control Unit (CU), Line Module (LM), and Servo. Examples of servo data: Motor and encoder data p300 … and p400 …

Display machine data

The display machine data contain, for example, settings for display, in particular, of alarms and tooltips

Setting data

These setting data are used to store machining-specific information.
Like machine data, they are divided into the categories General, Channel-specific, and Axis-specific data.

MD – SINAMICS
Name

SIEMENS

Read and write capability

p...	Adjustable parameters (read and write)
r...	Display parameters (read only)

Examples

p0918	adjustable parameter 918
p0099[0...3]	adjustable parameter 99 indices 0 to 3
p1001[0...n]	adjustable parameter 1001, indices 0 to n (n = configurable)
r0944	display parameter 944

SITRAIN © Siemens AG 2017

Notes

MD – SINAMICS
Number ranges

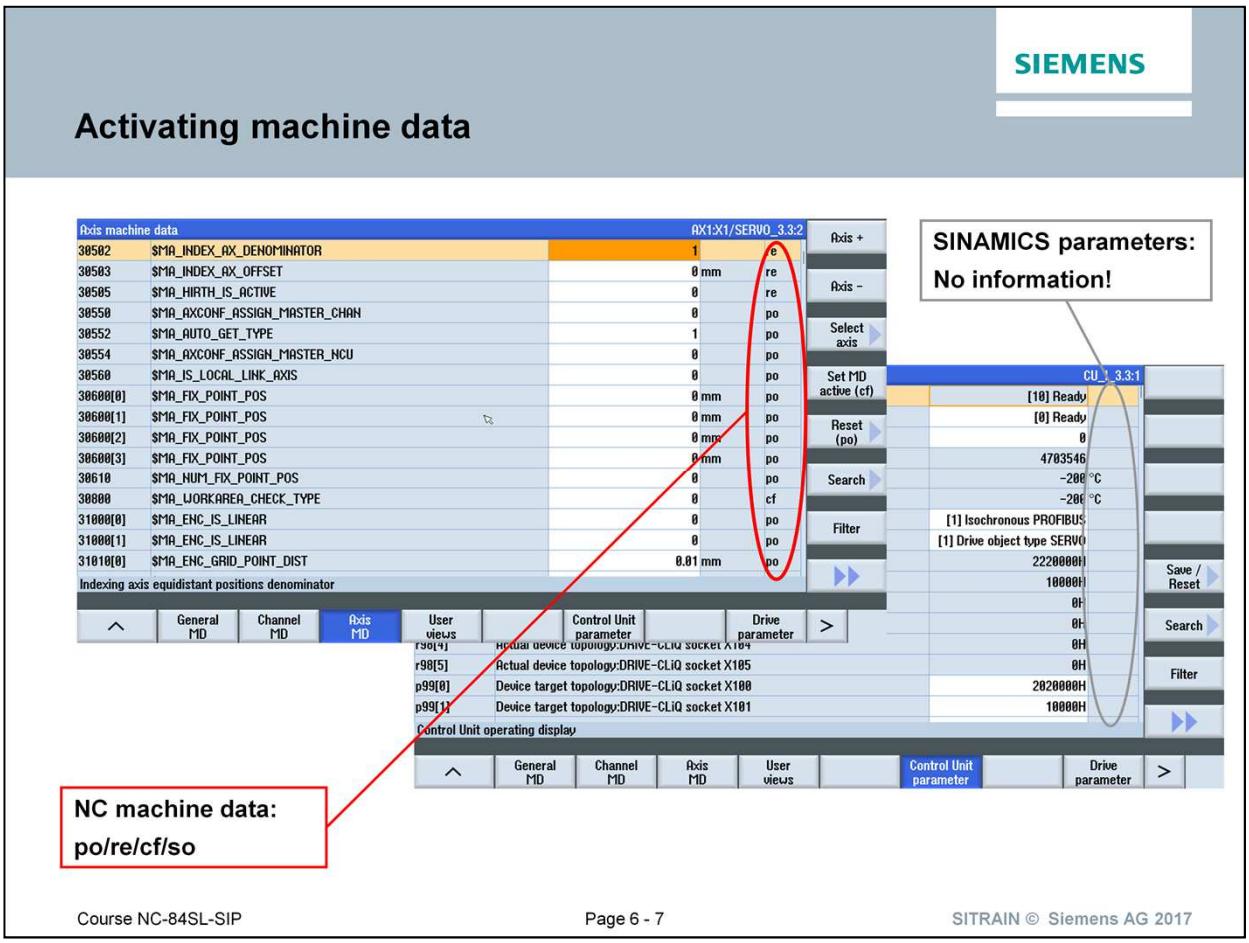
SIEMENS

Range		Description
from	to	
0	99	Operation and display
100	199	Commissioning
200	299	Power unit
300	399	Motor
400	499	Encoder
500	599	Technology and units
600	699	Thermal motor protection and motor model, maximum current
700	799	Command sources and terminals on the Control Unit, measuring sockets
800	839	CDS, DDS data sets (e.g. switch over, copy)
840	879	Sequence control (e.g. source for ON/OFF1)
880	899	Control and status words
900	999	PROFIBUS/PROFIdrive
1000	1199	Setpoint channel
1200	1299	Functions (e.g. motor holding brake)
1300	1399	U/f control
...	...	...

SITRAIN © Siemens AG 2017

Ranges

0	99	Operation and display
100	199	Start-up
200	299	Power unit
300	399	Motor
400	499	Encoder
500	599	Technology and units
600	699	Thermal motor protection and motor model, max. current
700	799	Command sources and terminals of the Control Unit, test sockets
800	839	CDS, DDS data sets (e.g. swichover, copy)
840	879	Sequential control (e.g. source for ON/OFF1)
880	899	Control and status words
900	999	PROFIBUS/PROFIdrive
1000	1199	Setpoint channel
1200	1299	Functions (e.g. motor holding brake)
1300	1399	U/f control
1400	1799	Closed-loop control
1800	1899	Gating unit
1900	1999	Power unit and motor identification
2000	2099	Communication (PROFIBUS)
2100	2199	Faults and alarms, monitoring functions
2200	2399	Technology controller
2900	2930	Fixed values (e.g. percentage, torque)
3400	3699	Closed-loop infeed control (Active Line Module)
3900	3999	Management parameters
4000	4199	Terminal Board, Terminal Module (e.g.. TB30, TM31)
4200	4399	Terminal Module (e.g. TM15, TM17)
6500	6599	External messages
7000	7499	Power units connected in parallel
7800	7899	EEPROM read-write parameters
8600	8799	CAN bus
8800	8899	Communication Board



Activating MDs

In general, machine data is not immediately effective.

NC machine data

The machine data menu shows which machine data must be activated. The following abbreviations are used:

po	POWER ON	Key "RESET" to NCU module or softkey "Reset (po)"
cf	NEW CONFIG.	Softkey "Activate MD" (first press the reset key on the machine control panel)
re	RESET	Reset key on the machine control panel
so	SOFORT	Effective immediately on input (German)
- Engl.: im		- English: immediately

SINAMICS parameters

Unfortunately there is no relevant information available for SINAMICS parameters.

In case of doubt, the following are helpful:

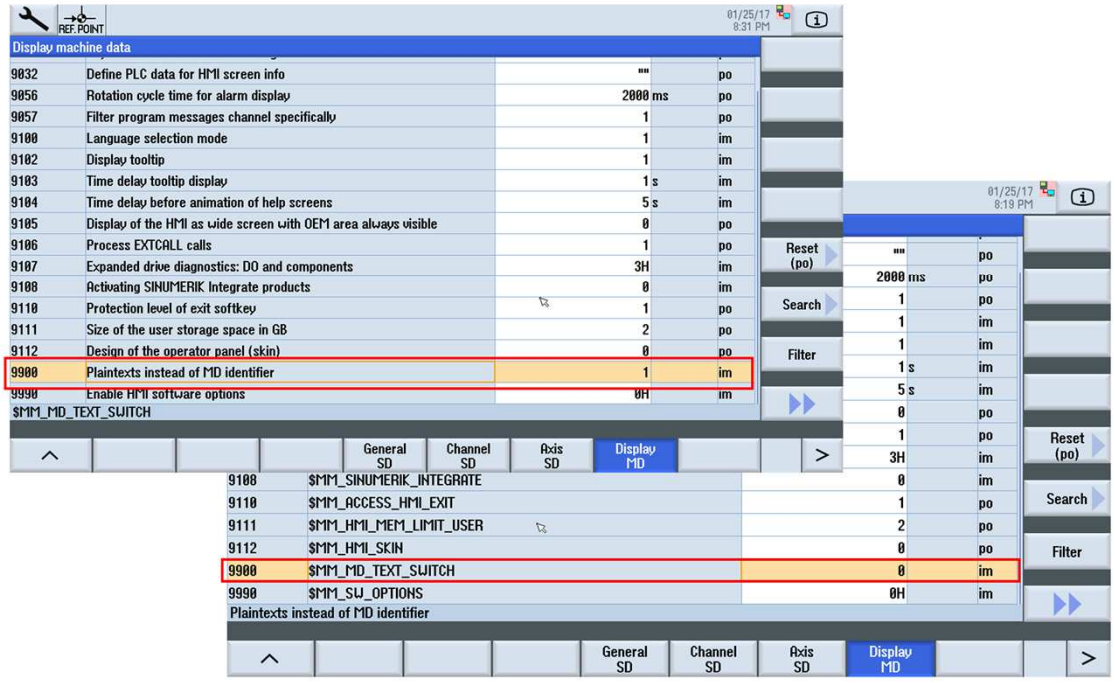
- Save drive system
- Power ON

Access authorization

At least **protection level 4** (key-operated switch position 3) is required to display the machine data.

To enter and change machine data you need at least **protection level 2** (password "EVENING").

Switching over the MD display to plain text



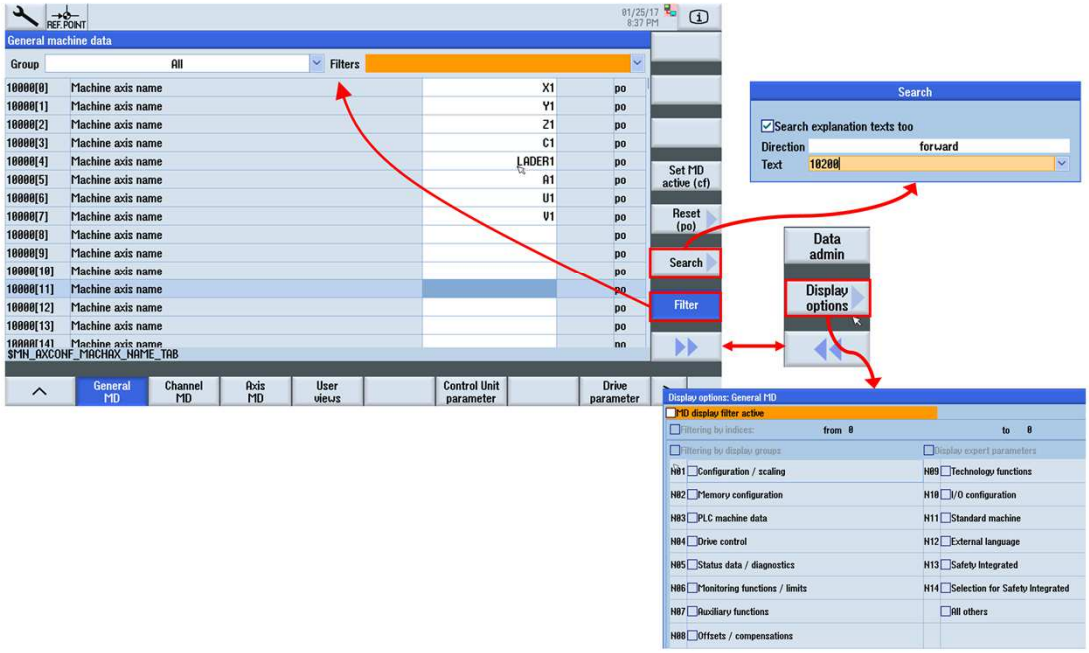
 SITRAIN © Siemens AG 2017

Switching over to plain text

Display MD 9900 is used to switch the machine data display between variable name and plain text.

MD 9900=0	Display variable name
MD 9900=1	Display plain text

Searching and display options



 SITRAIN © Siemens AG 2017

Searching

You can use the "Search" softkey to search for a machine data number or a character string in the machine data. The search is only performed within the selected machine data range.

If there are multiple occurrences of the specified character string, you can use the "Continue search" softkey to expand the search to the next machine data item.

Display options

You can use the filter to selectively reduce the number of machine data items displayed and thus tailor the search to meet the user's requirements. All machine data in the different areas is assigned to specific groups.

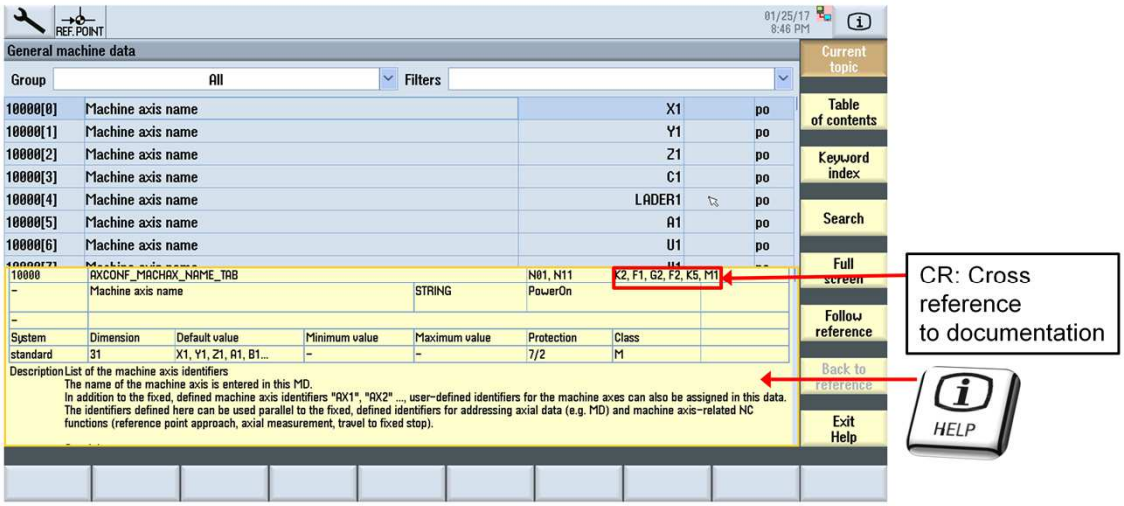
Assignment of machine data to a group is described in the "Lists Book1" documentation.

MD number	Identifier			Display filter	Reference	
Unit	Name			Data type	Activation	
Attributes						
System	Dimension	Default value (LIN/RED)	Minimum value (LIN/RED)	Maximum value (LIN/RED)	Protection	Class

Filtering

With the filtering function, you can search both for functional groups and for strings within the selected group.

Help function for MD



Help function

An online help function is provided for machine data.
Select an item of machine data and press the help key on the OP to call up the help.
The content of the online help corresponds to the machine data description in the "Function Manual".

Cross reference

The cross reference points to a chapter name in the function descriptions. In this example, a reference to Chapter G2.
The function descriptions are to be found on the DOConCD or the documentation DVD of the course.

Documentation for machine data

10000	AXCONF_MACHAX_NAME_TAB			N01, N11	K2, F1, G2, F2, K5, M1	
-	Machine axis name			STRING	PowerOn	
-						
-	31	X1, Y1, Z1, A1, B1, C1, U1, V1	-	-	7/2	M

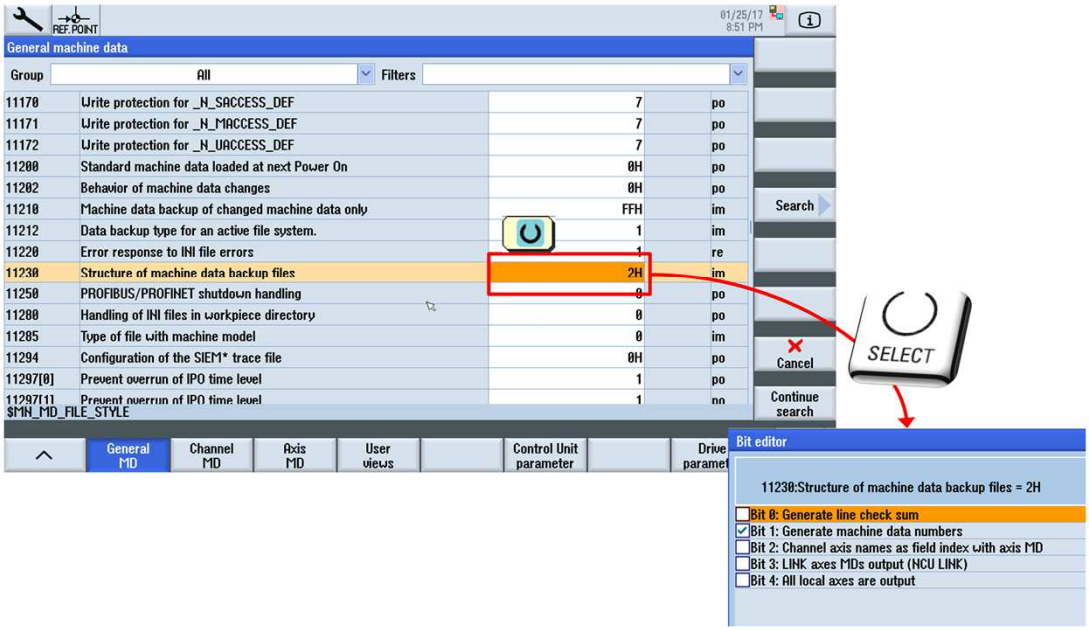
MD number	Identifier			Display filter	Reference	
Unit	Name			Data type	Activation	
Attributes						
System	Dimension	Default value (LIN/RED)	Minimum value (LIN/RED)	Maximum value (LIN/RED)	Protection	Class

Course NC-84SL-SIP Page 6 - 11 SITRAIN © Siemens AG 2017

Documentation

A brief description of all machine data is provided in the "lists" (see above). The "lists" provides a cross-reference to detailed descriptions in the various different "Function Manuals".

Using the Select key to enter MD bits



SITRAIN © Siemens AG 2017

Bit editor for HEX machine data

A bit editor has been implemented to simplify setting certain machine data bits. You can open the editor by positioning the cursor on a machine data item in HEX format in the MD list, then pressing the Select key.

The individual bits can be set or reset after selection by using the cursor keys and pressing the Select key.

BICO

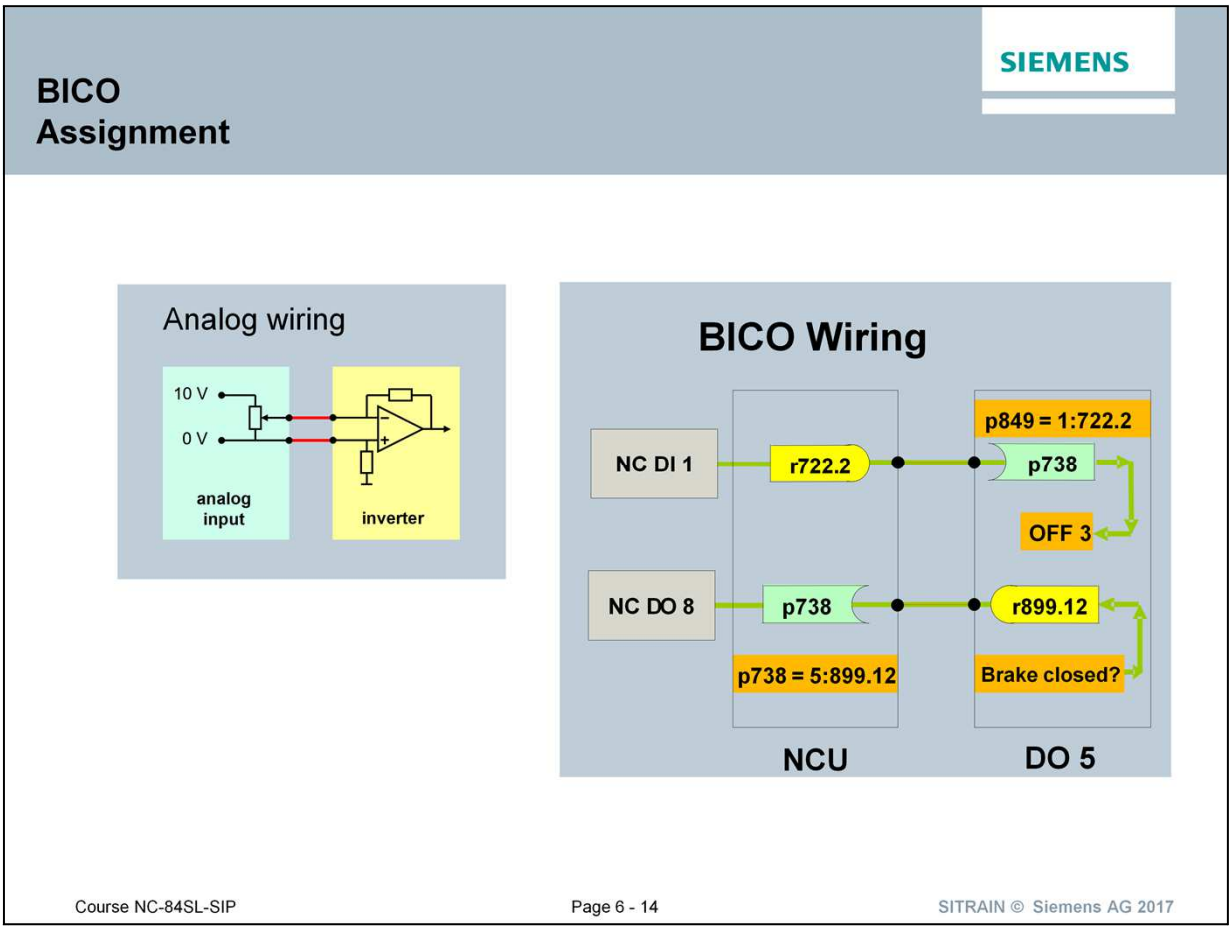
BICO = Binector/Connector
- A binector connects binary inputs/outputs,
- Binector inputs/outputs: BI/BO
  (possible states: 0/1)

Shortcut Symbol	Name	Description
	binector input (signal sink)	Can be interconnector to a binector output as source. The number fo the binector output must be entered as a parameter value.
	binector output (signal source)	Can be used as a source fo a binector input.

Course NC-84SL-SIP Page 6 - 13 SITRAIN © Siemens AG 2017

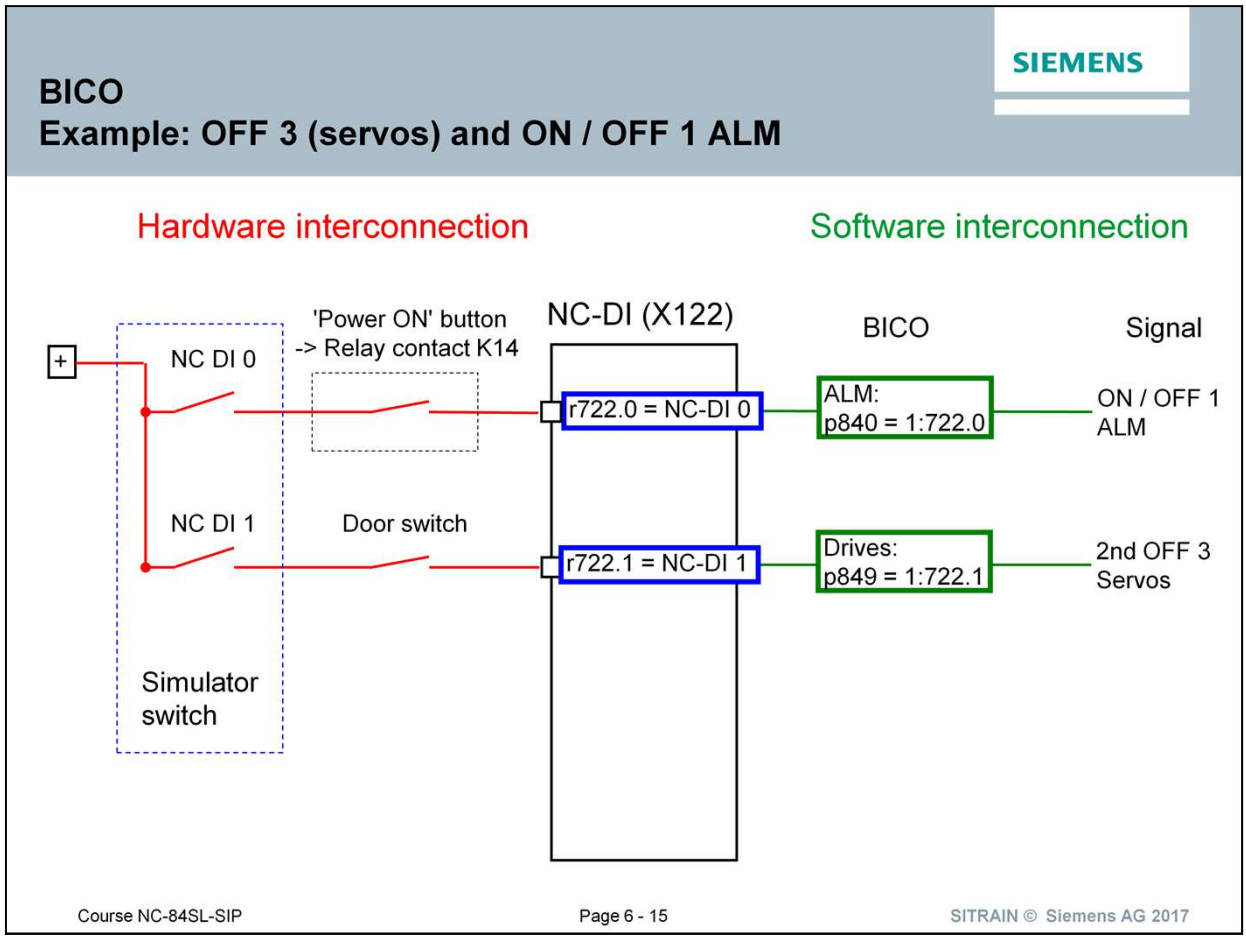
BiCo A connector interconnects a number of bits, e.g. analog signals
Connector ON/OFF = Ci/Co

Shortcut Symbol	Name	Description
	connector input (signal sink)	Can be interconnected to a connector output as source. The number of the connector output must be entered as a paramter value
	connector output (signal source)	Can be uses as a source for a connector input.



Signal sources These are shown with a convex end in the display.

Signal targets (sink) These are shown with a concave end.



Example of BICO technology

Assignment of values to the two basic signals ON/OFF 1 (ALM) and 2nd OFF 3 (servos) will be explained using the example of the training rack.

ON / OFF 1 (ALM)

This signal is required to switch on the infeed.
It should only switch when the "Infeed ready" signal is present at NC DI/DO 13.
The signal source is assigned using BICO technology parameters:

p840 = 1:722.0 = **Object: Parameter. Bit**

The value of parameter p840 indicates the signal source for the 'ON/OFF 1 (ALM)' signal. In an actual application, the signal comes from:
Object 1 (CU), parameter 722 (state of NC DI X122/X132), bit 0

In other words:

ON/OFF 1 is taken from X122.1 on the CU

Requirements for 1-signal (see hardware interconnection above):

- 'NC DI 0' switch ON
- Infeed switched on ('Power ON' button), so that contact K14 is closed.

The line contactor control then switches output NC DO14, and thus line contactor K1

2nd OFF 3

This signal triggers an 'emergency stop' on the drive with state 0' (rapid deceleration followed by pulse suppression).

It is configured on terminal X122.2 for all servos after the device commissioning with:

p849 = 1:722.1

The value of p849 indicates the signal source for the 2nd OFF 3.

2 conditions apply for the 1-signal on account of the hardware wiring

- 'NC DI 1' switch ON
- Protective door (motors) closed

BICO
OFF signals

SIEMENS

Signal	Meaning
ON/OFF1	0 = OFF, drive is stopped via the ramp-function generator ramp (p1121) followed by pulse inhibit 1 = ON, operating condition
OFF2	0 = Coast down, pulse inhibit: Motor coasts down 1 = No coast down, operating condition
OFF3	0 = Quick stop, stop via p1135, followed by pulse inhibit 1 = No quick stop, operating condition

SITRAIN © Siemens AG 2017

OFF signals

OFF1

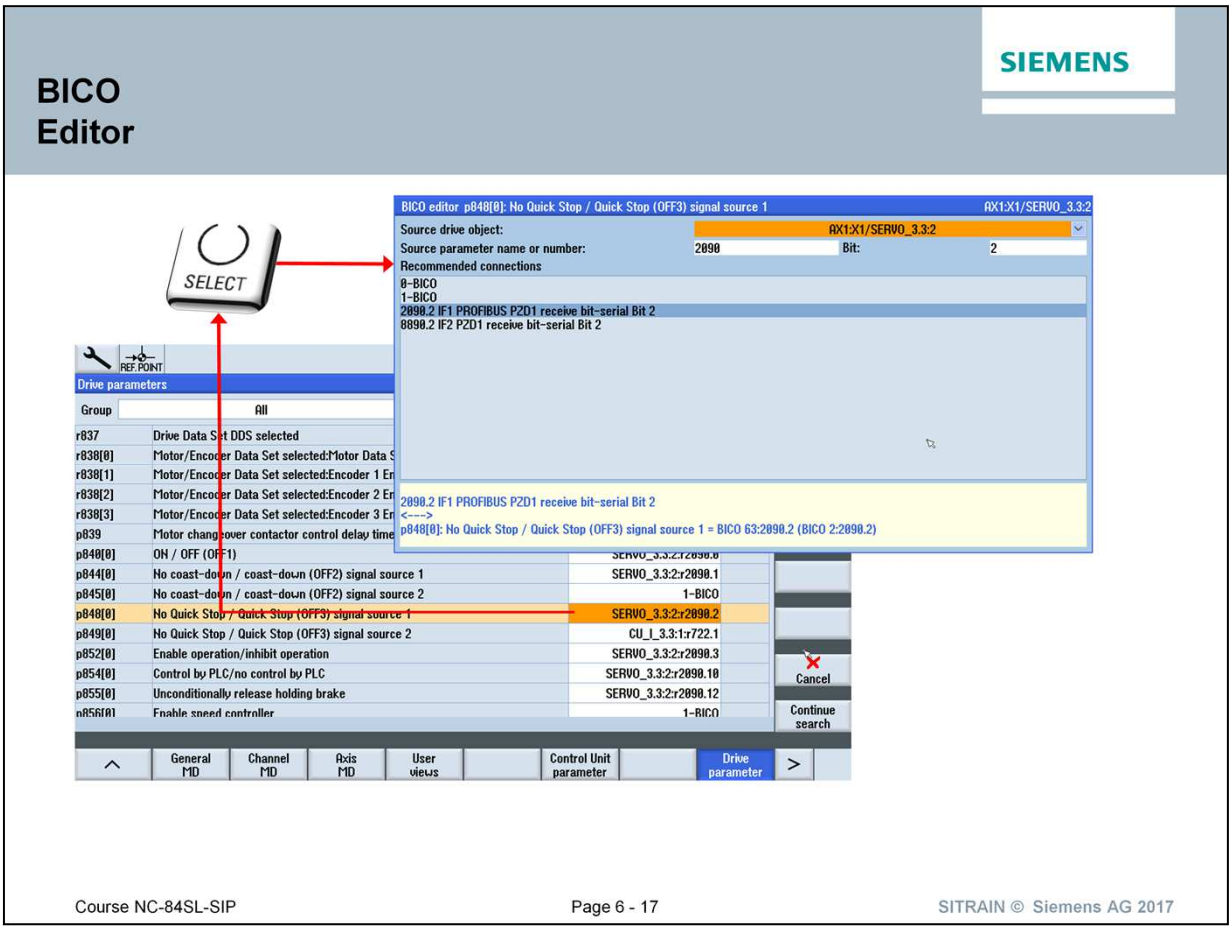
- Immediately setting n_set = 0 decelerates the drive along the ramp-function generator ramp down (p1121).
- When zero speed is detected, the motor holding brake (if parameterized) is applied (p1215). The pulses are suppressed when the brake application time (p1217) expires. Zero speed is detected if the actual speed drops below the threshold (p1226) or if the monitoring time (p1227) started when the speed setpoint <= speed threshold (p1226) has expired.

OFF2

- Immediate pulse suppression, the drive "coasts" to a standstill.
- The motor holding brake (if parameterized) is closed immediately.
- Switching-on inhibited is activated.

OFF3

- Immediately setting n_set = 0 decelerates the drive along the OFF3 ramp down (p1135).
- When zero speed is detected, the motor holding brake (if parameterized) is applied. The pulses are suppressed when the application time of the holding brake (p1217) expires. Standstill is detected when the speed actual value falls below the speed threshold (p1226) or when the monitoring time (p1227) started when speed setpoint <= speed threshold (p1226) has expired.
- Switching-on inhibited is activated.



BICO editor

In the SINUMERIK Operate, you can call the BICO editor by pressing the SELECT key.

Structure of BICO

A complete specification of a BICO consists of:

- **Object** Drive object No.; 63 specifically points to its own object
- **Parameter No.** No. **nnn** of parameter r **nnn** or p **nnn**
- **Bit/Index** must always be assigned values.
  (=0, if the relevant parameter does not yet have an index!)

Overview: Inputs/outputs after commissioning

Result of commissioning: The NC inputs and outputs are preassigned with functions according to the following lists:

NCU 7x0.2, X122:

Pin-No.	Function	Signal	BiCo: Source	Parameter	Remark
1	Input DI0	ON / OFF 1: LM with Drive CLiQ	CU: r0722.0	LM: p840	
		Infeed ready: LM without DQ	SLM X21.1	DRV: p864	
2	Input DI1	OFF 3	CU: r0722.1	DRV: p849	
3	Input DI2	Deselect SH Drive Group 1	CU: r0722.2	DRV: p9620	(only reserved)
4	Input DI3	Deselect SH Drive Group 2	CU: r0722.3	DRV: p9620	(only reserved)
5	Ground for Pins 1 - 4				
6	Ground for Pins 7,8,10,11				
7	In/Output DI/DO8	SH active Drive Group 1	DRV: r9774 Bit 1	CU: p0738	(only reserved)
8	In/Output DI/DO9	SH active Drive Group 2	DRV: r9774 Bit 1	CU: p0739	(only reserved)
9	Ground fors Pin 7,8,10,11				
10	In/Output DI/DO10	Bero 1	CU: r 0722.10	p495 = 2	p728.10=0
11	In/Output DI/DO11	Measuring probe 1 (central)	CU: r722.11	CU: p0680[0] = 3	p728.11=0
		Measuring probe 1 (distributed)	CU: r722.11	DRV: p488	(alternative)
12	Ground for Pins 7,8,10,11				

NCU 7x0.2, X132:

Pin-No.	Function	Signal	BiCo: Source	Parameter	Remark
1	Input DI4	Digital Input $A_IN[1]	CU: r0722.4	CU: p2082[0]	
2	Input DI5	Digital Input $A_IN[2]	CU: r0722.5	CU: p2082[1]	
3	Input DI6	Digital Input $A_IN[3]	CU: r0722.6	CU: p2082[2]	
4	Input DI7	Digital Input $A_IN[4]	CU: r0722.7	CU: p2082[3]	
		Infeed Line contactor feedback (NS*)	CU: r0722.7	LM: p860	(alternative)
5	Ground for Pins 1 - 4				
6	Ground for Pins 7,8,10,11				
7	In/Output DI/DO12	Infeed operation (LM with DQ)	LM :r0863.0	CU: p0742	p728.12=1
		Digital Output $A_OUT[4]	CU : r2091.3		p728.12=1
8	In/Output DI/DO13	Infeed Ready to start (LM with DQ)	LM: r0899.0	CU: p0743	p728.13=1
		Digital Output $A_OUT[3]	CU : r2091.2		p728.13=1
9	Masse für Pin 7,8,10,11				
10	In/Output DI/DO14	Digital Output $A_OUT[2]	CU: r2091.1	CU: p744	p728.14=1
		Control Line contactor (NS*)	LM : r0863.1	CU: p744	p728.14=1
		Bero 2 – External zero mark	CU: r 0722.14	DRV: p0495 = 5	p728.14=0
11	In/Output DI/DO15	Digital Output $A_OUT[1]	CU: r2091.0	CU: p0745	p728.15=1
		Input probe 2 - central measurement	CU: p0680[1] = 6	DRV: p489	p728.15=0
12	Masse für Pin 7,8,10,11				

NS*: If line contactor control is active

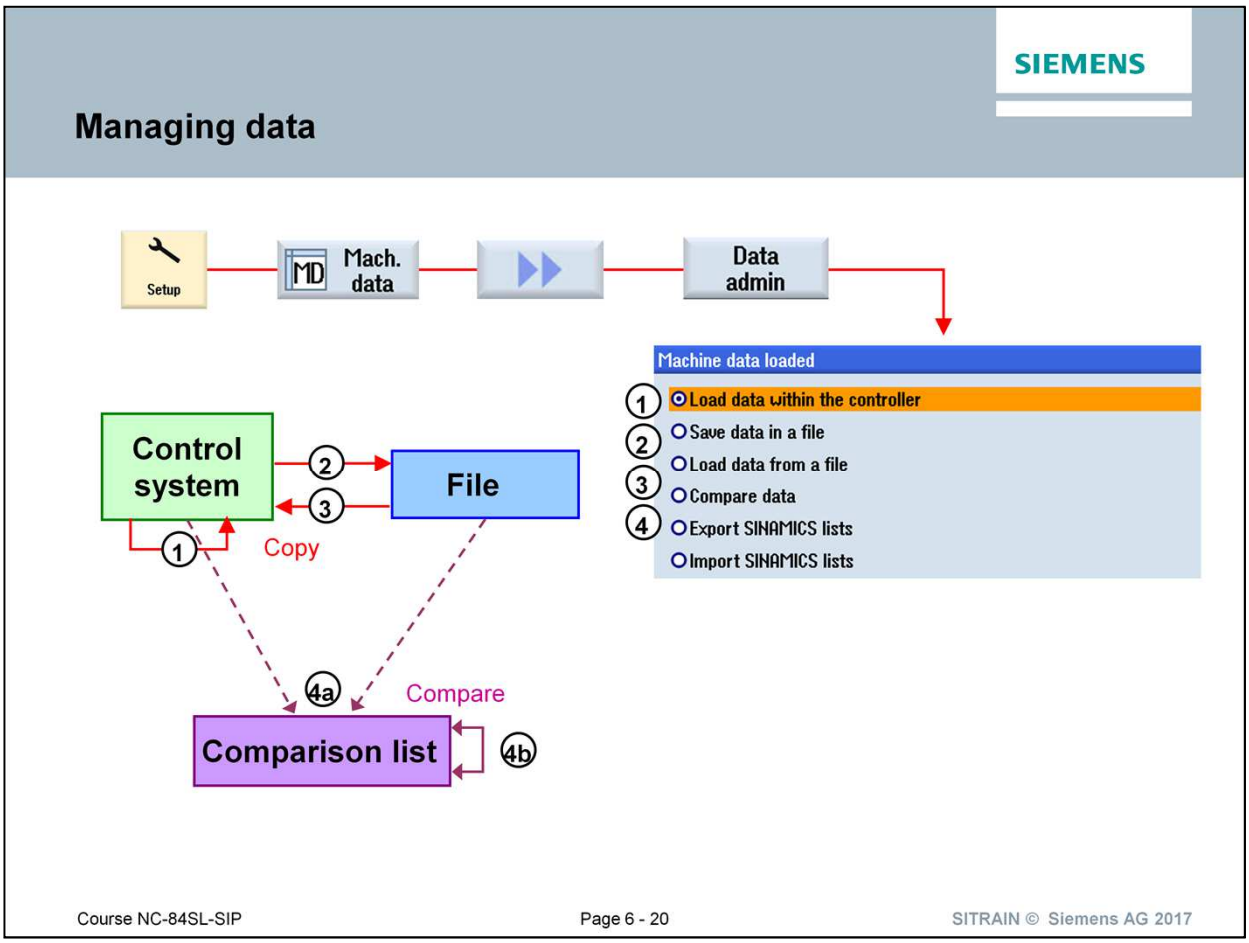
NCU 7x0.3, X122:

Pin-No.	Function	Signal	BiCo: Source	Parameter	Remark
1	Input DI0	ON / OFF 1: LM with Drive CLiQ	CU: r722.0	LM: p840	
		Infeed ready: LM without DQ	SLM X21.1	DRV: p864	
2	Input DI1	OFF 3	CU: r722.1	DRV: p849	
3	Input DI2	Deselect SH Drive Group 1	CU: r722.2	DRV: p9620	(only reserved)
4	Input DI3	Deselect SH Drive Group 2	CU: r722.3	DRV: p9620	(only reserved)
5	Input DI16	Free	CU: r722.16		
6	Input DI17	Free	CU:r722.17		
7	Ground for Pins 1 - 6				
8	Ground for Pins 9,10,12,13				
9	In/Output DI/DO8	SH active Drive Group 1	DRV: r9774 Bit 1	CU: p0738	(only reserved)
10	In/Output DI/DO9	SH active Drive Group 2	DRV: r9774 Bit 1	CU: p0739	(only reserved)
11	Ground for Pins 9,10,12,13				
12	In/Output DI/DO10	Bero 1	CU: r 0722.10	p495 = 2	p728.10=0
13	In/Output DI/DO11	Measuring probe 1 (central)	CU: r722.11	CU: p0680[0] = 3	p728.11=0
		Measuring probe 1 (distributed)	CU: r722.11	DRV: p488	(alternative)
14	Ground for Pins 9,10,12,13				

NCU 7x0.3, X132:

Pin-No.	Function	Signal	BiCo: Source	Parameter	Remark
1	Input DI4	free	CU: r722.4		
2	Input DI5	free	CU: r722.5		
3	Input DI6	free	CU: r722.6		
4	Input DI7	Infeed Line contactor feedback (NS*)	CU: r722.7	LM: p860	
5	Input DI20	free	CU: r722.20		
6	Input DI21	free	CU: r722.21		
7	Ground for Pins 1 - 6				
8	Ground for Pins 9,10,12,13				
9	In/Output DI/DO12	Infeed operation (LM with DQ)	LM: r863.0	CU: p742	p728.12=1
10	In/Output DI/DO13	Infeed Ready to start (LM with DQ)	LM: r899.0	CU: p743	p728.13=1
11	Ground for Pins 9,10,12,13				
12	In/Output DI/DO14	Control Line contactor (NS*)	LM: r863.1	CU: p844	p728.14=1
13	In/Output DI/DO15	Input probe 2 - central measurement	CU: r722.15	DRV: p489	p728.15=0
14	Ground for Pins 9,10,12,13				

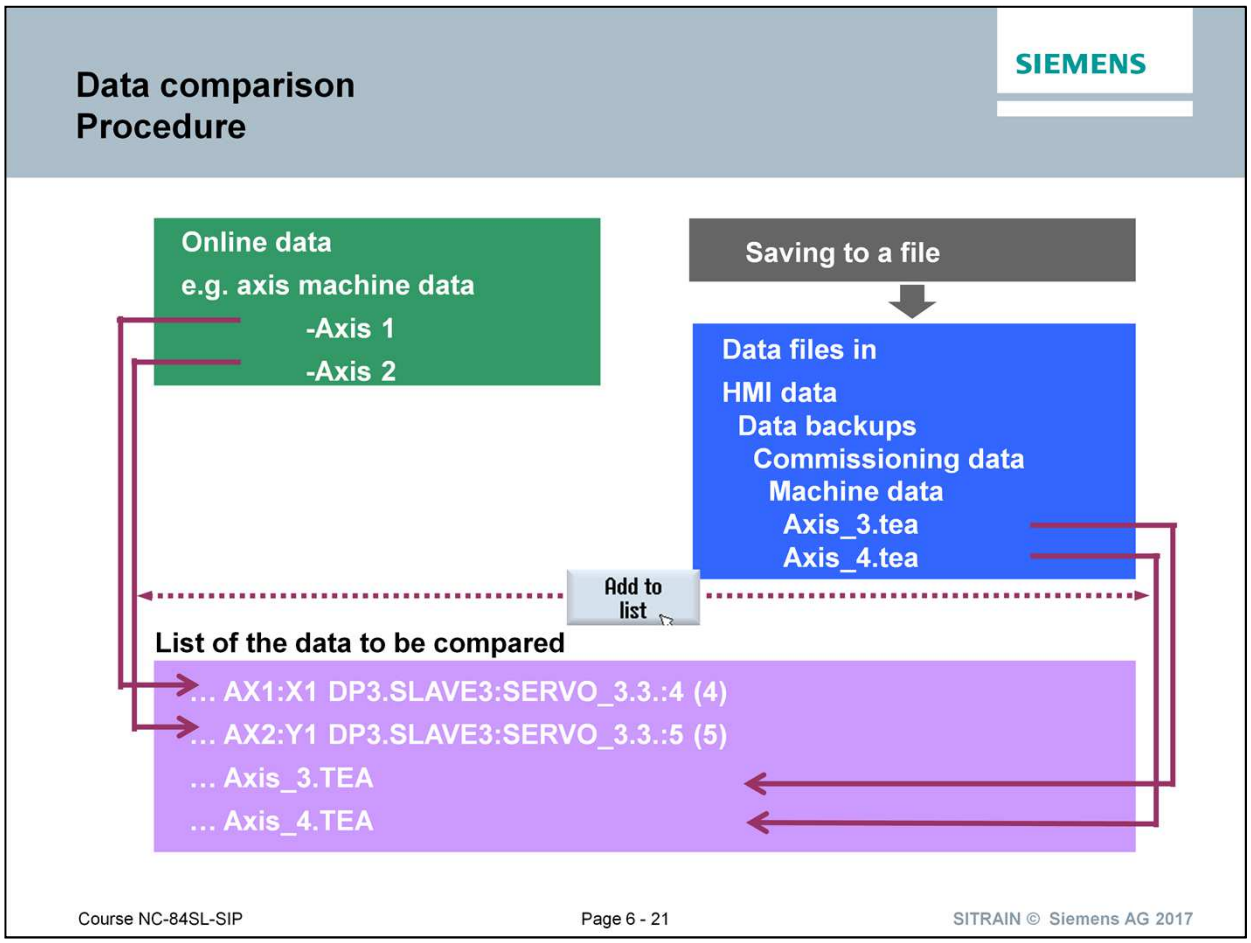
NS*: If line contactor control is active



Managing data

Using the softkey: "Manage data", the following functions are possible:

- Transfer data within the control
  e.g. the drive machine data of one drive to another drive
- Save data to a file
  e.g. axis machine data into an editable file for service purposes
- Load data from a file
  e.g. to transfer a data set of one channel to another channel.
  This only makes sense in rare special cases and is no substitute for a commissioning archive.
- Compare data
  This function is especially useful for service purposes.

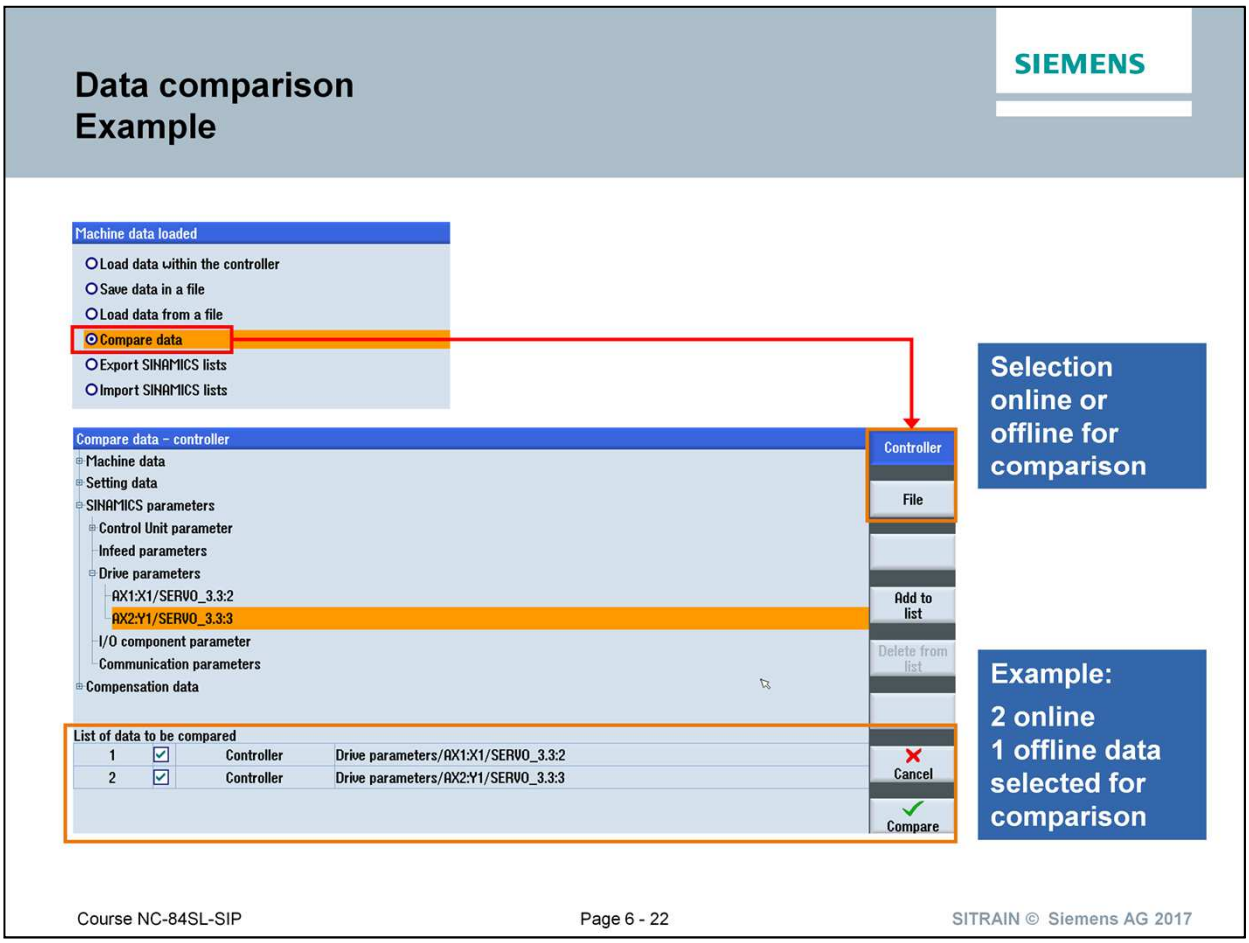


Compare data

When you select "Compare data", it is possible to compare machine data.

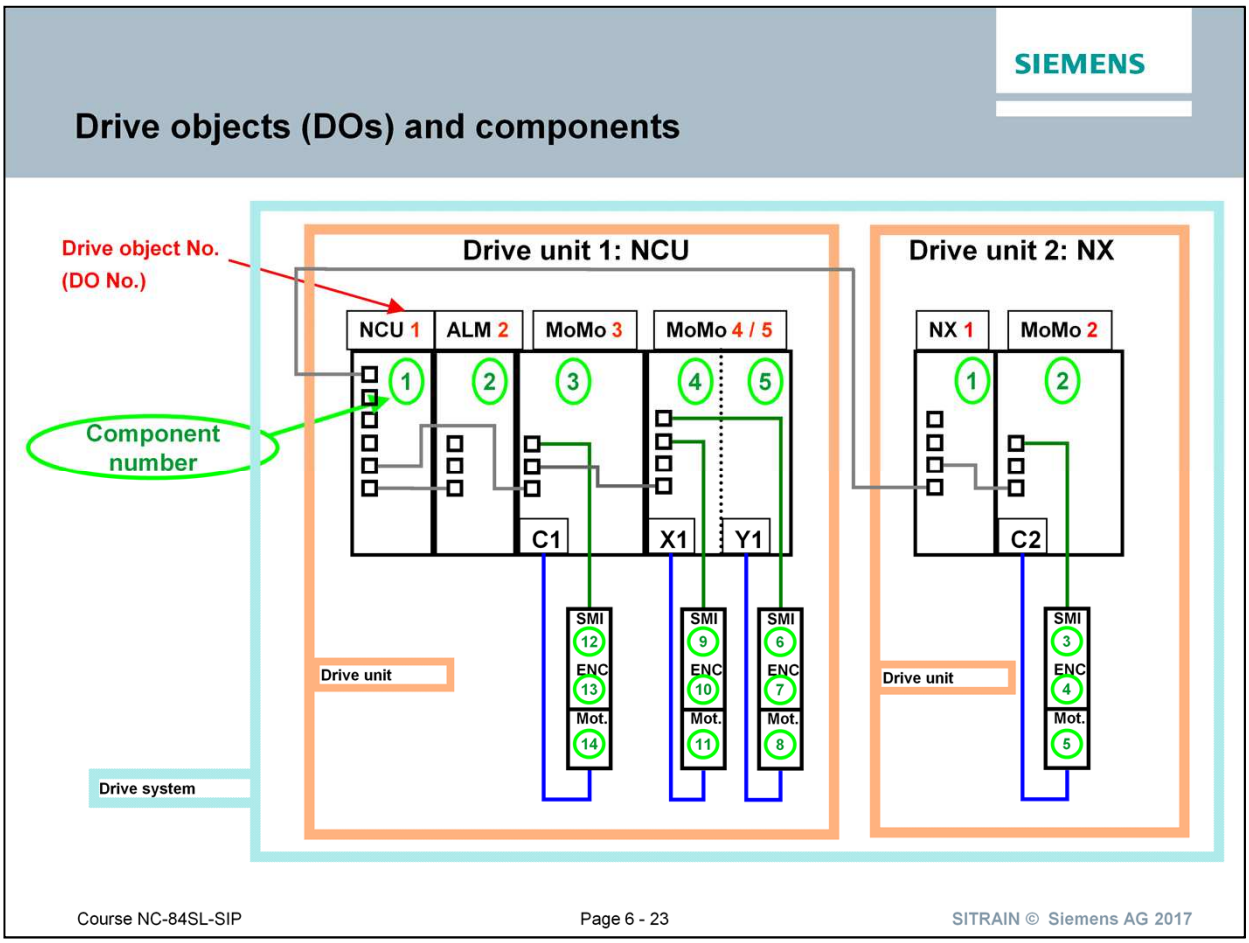
In the first step, a selection must be made of data of the control (online) or data from a file (offline) with the **softkey: "Add to list"**

The second step is to select lines from the selection and compare them using **softkey: "Compare"**



Example "compare data"

The picture above shows an example of data comparison.
2 x online data (axis data X, Y) and 1 x offline data (axis data X) are selected.



Drive objects (DO)

Each drive object has its own parameter assignment.

It consists of drives comprising the **components:**

- **Power unit** component number: p121
- **Motor** component number: p131
- **Encoder interface** component number: p141
- **Encoder** component number: p142

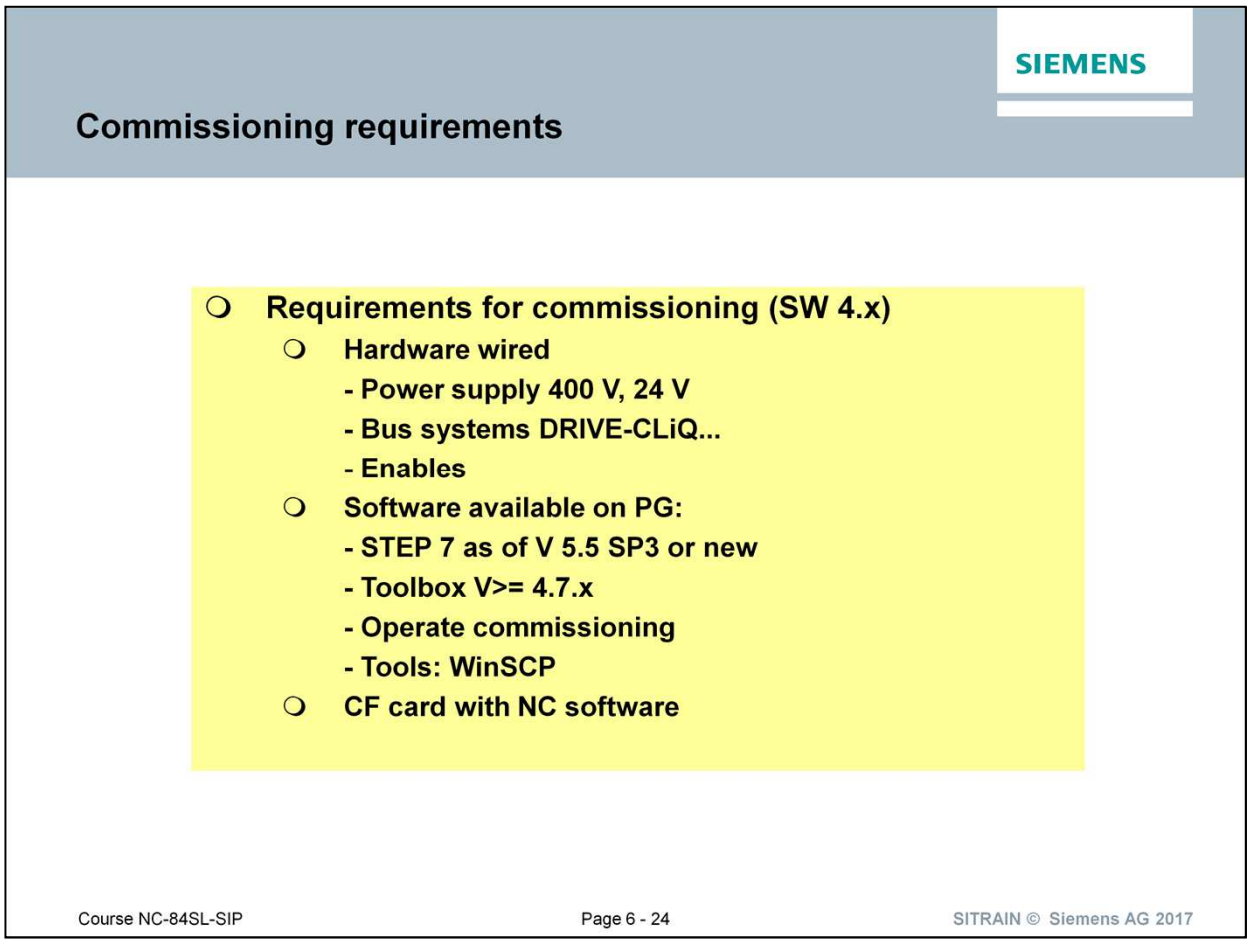
Automatic assignment of component number

With automatic topology recognition, the process of assigning component numbers automatically works like this:

1. DRIVE-CLiQ node on the main line segment	= MoMos / NCUs / ALMs
2. Other DRIVE-CLiQ nodes	= SMI / SME / SMC
	= Encoder interfaces
3. Components on SMI / SME / SMC	= encoder
4. Other components	= motors

For the MoMos detected under 1., the last 3 points are executed in reverse sequence (starting with the last module).

Therefore, the last drive (DO) is assigned the lowest component numbers for Sensor Module, encoder and motor, the one before last is assigned the next numbers, etc.



Commissioning requirements

The essential commissioning requirements are as follows:

Hardware wiring

NCU and SINAMICS wired completely

- Power supply 400 V / 24 V
- DRIVE-CLiQ, Ethernet, PROFIBUS
- Enables, e.g.

  Terminal EP(ALM) <-- 24 V
  NCU X122.1 (ON/OFF1 LM) <--- 24V

(See also the chapter of the machine configuration manual dealing with signal interconnection.)

Software on PG

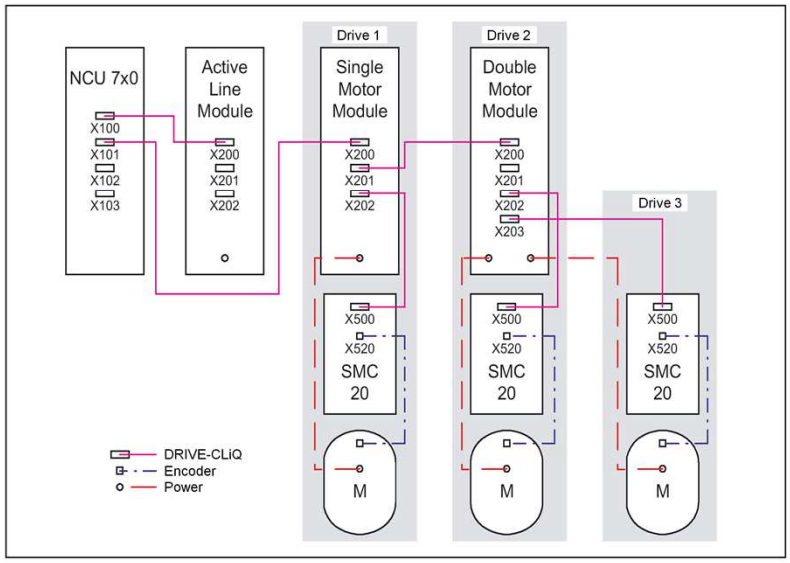
- STEP 7 (see upgrade instructions for the NC software)
- Toolbox "
- Commissioning tool
- Tools: WinSCP, Access MyMachine P2P

CF card with NC software

Alternatively: Empty CF card (consider version standard/export!) and tgz file for the NC software.

DRIVE-CLiQ rules

- **Max. 14 nodes per port**
- **s in series**
- **ler to associated MoMo**
- **ng**

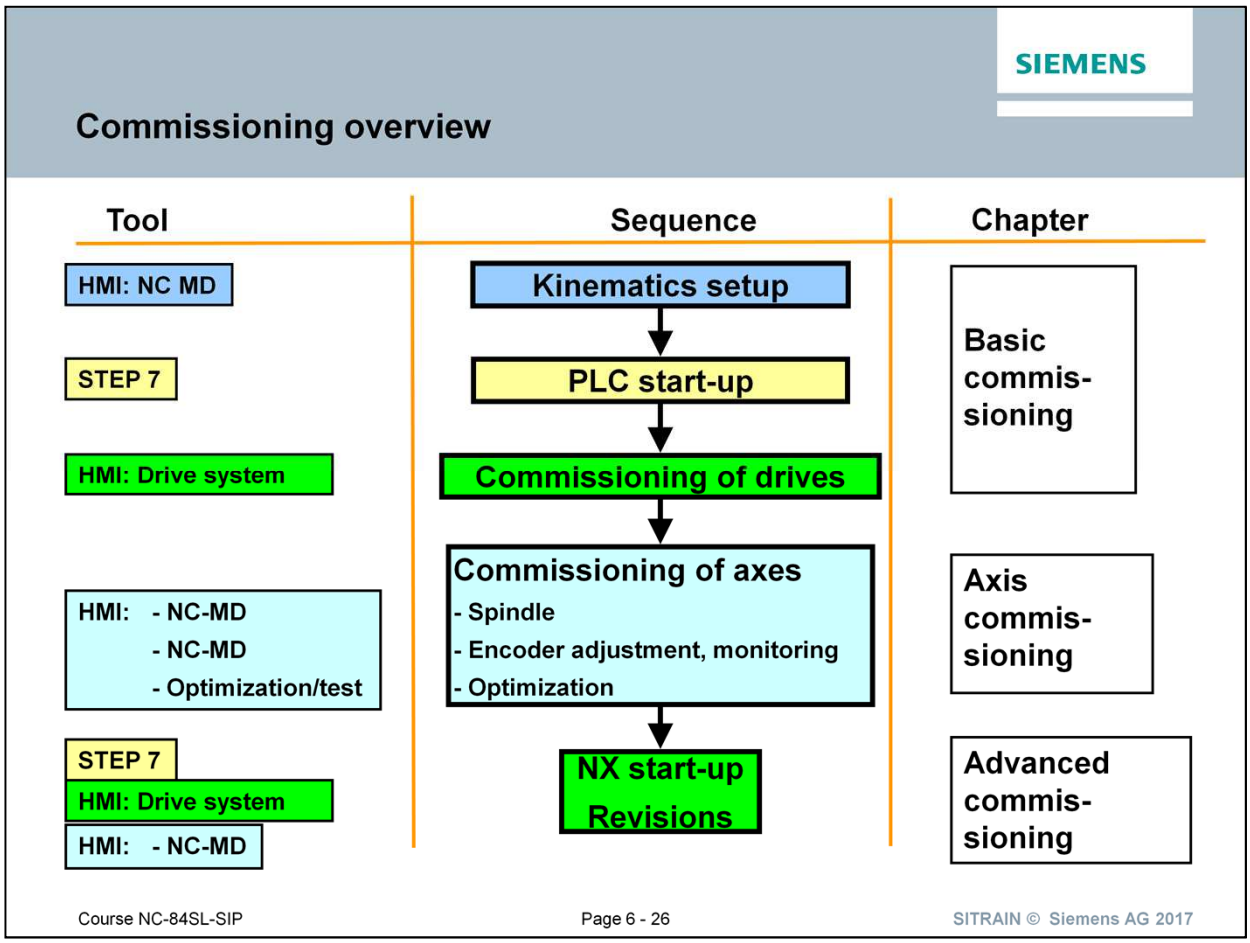


Course NC-84SL-SIP Page 6 - 25 SITRAIN © Siemens AG 2017

DRIVE-CLiQ rules

Adherence to these rules enables automatic topology detection.

- For one Motor Module, the related motor encoder must also be connected.



Commissioning workflow

Kinematics Here, you first have to clarify which axes exist on the machine and which axis groupings are to be moved with dedicated NC programs.
On the basis of this information, the axes are assigned to corresponding NC channels (auxiliary axes can also be traversed separately by the PLC, e.g. axes of a tool changer).

PLC The PLC start-up procedure comprises the following steps:

- HW Config
- PLC program for supplying MCP, enables, etc.

Drives Drive commissioning comprises the following steps:
- Device configuration (detection of the DRIVE-CLiQ wiring = topology)
- Start-up power supply
- Manual commissioning of not completely detected drives; second encoder, etc.
- Assignment of axes

Axes The axis commissioning requires definition of:
- Axis dynamics (velocities, etc.)
- Encoder adjustment
- Monitoring functions

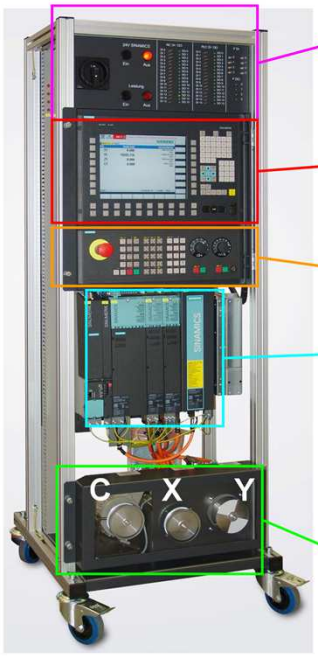
This information is necessary to be able to perform full optimization, i.e.:
- Automatic controller optimization

Information: Software on training PG

Objective: This information should help you to find the required software on the programming device provided for the exercises.

Software The software required for commissioning is located on the PG in directories:
D:\840Dsl_SW_2x_4x\software\ncu_pcu
D:\840Dsl_SW_2x_4x\pg_tools
The software is already preinstalled.

Overview of the training rack



Simulator

Operator panel with TCU

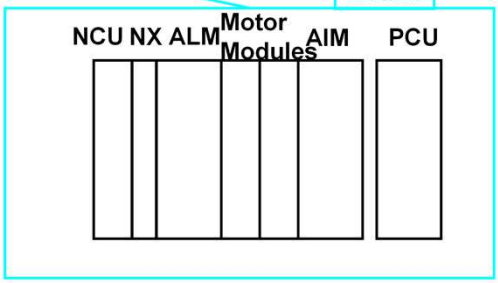
Machine control panel: MCP 483C PN

NCU | Drives | PCU

Motors

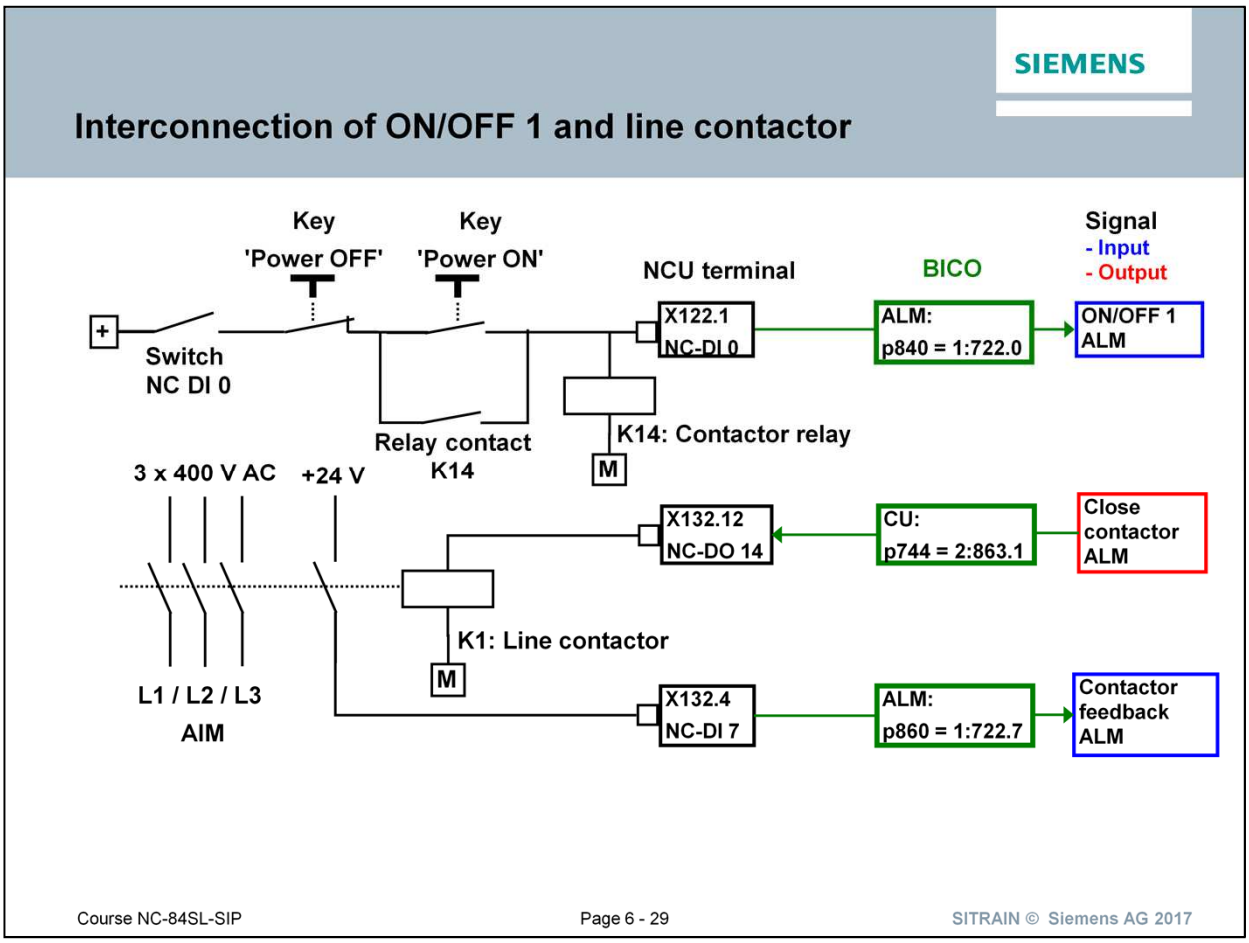
C | X | Y

Detail:



Course NC-84SL-SIP Page 6 - 28 SITRAIN © Siemens AG 2017

Notes



Interconnection ON/OFF 1 = NC-DI 0 (BICO p840[ALM] = 1:722.0)

The signal is switched using the switch NC-DI 0 and the latching relay contact of the contactor relay K14.
The relay K14 itself picks up if:
- Switch 'NC-DI 0' -> ON
AND
- Key 'Power ON' -> ON
Opening of K14 can be performed in 2 ways:
- Switch 'NC-DI 0' -> OFF
OR
- Key 'Power OFF' -> ON

Line contactor control = NC-DO 14 (BICO p744[CU] = 2:863.1)

If line contactor control is active, this output must switch the line contactor.

Line contactor feedback = NC-DI 7 (BICO p860[ALM] = 1:722.7)

The line contactor feedback is expected at this input (contactor is switched on).
The monitoring time is set in P861 = 100 ms (default)

Other ALM signals (regardless of line contactor control):

Control mode = NC-DO 12 (BICO p742[CU] = 2:863.0)
Infeed ready = NC-DO 13 (BICO 743[CU] = 2:899.0)

Exercise: 'Rack with ALM' training device,

Objective: This exercise will give you a better understanding of the main components of the SINUMERIK 840D sl training device and how it is set up and wired. Check the setup (e.g. components, DRIVE-CLiQ wiring, etc.) of the training control. Check that the information given below is correct and complete the sketch of the DRIVE-CLiQ connections.

Line filter Active Interface Module (AIM) 16 kW

Line Module Active Line Module (ALM) 16 kW

Motor Modules 1-axis module 3A
2-axis module 5A / 5A

Motors The motors used on the training rack have the following characteristics:

X: 1FK7044-7AF71 Synchronous motor High Dynamic, 6-pole
Incremental encoder sin/cos 2048 P/U

Nominal data:	1.1 kW	3.5 Nm	4 A	3000 rpm
Standstill data		4 Nm	3.5 A	
Mechanics	7.7 kg	0.000128 kgm²		

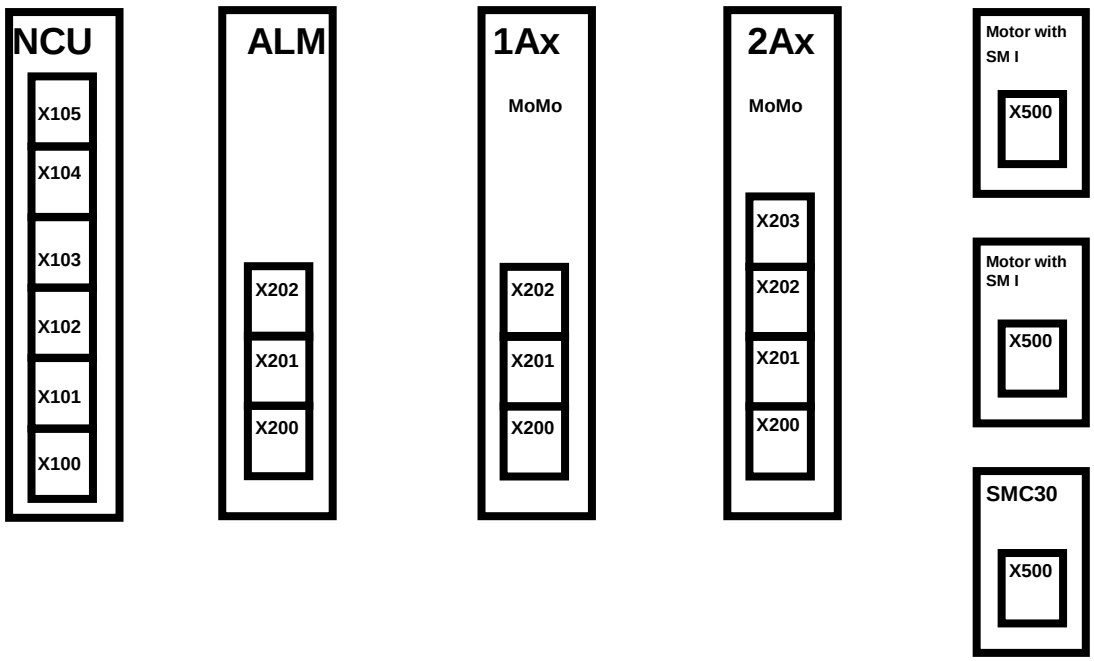
Y: 1FK7060-5AF71 Compact synchronous motor, 8-pole,
with holding brake
and EnDat absolute encoder, 2048 PPR

Nominal data:	1.48 kW	4.7 Nm	3.7 A	3000 rpm
Standstill data		6 Nm	4.5 A	
Mechanics	7.0 kg	0.000795 kgm²	(without brake)	

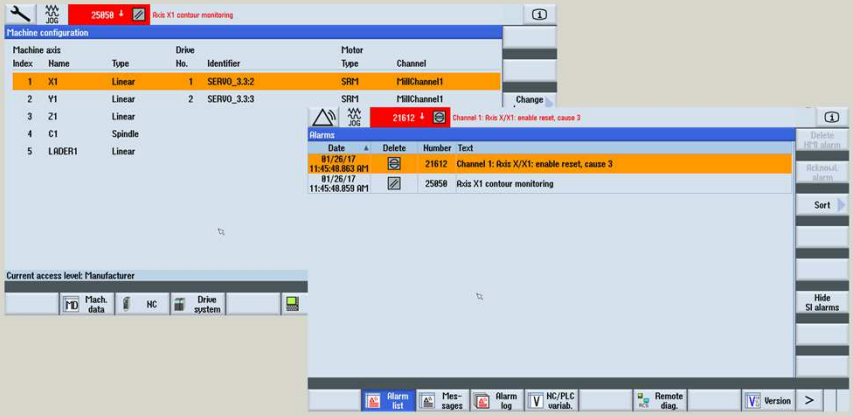
C: 1LA7070-4AB00 Standard induction motor, 4-pole
Encoder 1XP8001-1: HTL, 1024 P/U

Nominal data:	0.25 kW	1.8 Nm	0.77 A	1350 rpm
	Power factor = 0.79			
Mechanics	5.0 kg	0.00052 kgm		

Enter the DRIVE-CLiQ connections.



SIEMENS



Chapter 7

Diagnostics and Setup operating areas

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Setup operating area 3
Reset (po) 4
Setup: Language selection 5
HMI 6
Action log 8
System data 10
Diagnostics operating area 11
Alarms/messages and online help 12
Alarm log 13
NC/PLC variables 14
Remote diagnostics 15
Version 17
Bus TCP/IP 20
Axis diagnostics 25
OFF signals in SINAMICS 26
Axis diagnostics 27
Safety 29
Trace 30
System utilization 31
Drive system 32
Diagnostics exercise: Topology 38

Learning objectives

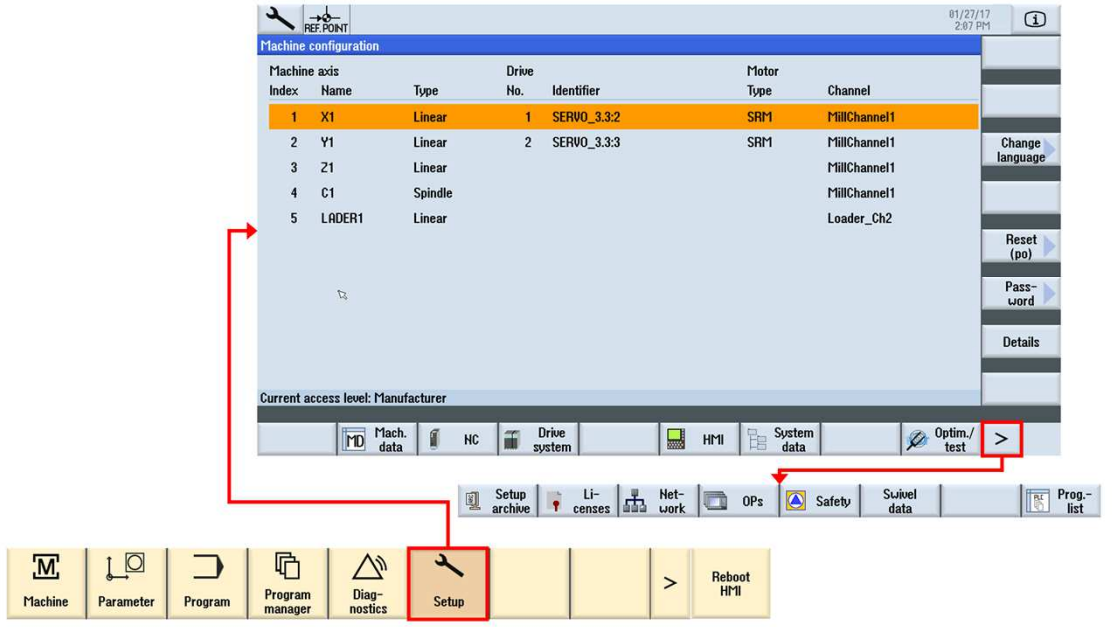
- You will be able to interpret fault displays for axes and drives
- You will be able to set the date, time, and language
- You will be able to evaluate version displays
- You will be familiar with adaptations for NC, PLC, and HMI using the Setup operating area



SITRAIN © Siemens AG 2017

Notes

Setup operating area



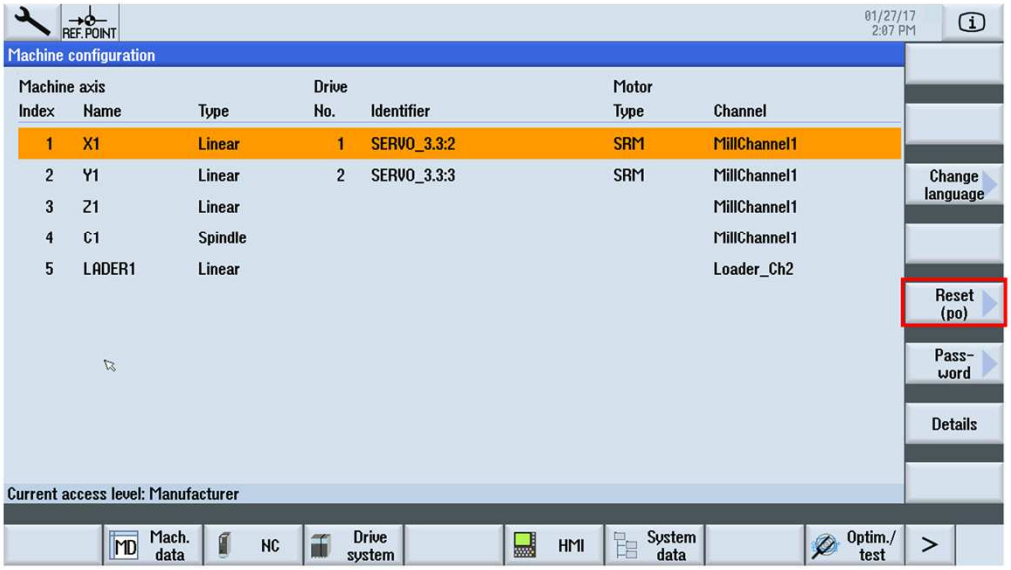
 SITRAIN © Siemens AG 2017

'Setup' operating area

	The following slides provide a description of the menus in the "Setup" operating area. You need to enter a password for all functions.
SK: Mach. data	Display and change machine and setting data. You can find a description of the "Machine data" menu in the chapters on commissioning.
SK: NC	Activate compensations; save your configuration
SK: Drive system	Display and target-actual comparison of detected drive components, including network structure (=topology).
SK: HMI	Selected HMI settings (alarm texts, date/time, logical drives, action log settings)
SK: System data	Data management (archiving/de-archiving/copying/pasting any data).
SK: Optim./test	To optimize controllers (drives, position controllers). Including automatic optimization (AST = Automatic Servo Tuning).
SK: Setup archive	Setup archive and original state (back up or read in data from the NC/PLC/drives)
SK: Licenses	Overview and activation of software licenses by entering a license key.
SK: Network	Display and change the network configuration
SK: OPs	Display of operator panels and VNC connections
SK: Safety	For setting up Safety Integrated

Hint **SK =** Softkey

Reset (po)



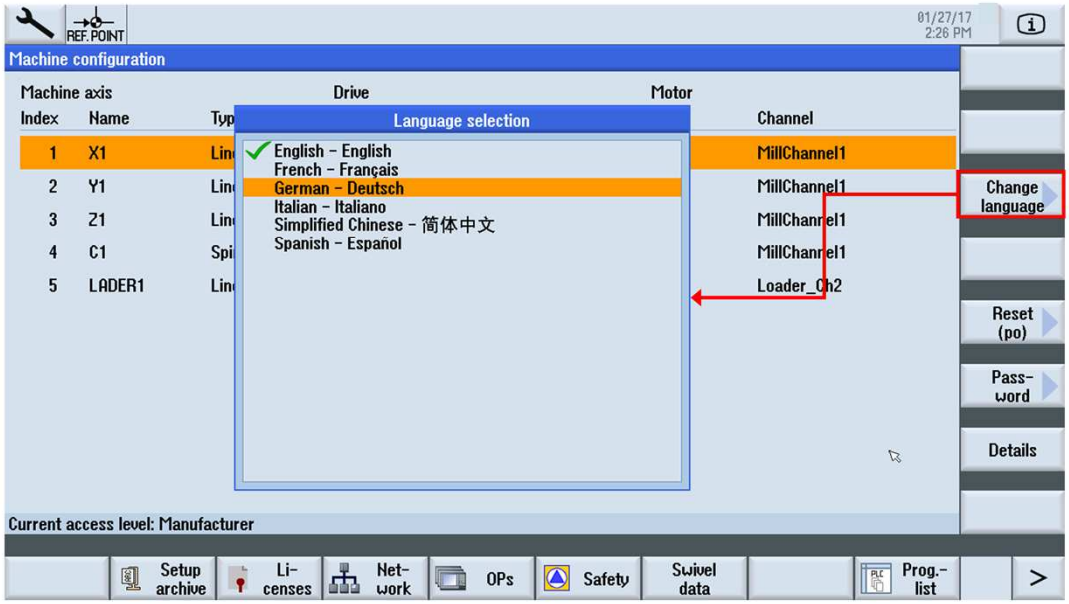
SITRAIN © Siemens AG 2017

SK: Reset (po)

Reset (po) initiates a reboot of the NC, PLC and drive software.
The softkey is selected with:

Operating area ‘Setup’ > SK: ‘Reset (po)’

Setup: Language selection

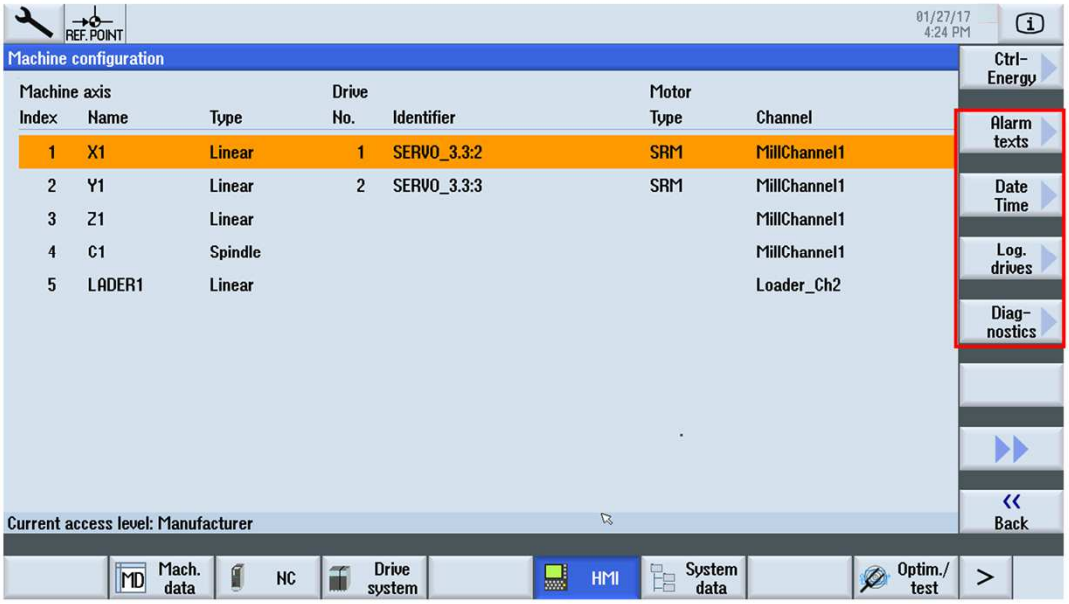


SITRAIN © Siemens AG 2017

Language selection

The language is selected with the "Change Language" softkey.
The new language can be selected directly from the list.

HMI
Alarm texts, date, time ...



SITRAIN © Siemens AG 2017

SK: HMI The "HMI" SK is available in SINUMERIK Operate.

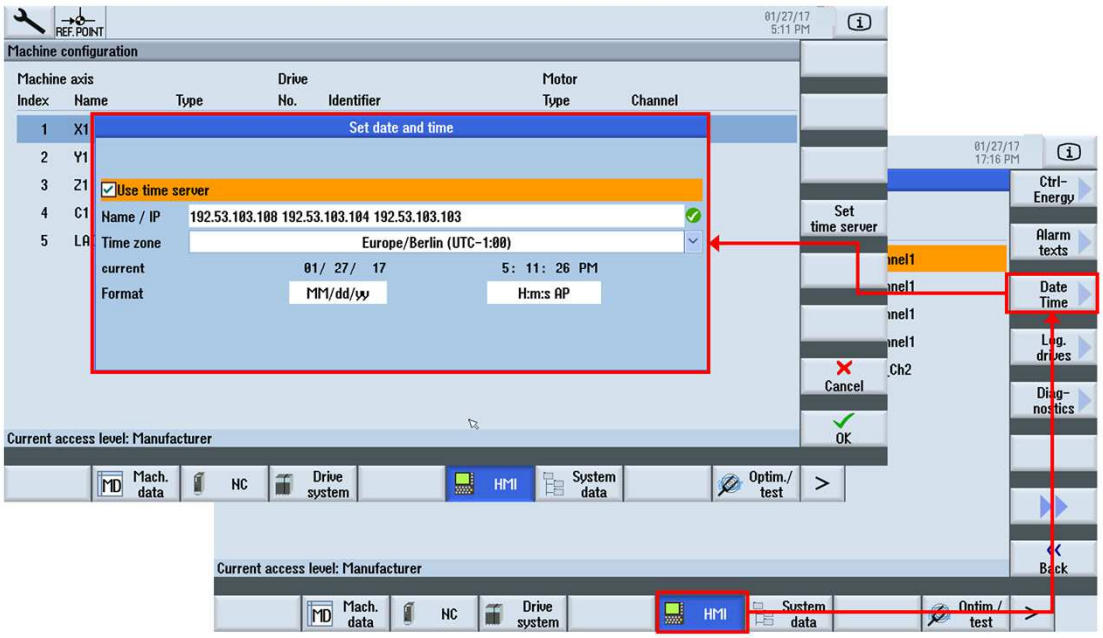
SK: Alarm texts Entering or changing alarm texts at the control.
However, editing the sources and regenerating the alarm texts using the alarm text converter is the preferred approach.

SK: Date/time The date and the time can be set

SK: Log. Drives Logical drives can be configured.

SK: Diagnostics The action log can be set here

HMI
Set date and time



SITRAIN © Siemens AG 2017

Date/time

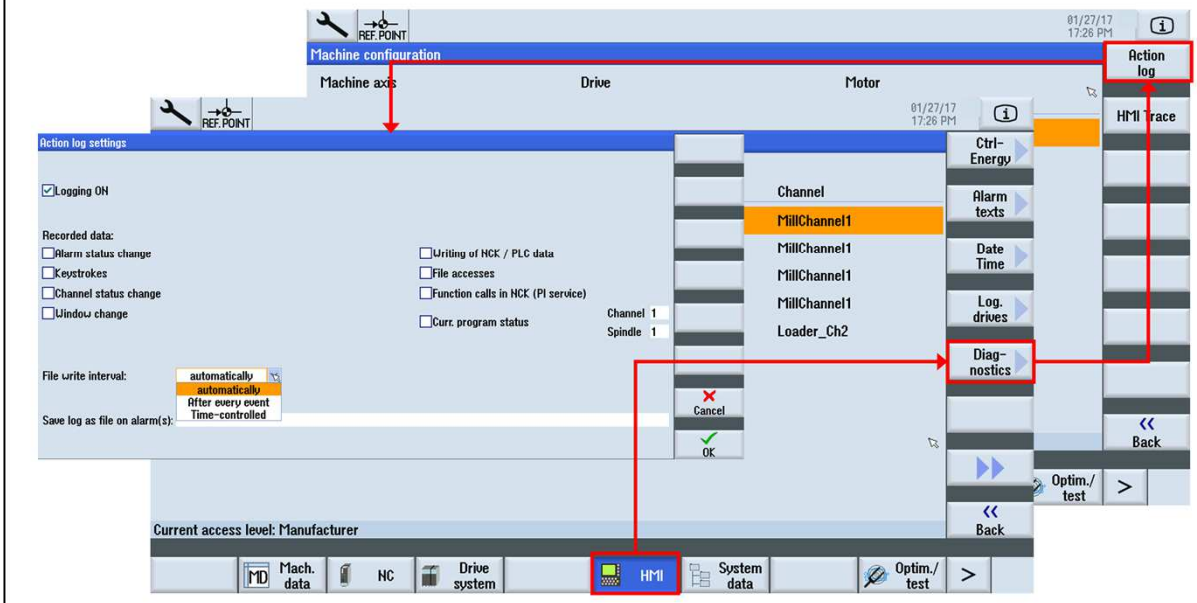
The date and the time can be set in the "Date Time" menu.
Press the "OK" softkey to activate the values entered.

For the alarm display, the date and time are also output as a time stamp; if there is no alarm, the date and time are displayed on the top right.

Time server

If the control system is networked, this flag can be used to obtain the system time from a server. This can be parameterized with the **SK: Set time server**.

Action log Settings



 SITRAIN © Siemens AG 2017

Action log settings

Select with

Operatin Area 'Setup' > SK: 'HMI' >SK: 'Diagnostics' > SK: 'Action log'

Save log as file on alarm(s)



Alarms entered here (entered using their alarm numbers, separated by commas) are direct triggers for immediately saving the current action logs.

Write interval file

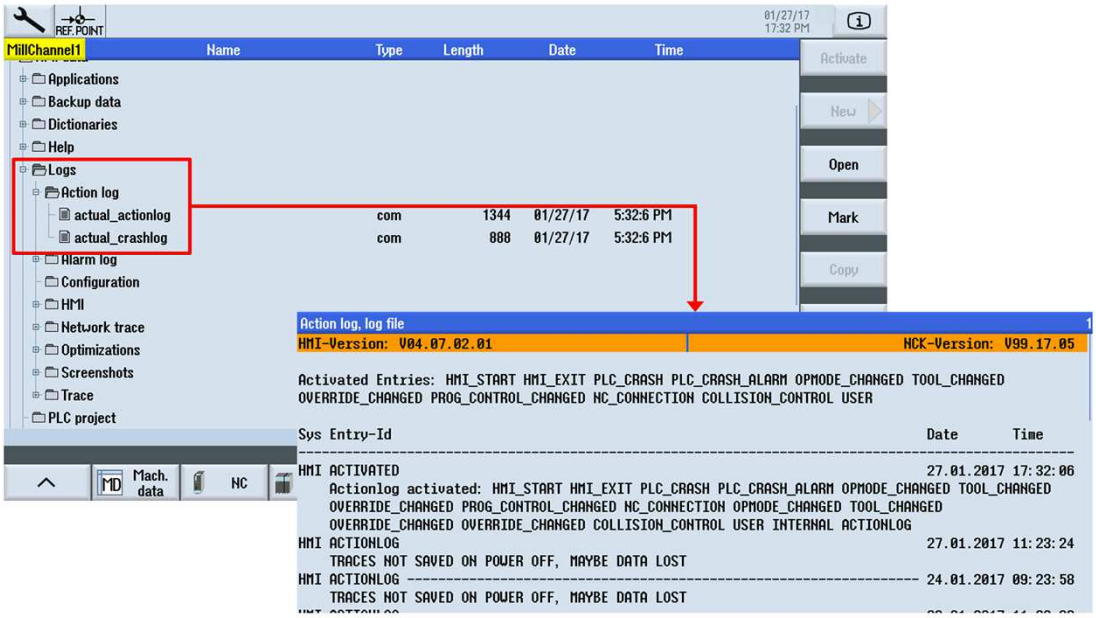


The following can be selected:

- automatically: Entry in internal RAM buffer; if the internal buffer is full, the content is saved on the CF card.
  Entries may be lost!
- After every event: **Not recommended as a default setting due to restrictions on the number of write access attempts to the CF card!**
- Time-controlled (with the time specified in seconds)

Action log
Display

SIEMENS



SITRAIN © Siemens AG 2017

Action log: You can find the log file of the action log like this:

Display

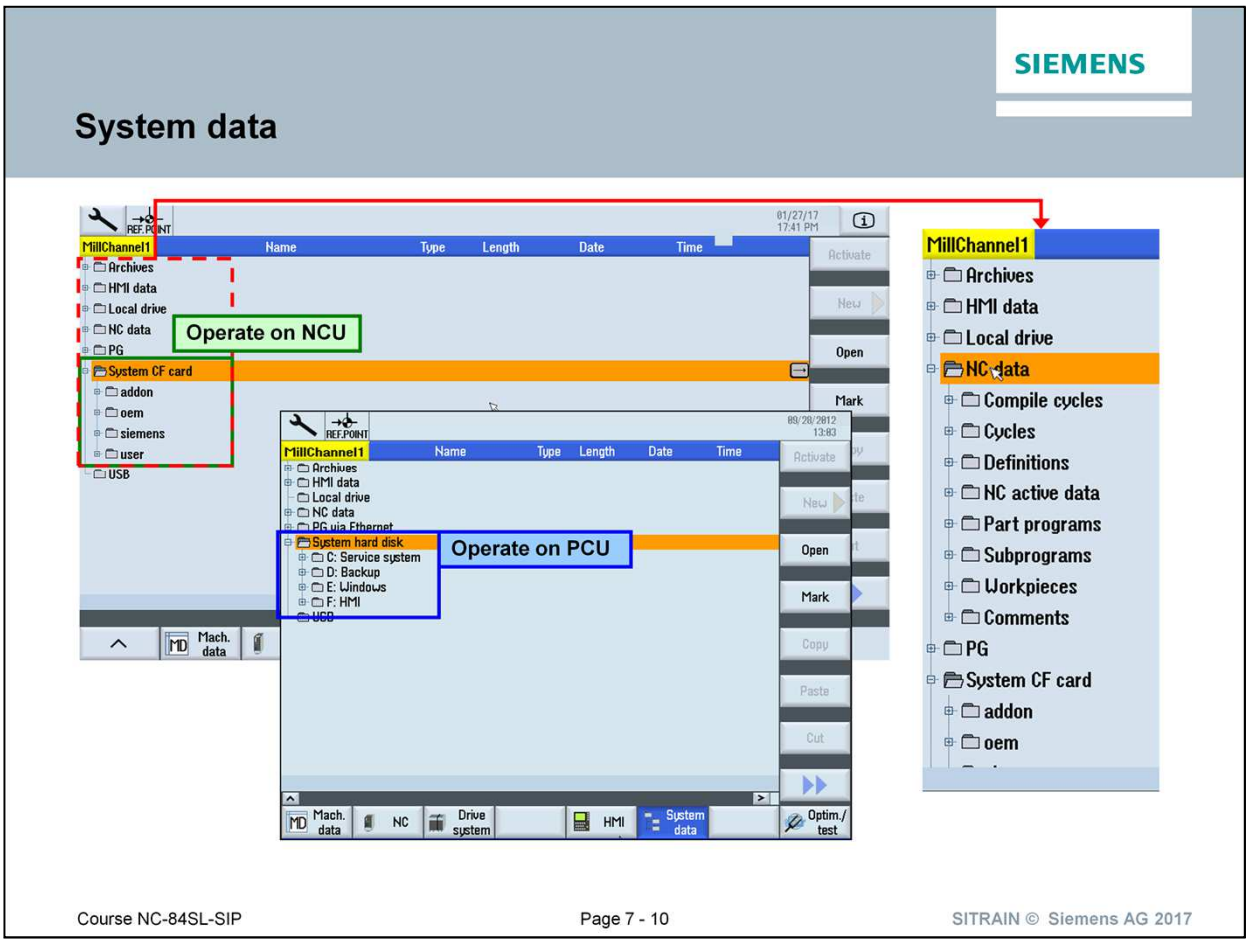
Operating Area: 'Setup' > SK: 'System data' -> directory:
HMI data/Logs/Action log
(This is the /card/user/SINUMERIK/hmi/log directory)

> File: **actual_actionlog.com**

The file can be edited with the **Input** key. In the editor, the softkey:
Save log appears, which generates 2 files at the specified path:

action.com	binary file (copy of file ../hmi/ac_log/action.com)
action.log	text file (can be edited with a text editor)

More details are available in the "Customizing SINUMERIK Operate" chapter



Function

SK: System data is used to access the following drives:
System CF card (Operate on **NCU**)
or **System hard disk** (Operate on **PCU**)

Selected directories on the CF card (hard disk) can be accessed more easily:

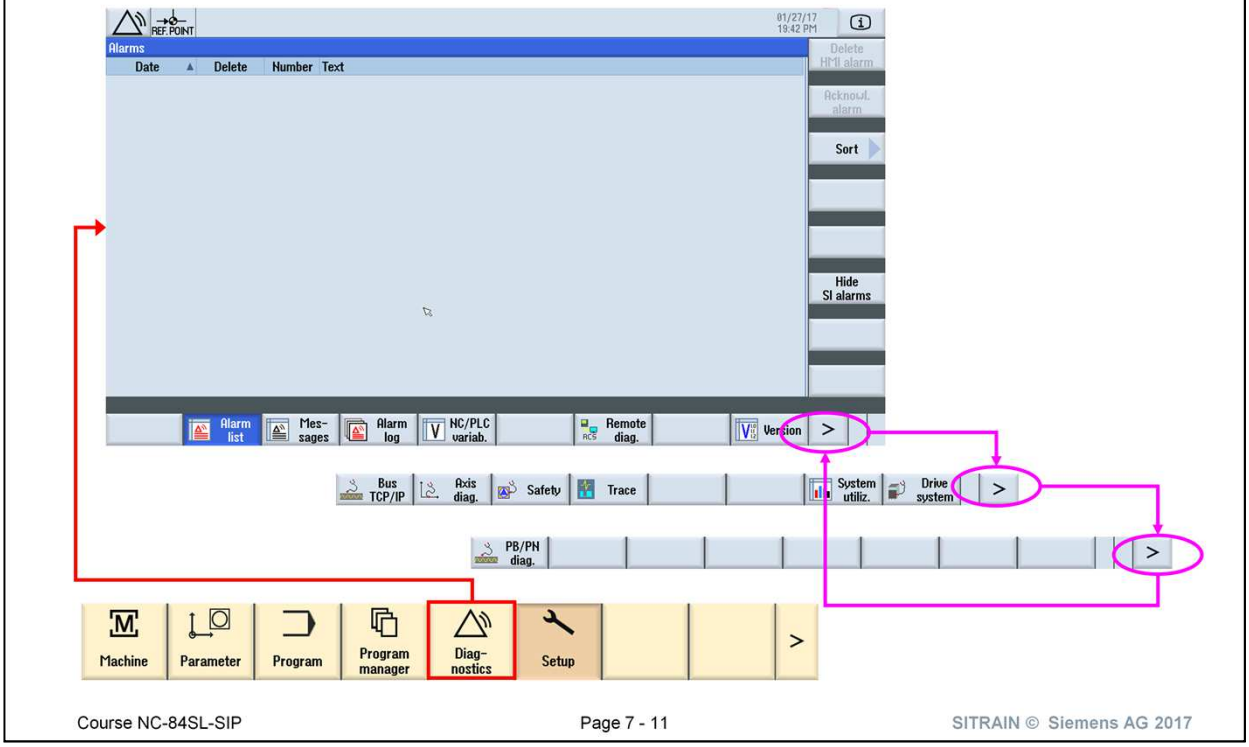
- Archives
- HMI data/local drive
- NC data (= NC control data)
- USB drive
- Additional USB or network drives (logical drives)

Important directories

Some important directories and what they contain:

Directory	File	Meaning
Archive / Manufacturer	*.ARC	Archives of the Series startup
HMI data / Logs / alarm log	alarmlog.com	Alarm log
/ action log	actual_actionlog.com	Action log
NC data / cycles	*.SPF	Cycles
/ definitions /	*.DEF	User data, macros
/ NC active data		NC active data, e.g. compensations
/ part programs	*.MPF	Part programs
/ subprograms	*.SPF	Subprograms
/ workpieces /..	*.WPD	Workpiece directories

Diagnostics operating area



 SITRAIN © Siemens AG 2017

'Diagnostics' operating area

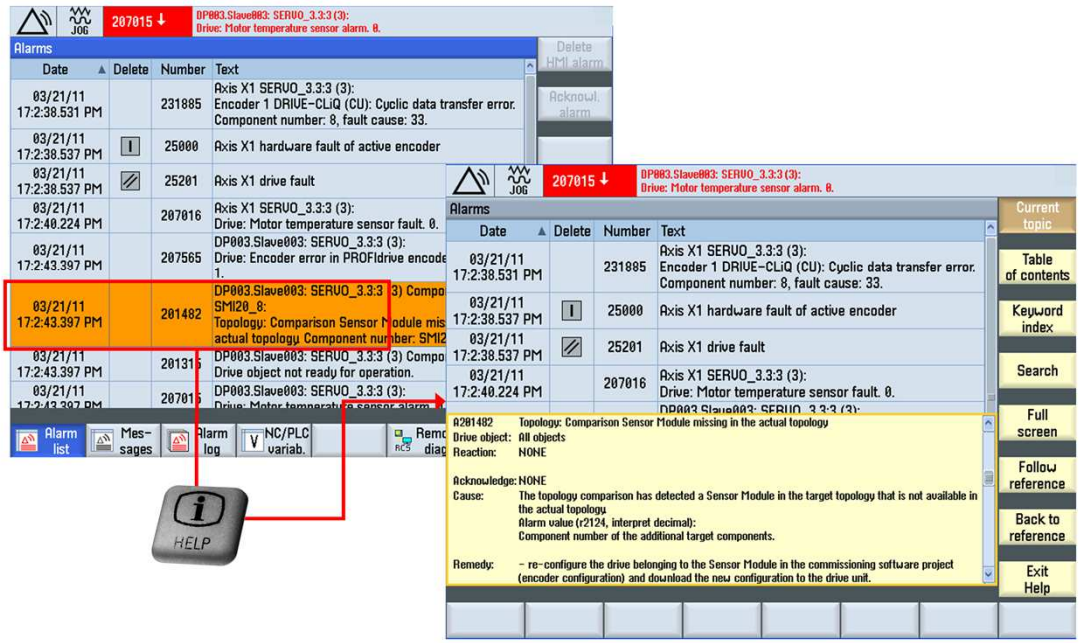
The functions of the "Diagnostics" operating area are explained on the following slides.

SK: Alarm list and SK: Messages

The actual alarms or messages are displayed in the main window.

SK: Alarm log	Displays the alarm log. The alarm and message history is displayed and can be configured.
SK: NC/PLC variab.	NC variables and PLC signals can be monitored/visualized.
SK: Remote diag.	Access via remote diagnostics (VNC Viewer) can be configured.
SK: Version	Version display of the installed software and hardware
SK: Bus TCP/IP	Diagnostics for PROFIBUS and Ethernet interfaces
SK: Axis diag.	Information about the axis and its drive, e.g. following error, motor temperature, DC link voltage, etc.
SK: Safety	State of Safety Integrated monitoring functions
SK: Trace	Tracing NC/PLC or drive signals
SK: System utiliz.	Display of the system utilization
SK: Drive system	Diagnostics information of the drives (enable signals, alarms, interconnection errors for DRIVE-CLiQ, etc.)
SK: PB/PN diag,	PROFIBUS and PROFINET diagnostics

Alarms/messages and online help



SK: Alarm list

All pending alarms are displayed in the alarm list with alarm numbers, date, delete criteria, and explanation.

Press the Info key to display information for troubleshooting (Diagnostics Guide) (PCU 50 or PG/PC).

Clear the alarm by pressing the key that is displayed as a symbol:



Power ON or softkey: "Reset (po)"



Reset key on the machine control panel



"Acknowledge alarm" key on the operator panel



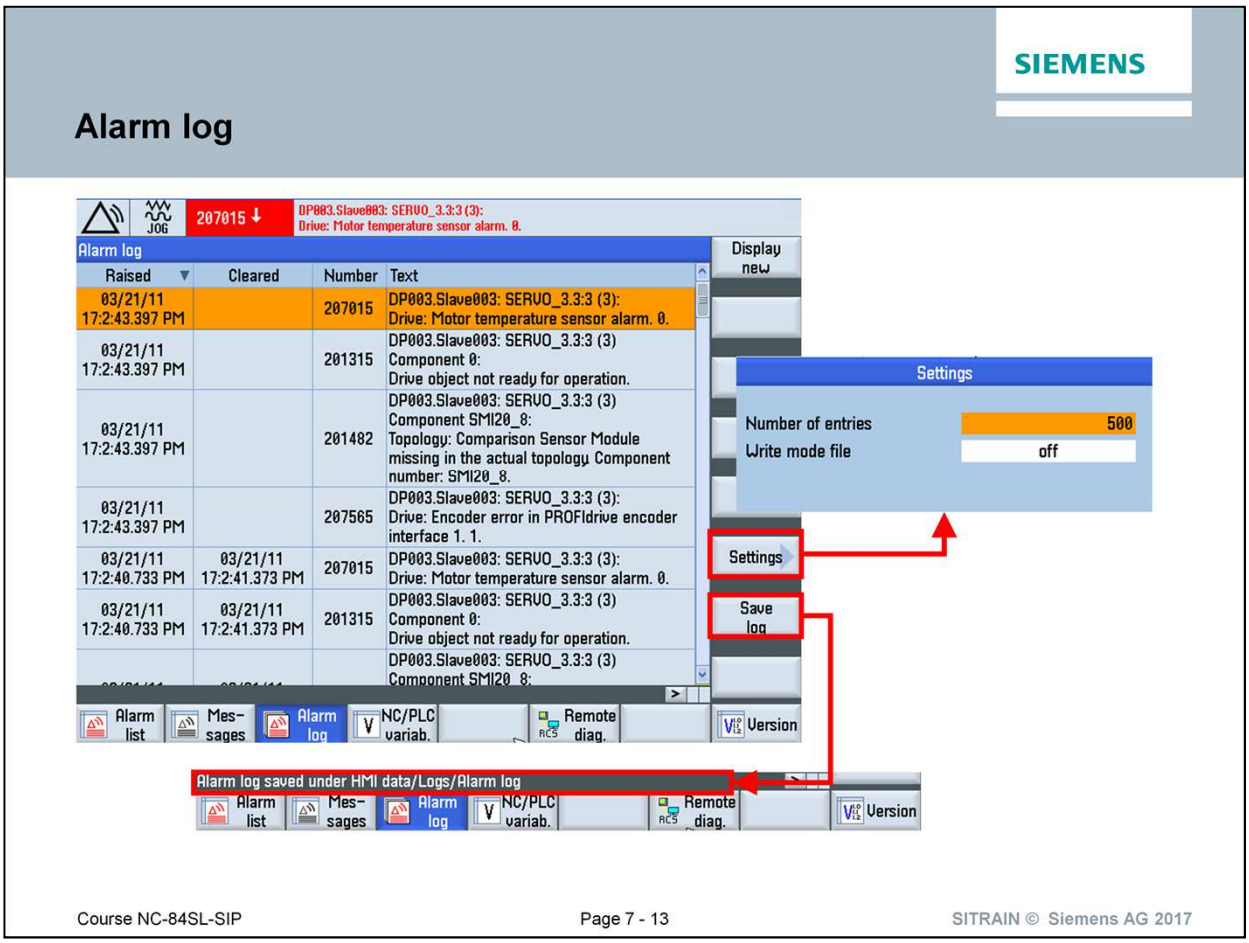
"NC Start" key on the machine control panel



"Recall" key on the operator panel

SK: Messages

PLC status messages that do not have to be acknowledged are displayed.



SK: Alarm log A log containing the alarms and messages that have occurred so far is displayed.

- 'Raised' column Time stamp when the alarm occurs

- 'Cleared' column Time stamp when the alarm disappears

- Current alarms These are listed under 'Raised', but not under 'Cleared'

SK: Save log The alarm log is saved in directory

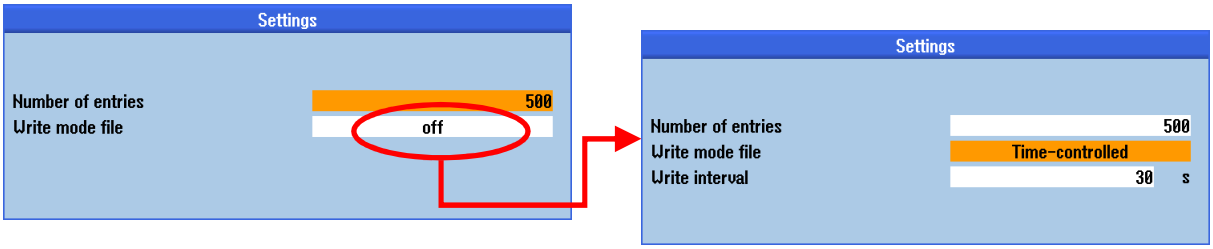
HMI data/Logs/Alarm log

in file

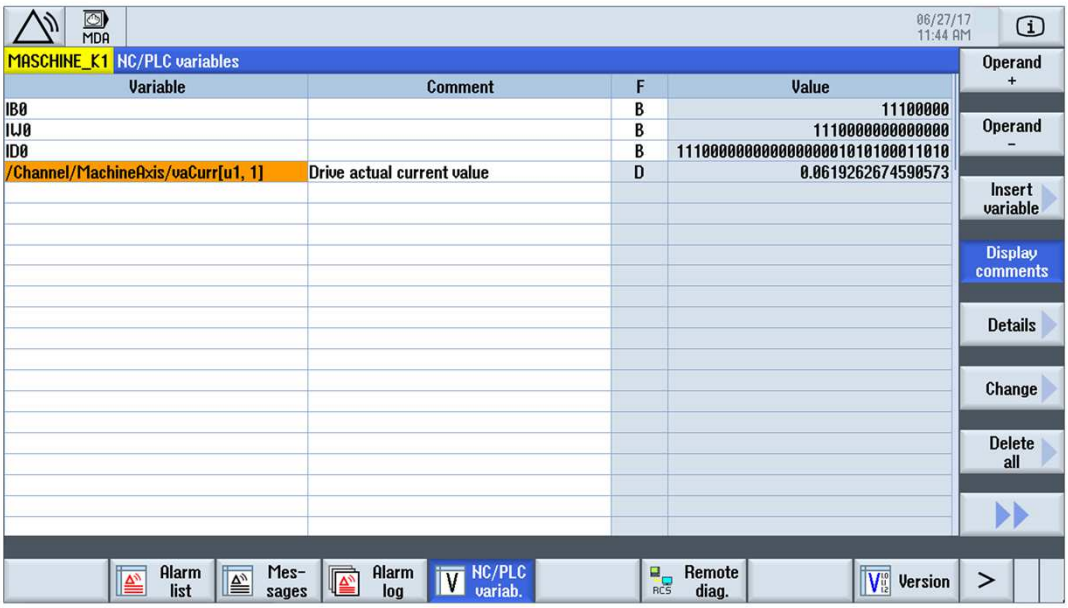
Alarmlog.txt.

SK: Settings Here, the number of entries and the write mode can be set.

Example: Time-controlled (but CAUTION: Memory can run low!)
The write interval is specified in seconds.



NC/PLC variables



 SITRAIN © Siemens AG 2017

Display of the variables

The "Diagnostics" menu contains the "NC/PLC variables" function where PLC interface signals and NC variables can be monitored (and in some cases changed).

'Format' column Default setting for the display format (binary, hex, decimal, floating point)

SK: Operand + / - The preselected operand is incremented or decremented by 1 byte or 1 bit, etc.

SK: Insert variable The variable can be selected from a list here.

SK: Display comments Comments previously entered by the user about the variables are displayed

SK: Change Setting and resetting PLC signals.
The PLC user program resets variables that have been set.
This function requires a password, e.g. SUNRISE.
The **SK: OK** is available to activate values that have been changed.

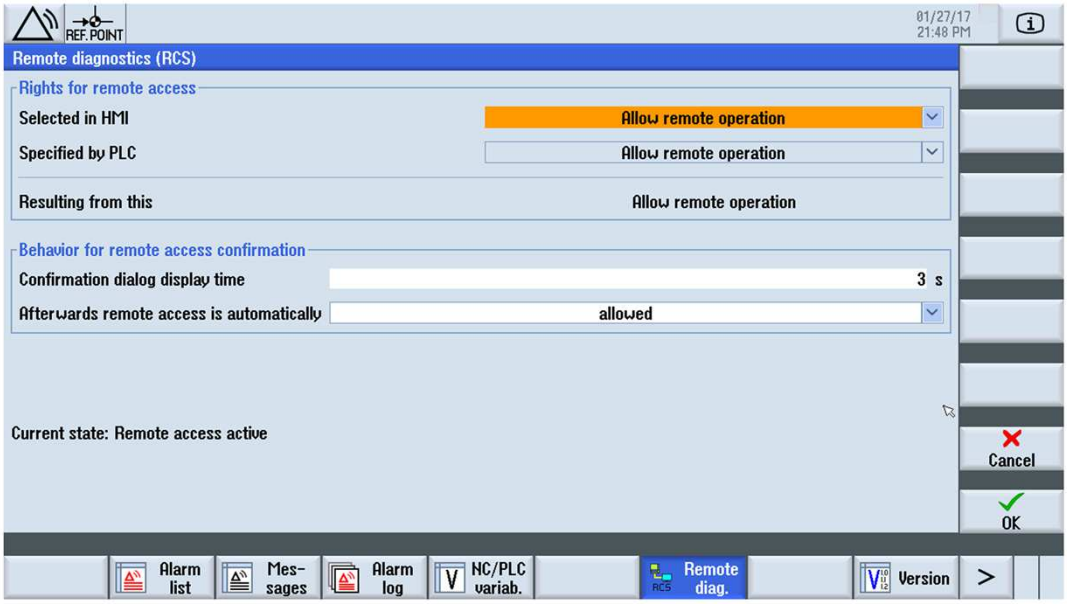
SK: Delete all All entries in the table are deleted.

>> Load symbols A *.snh file generated using the PLC symbol generator is loaded.

>> Save screen form Saves the selected entries in an xml file.

>> Load screen form Loads the entries from the selected xml file.

Remote diagnostics

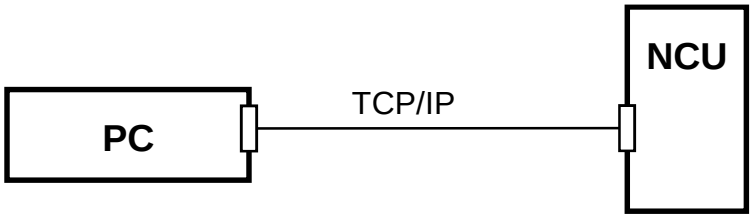


 SITRAIN © Siemens AG 2017

Function The configuration of the remote diagnostics is displayed and can be modified.

The remote diagnostics is based on the following configuration:

Hardware:



Software: **VNC Viewer** **VNC Server**

PC and NCU are connected via TCP/IP.

A VNC server runs on the NCU - and a VNC viewer runs on the PC (**VNC** = **V**irtual **N**etwork **C**omputing).

SK: Change The settings can be changed

Example In the example shown above, remote control is in principle permitted.

However, an access attempt must be acknowledged within 20 seconds at the control using OK - otherwise, the attempt is rejected.

tcu.ini The settings are saved in file:

/card/user/system/etc/tcu.ini for Operate on NCU

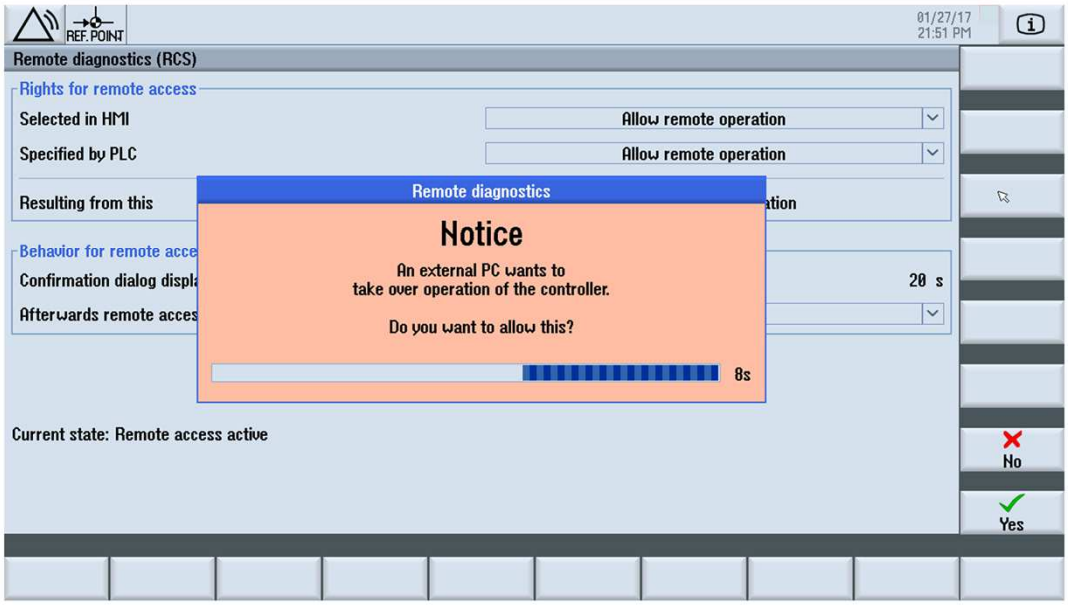
or **c:\ProgramData\Siemens\MotionControl\system\etc\tcu.ini** for Operate on PCU

in section:

[VNCViewer]

Remote diagnostics
Access attempt

SIEMENS



Access attempt

The attempt to access the control from a remote PC is displayed on the machine.

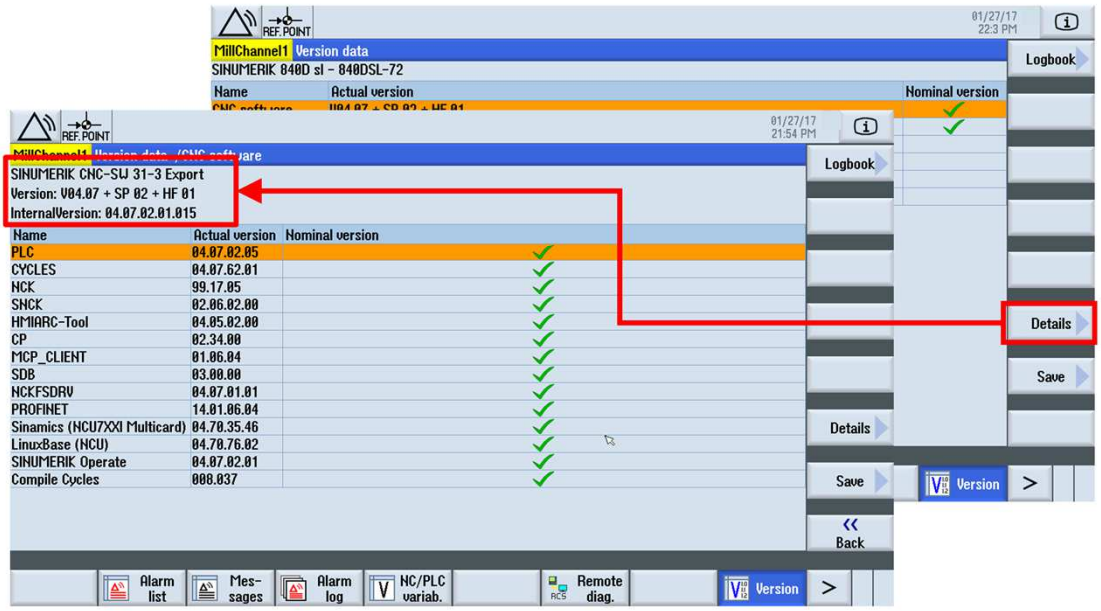


Permits access



Rejects access

Version

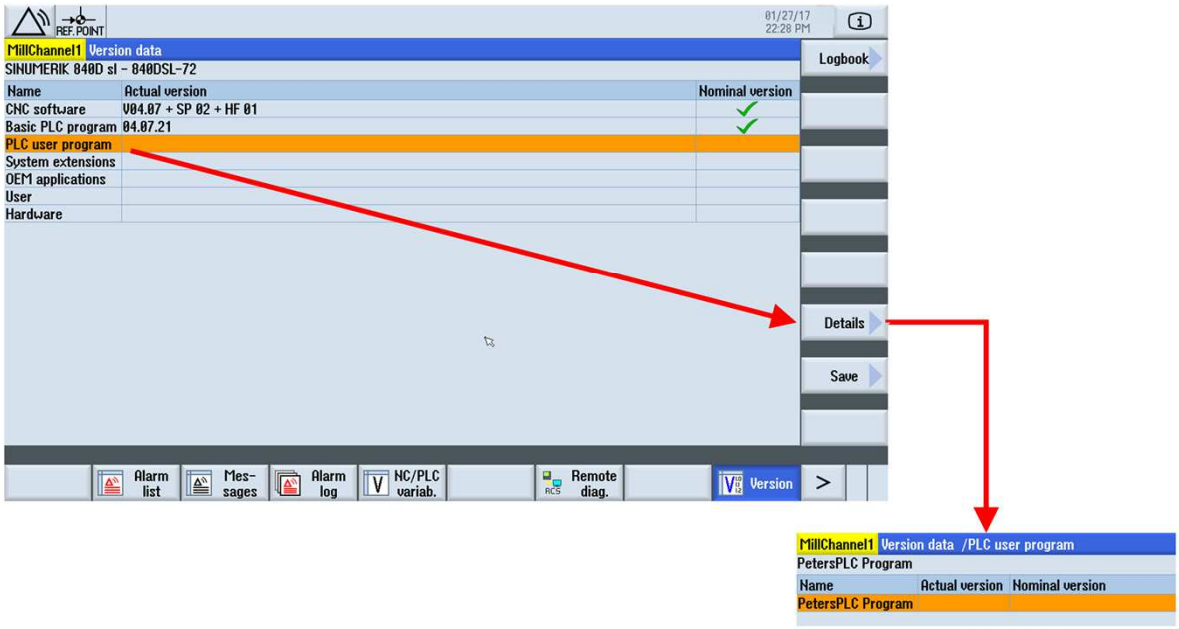


SITRAIN © Siemens AG 2017

Function The version data of the installed system software and the hardware is displayed.

Version
PLC: Details

SIEMENS



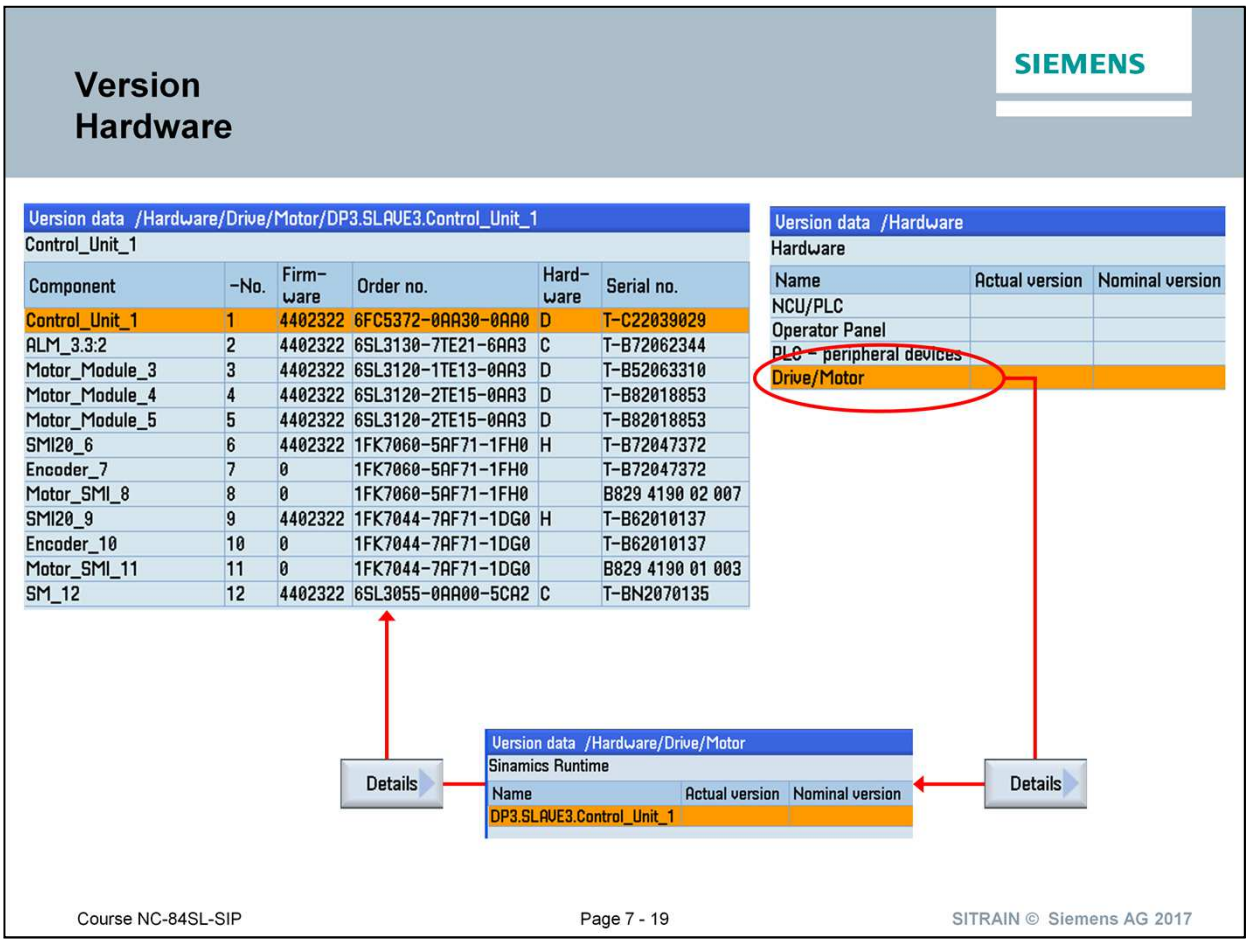
 SITRAIN © Siemens AG 2017

Function

The version data of the basic PLC program and a version string of the PLC user program are displayed.
Via the softkey Details, you can, for example, view the internal version designations of the function unit.

PLC user program

With the softkey Details, you can display a string generated by the machine manufacturer here.
This string is from the FB1 call in OB100 and is entered in the 'UserVersion' parameter.



Hardware

-> SK: Details

The hardware version of the DRIVE-CLiQ components is displayed.

> Selection: 'Hardware' in the version display

> SK: Details -> Name 'Drive/Motor' is displayed

> SK: Details -> Name 'DP3.SLAVE3.Control_Unit_1' is displayed (possibly along with other CUs, if there are NX modules)

> SK: Details

-> The hardware versions of the DRIVE-CLiQ components are now displayed.

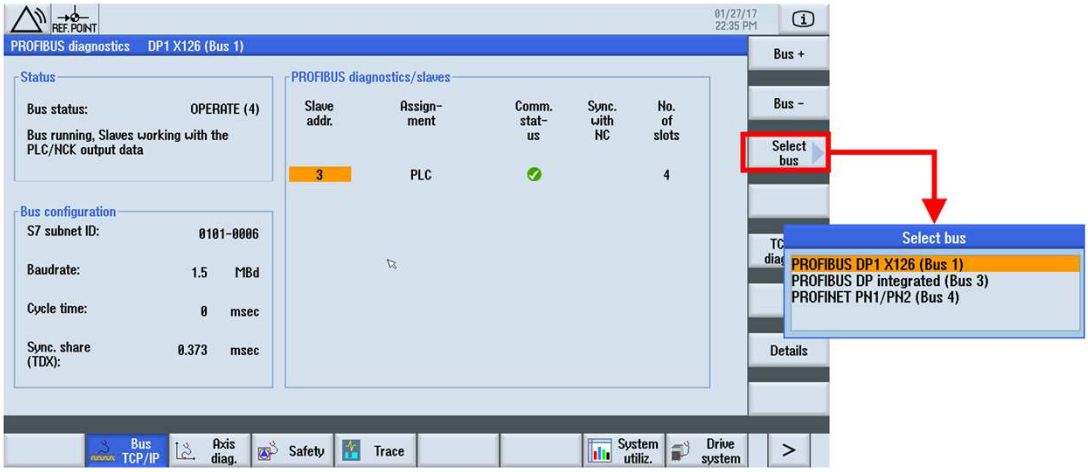
The following information is displayed:

- Firmware version
- Order number (MLFB)
- Serial number

NOTICE

You might have to open the 'Topology' screen first before the hardware versions are displayed.

Bus TCP/IP
PROFIBUS/Ethernet diagnostics



SITRAIN © Siemens AG 2017

PROFIBUS diagnostics In the Diagnostics operating area > softkey: Bus TCP/IP, information can be found about PROFIBUS nodes of the PROFIBUS interfaces. In the overview, use the **Bus + /Bus -** or Select bus keys to select the bus you wish to display.

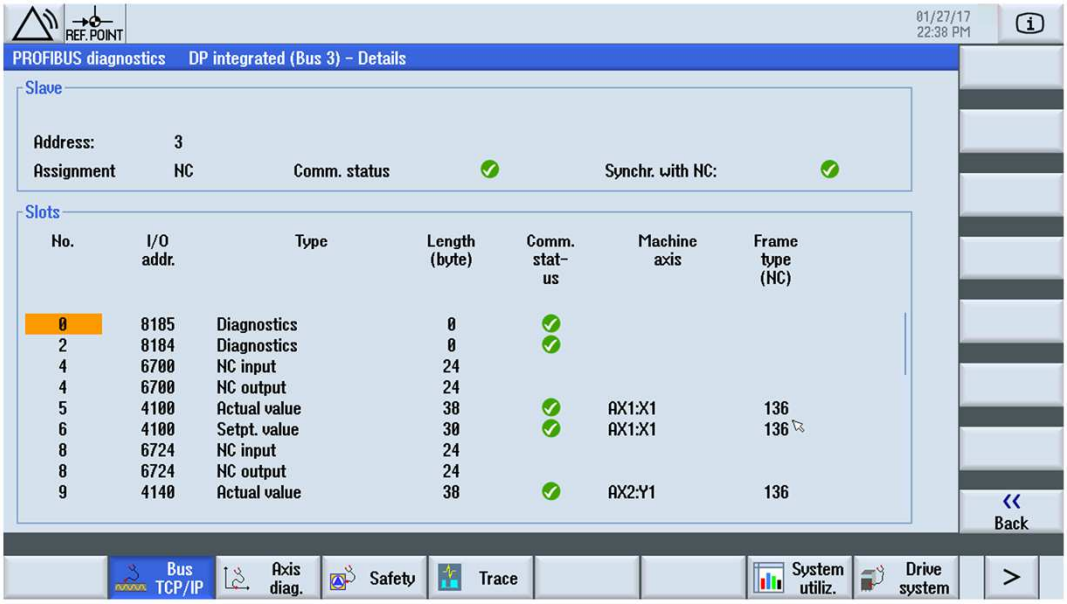
SK: Bus + / Bus - Using these softkeys, you can toggle between PROFIBUS systems.

SK: Select bus This softkey allows the PROFIBUS system to be selected directly from a list.

Remark A PROFIBUS DP1 X126 or DP2 X136 is only displayed (using SK: 'Select bus' or 'Bus +' / 'Bus -') if, in HW Config, at least one station is connected to the bus!

TCP/IP diagnostics You can select TCP/IP diagnostics using this softkey.

Bus TCP/IP
PROFIBUS diagnostics – Drive details

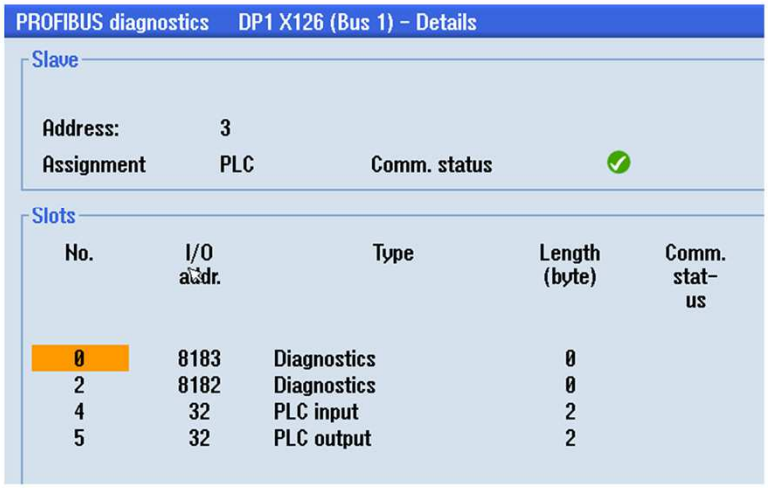


SITRAIN © Siemens AG 2017

Function The logical drive number, the PLC I/O address, and the axis assignment, for example, can be taken from the details about a drive bus node.

Note Press the "Back" button to go back to the overview screen.

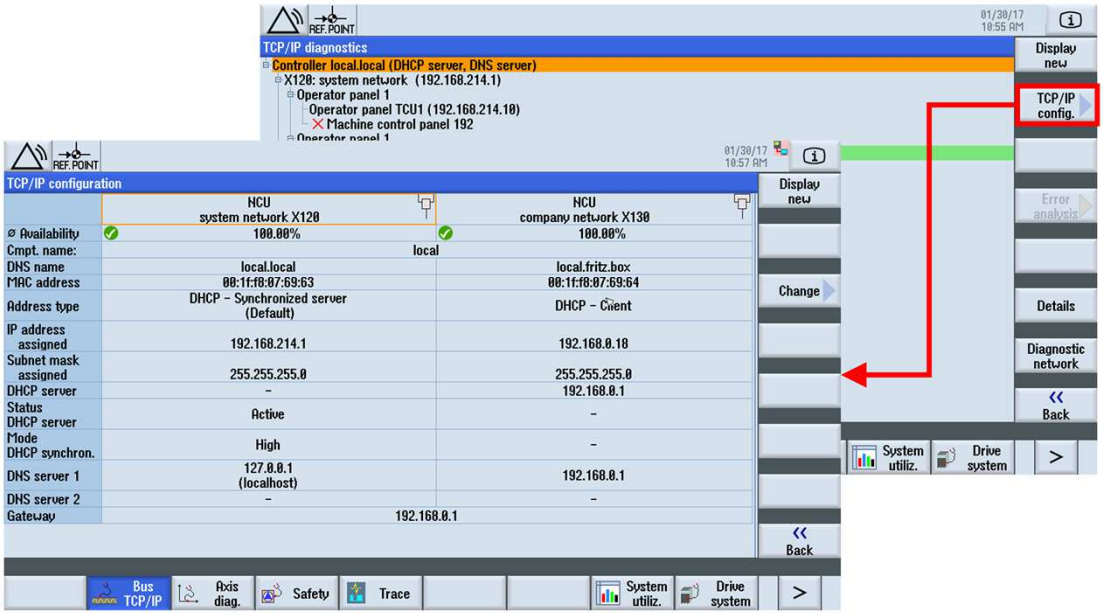
Bus TCP/IP
PROFIBUS diagnostics – PLC I/O details



SITRAIN © Siemens AG 2017

Function The assignment and addressing of the I/O modules is displayed.

Note Press the "Back" button to go back to the overview screen.



TCP/IP diagnostics Using this softkey, the TCP/IP diagnostics for X120 and X130 of the NCU are displayed.

-SK: TCP/IP Config.

Details on the Ethernet configuration can be seen, e.g.:

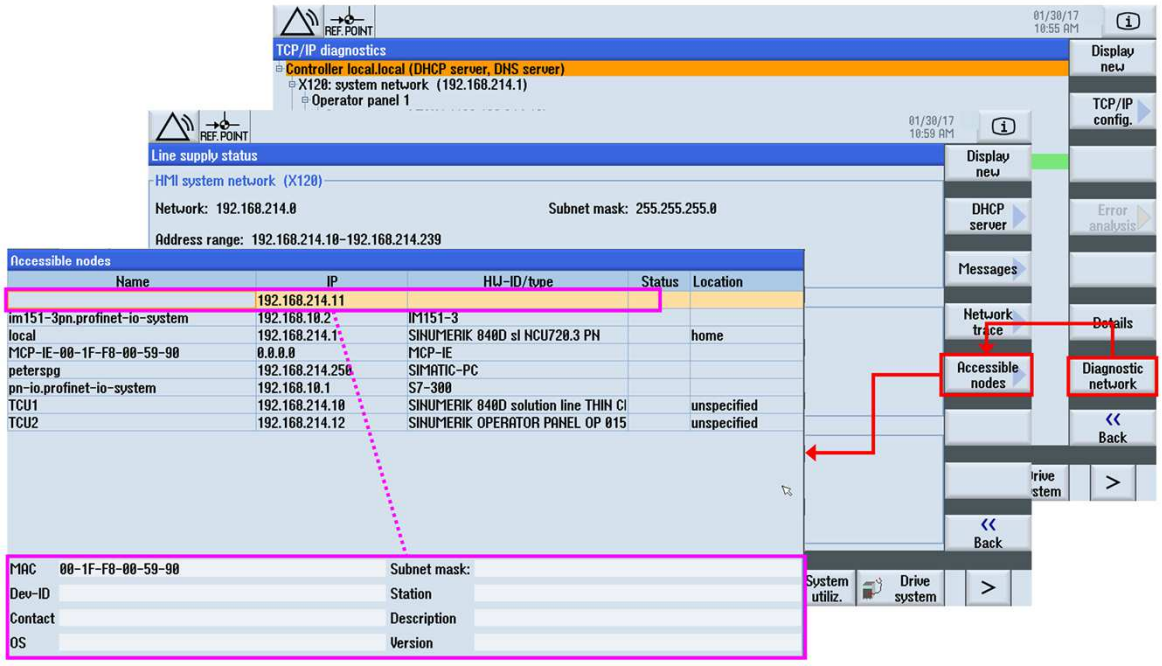
- Computer name
- DHCP configuration
- IP address, subnet mask

- SK: Diagnostic network

Here, DHCP servers and accessible nodes can be displayed.

Note Press the "Back" button to go back to the overview screen.

Bus TCP/IP
Diagnostic network -> Accessible nodes



Diagnostic network With this softkey, the network diagnostics of the NCU (or PCU) is displayed.
The complete path to it is:
Operatin Area: Diagnostics > ETC key > SK: Bus TCP/IP > SK: TCP/IP diagnostics > SK: Diagnostic network

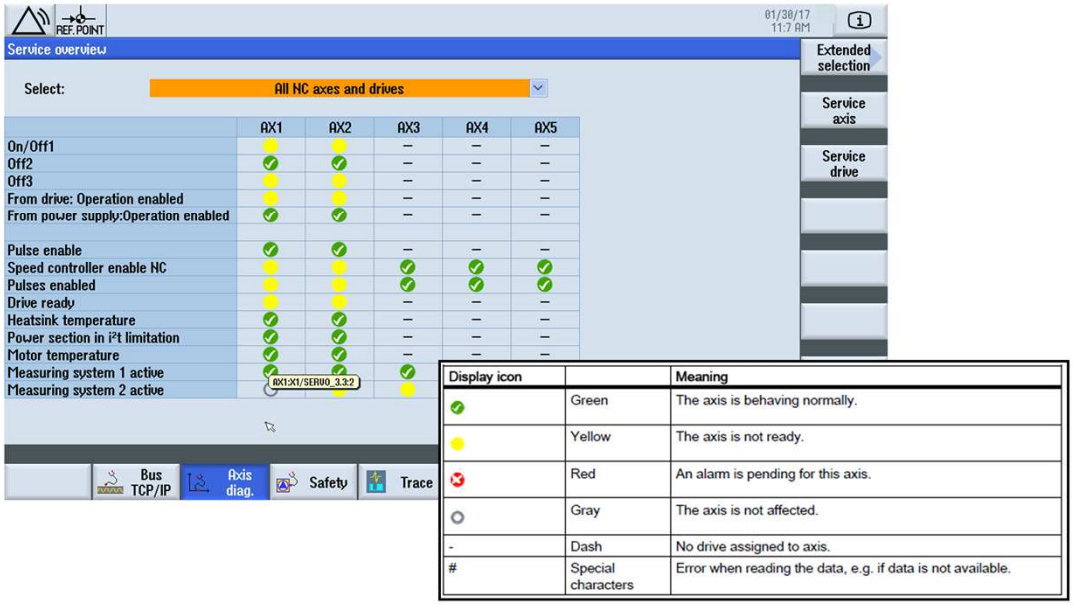
Accessible nodes These can be displayed as follows:
> SK: Accessible nodes
That triggers an IP scan in the address range of the NCU or PCU.
This screen lists all accessible TCP/IP network nodes.
Further details on each node appear below (MAC address ...)

Additional nodes With this softkey, further IP scans are triggered.

Save This softkey can be used, e.g. to save the actual topology as
- Target topology
- Manufacturer topology
- Current topology (XML or CSV format)

Compare This compares the actual topology with a saved topology
- Target topology
- Manufacturer topology
- Current topology (XML format)

Axis diagnostics



Display icon		Meaning
	Green	The axis is behaving normally.
	Yellow	The axis is not ready.
	Red	An alarm is pending for this axis.
	Gray	The axis is not affected.
-	Dash	No drive assigned to axis.
#	Special characters	Error when reading the data, e.g. if data is not available.

SITRAIN © Siemens AG 2017

Function

With the softkey: "Axis diagnostics" - drive and axis signals are displayed, which are important for axis travel.

Drive diagnostics

- ON/OFF1
- OFF2
- OFF3
- From drive: Operation enabled
- From infeed: Operation enabled

Axis diagnostics

- Pulse enable NC ->DB31 ... DBX21.7
- Speed controller enable NC -> DB31 ... DBX2.1
- Pulses enabled
- Drive ready
- Heatsink temperature (warning)
- Power unit in i²t limitation
- Motor temperature (warning)
- Meas. system 1 active -> DB31 ... DBX1.5
- Meas. system 2 active -> DB31 ... DBX1.6

Note

Other signals can also be reset as a consequence of a missing enable signal!

SK: Service axis

Detailed information on individual axes of the NC is displayed.

SK: Service drive

Detailed drive information is displayed.

Further diagnostics

- "Drive system", "Safety" menus

OFF signals in SINAMICS

Signal	Meaning
ON/OFF1	**0 = OFF,** drive is stopped via the ramp-function generator ramp (p1121) followed by pulse inhibit **1 = ON,** operating condition
OFF2	**0 = Coast down**, pulse inhibit: Motor coasts down **1 = No coast down,** operating condition
OFF3	**0 = Quick stop,** stop via p1135, followed by pulse inhibit **1 = No quick stop,** operating condition

SITRAIN © Siemens AG 2017

OFF1

- Immediately setting n_set = 0 decelerates the drive along the ramp-function generator ramp down (p1121).
- When zero speed is detected, the motor holding brake (if parameterized) is applied (p1215). The pulses are suppressed when the brake application time (p1217) expires. Standstill is detected when the speed actual value falls below the speed threshold (p1226) or when the monitoring time (p1227) started when speed setpoint <= speed threshold (p1226) has expired.

OFF2

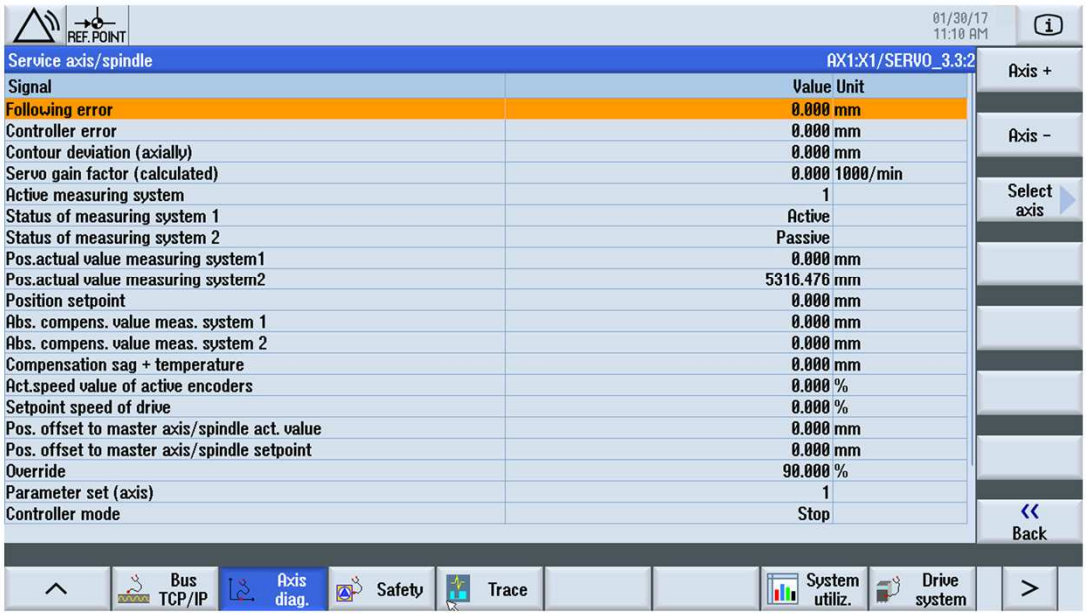
- Immediate pulse suppression, the drive "coasts" to a standstill.
- The motor holding brake (if parameterized) is closed immediately.
- Switching-on inhibited is activated.

OFF3

- Immediately setting n_set = 0 decelerates the drive along the OFF3 ramp down (p1135).
- When zero speed is detected, the motor holding brake (if parameterized) is applied. The pulses are suppressed when the application time of the holding brake (p1217) expires. Standstill is detected when the speed actual value falls below the speed threshold (p1226) or when the monitoring time (p1227) started when speed setpoint <= speed threshold (p1226) has expired.
- Switching-on inhibited is activated.

Axis diagnostics
Service axes

SIEMENS



 SITRAIN © Siemens AG 2017

Function

The information in the "Service axis" display is used to

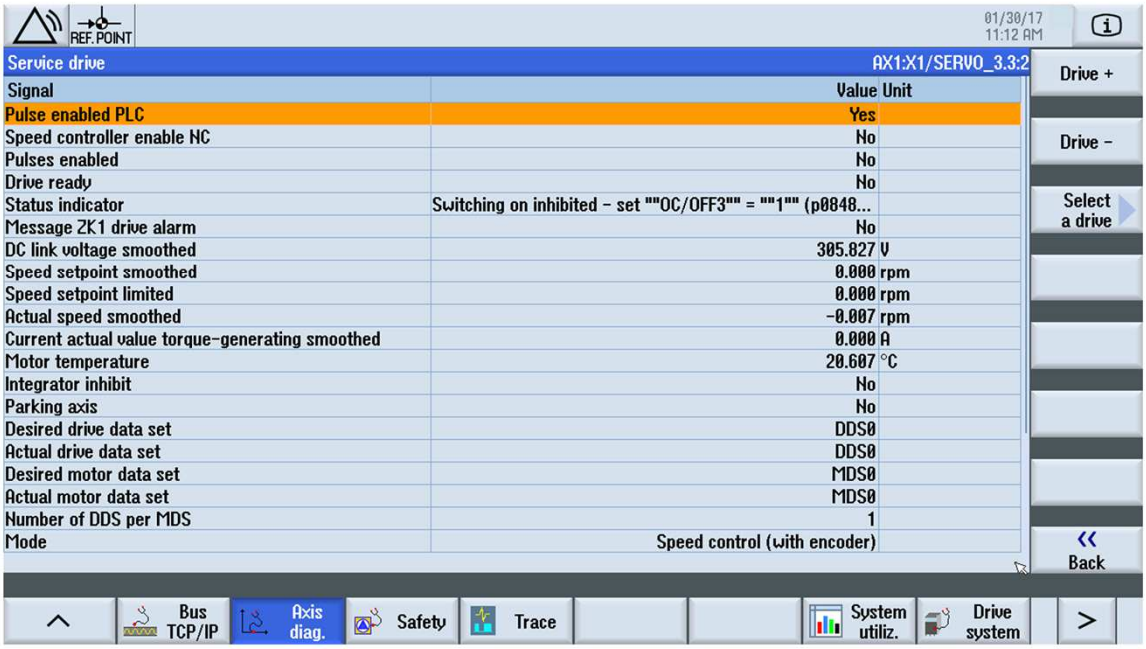
- Check the setpoint branch (e.g. position setpoint, speed setpoint, spindle speed setpoint programmed)
- Check the actual value branch (e.g. actual position value, measuring system 1 or 2, actual speed value)
- Optimize the axis position control loop (e.g. following error, system deviation, servo gain factor)
- Check the entire control loop of the axis (e.g. through position setpoint/ actual-value comparison and speed setpoint/actual-value comparison)
- Check for hardware faults (e.g. check the encoder: if the axis is moved mechanically, the actual value must change)
- Check for compensations
  (e.g. compensation values = spindle pitch/measuring system error compensation, sag/temperature compensation)
- Check the axis monitoring functions

SK: Axis +/-

The service values of the next (+) and the previous (-) axes are displayed.

Axis diagnostics
Service drive

SIEMENS



Signal	Value	Unit
Pulse enabled PLC	Yes	
Speed controller enable NC	No	
Pulses enabled	No	
Drive ready	No	
Status indicator	Switching on inhibited - set ""OC/OFF3"" = ""1"" (p0848...	
Message ZK1 drive alarm	No	
DC link voltage smoothed	305.827	V
Speed setpoint smoothed	0.000	rpm
Speed setpoint limited	0.000	rpm
Actual speed smoothed	-0.007	rpm
Current actual value torque-generating smoothed	0.000	A
Motor temperature	20.607	°C
Integrator inhibit	No	
Parking axis	No	
Desired drive data set	DDS0	
Actual drive data set	DDS0	
Desired motor data set	MDS0	
Actual motor data set	MDS0	
Number of DDS per MDS	1	
Mode	Speed control (with encoder)	

Course NC-84SL-SIP Page 7 - 28 SITRAIN © Siemens AG 2017

Function

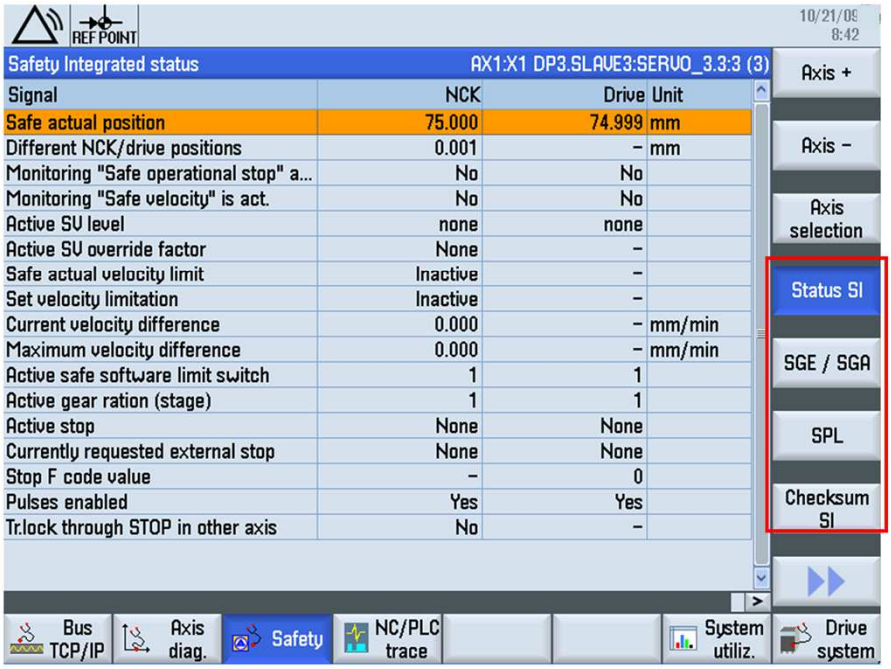
The information contained in the "Service drive" display is used to display and check the following information

- Enable and control signals
  (e.g. pulse enable PLC, controller enable NC)
- Status signals
  (e.g. enable pulses, drive ready)
- Special operating modes
  (e.g. setup mode, parking axis, axis operation)
- Parameter set drive
- Temperature alarms
- Drive state
- Current run-up phase
- Group error message (message ZK1)
- Drive status messages
  (e.g. threshold torque not reached, actual speed = set speed)
- Some characteristics such as
  - DC link voltage
  - Set/actual speed
  - Actual current value
  - Motor temperature
- Control mode, e.g. closed-loop speed control

SK: Drive +/-

The service values of the next (+) and the previous (-) drive are displayed.

Safety



 SITRAIN © Siemens AG 2017

Function

The service screens that can be selected here, e.g.:

- Status Si
- SGE / SGA
- SPL

...

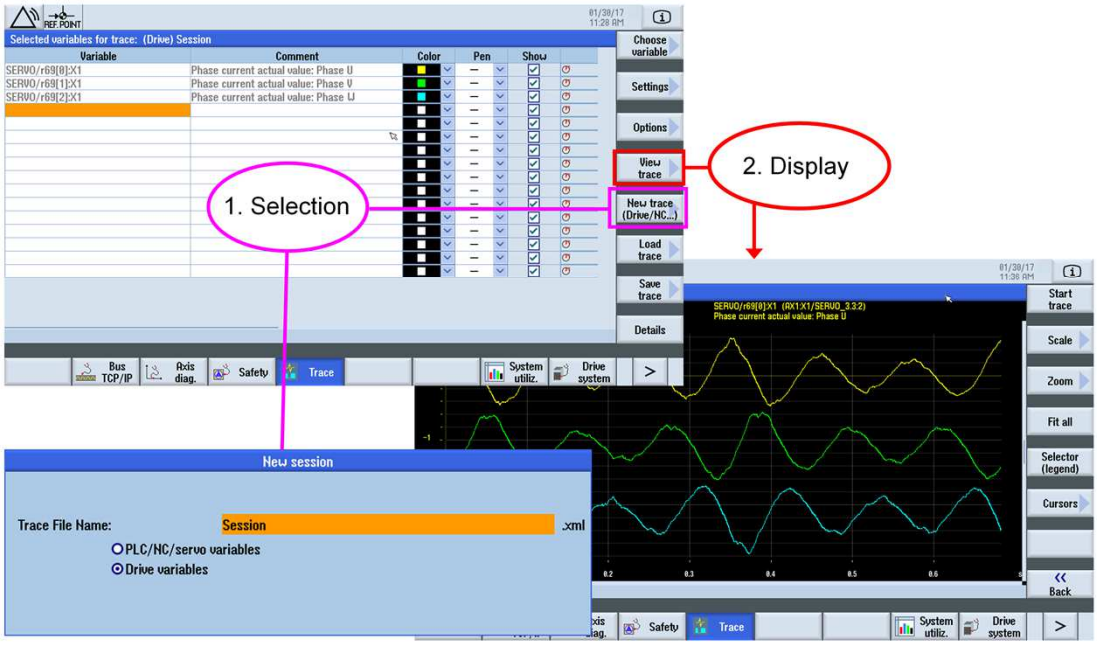
permit Safety Integrated diagnostics.

Example

According to the screen above, the following SI interlocks are active for the X1 axis:

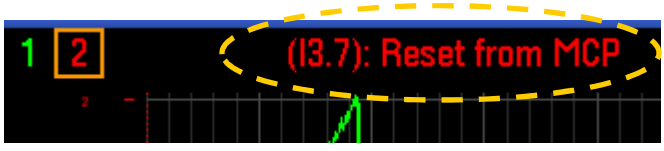
- "Safe operating stop" (travel inhibit, because e.g. the door is open)
- "Stop A/B" (pulse suppression, because e.g. EMERGENCY STOP pressed)

Trace

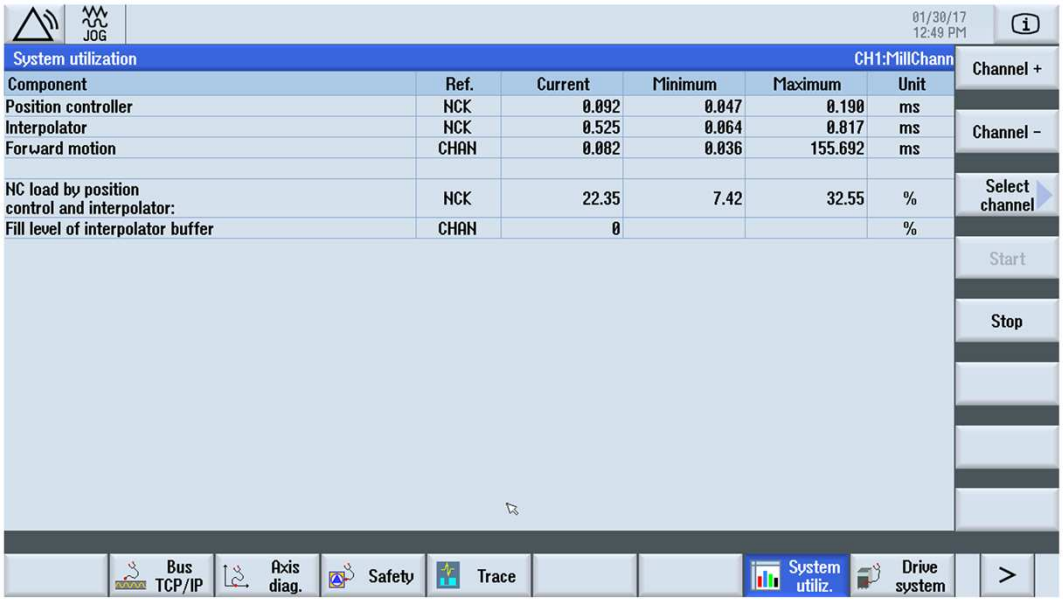


 SITRAIN © Siemens AG 2017

Function	This function allows a time diagram to be recorded for NC / PLC variables or drive variables.
Selection	Selection is performed using the **SK: New trace (drive/NC)**
Example	In the example, the phase currents U / V / W of the C-axis (drive parameters r69[0], r69[1], r69[2]) were recorded.
SK: Choose variable	With this, NC / PLC or drive variables (depending on selection) can be selected from lists and inserted.
SK: Trigger	The trigger and the stop condition can be set
SK: Settings	Recording cycle can be set as a multiple of the IPO or servo cycle (NC/PLC variable) or drive cycle (drive variable). The resulting recording duration is by default: 10 sec. for a NC/PLC trace (cycle time: 2 ms servo, 8 ms IPO, 40 kB memory) 3 sec. for a drive trace (cycle time 125 µs, 96 kB memory) The colors for grid lines, etc. can be selected.
SK: View trace	Here, the trace is displayed. It is started there with **SK: Start trace**
SK: Load trace	A trace that has been saved is loaded onto the screen.
SK: Save trace	A trace is saved.
Text display	The signal designation in the trace changes depending on the trace number that has been selected!
Example:	Trace 2 has been selected -> Variable name = "(I3.7): Reset from MCP":



System utilization



 SITRAIN © Siemens AG 2017

Function

The instantaneous system utilization is displayed.

The runtimes of the following are used as basis:

- Position controller
- Interpolator
- Preprocessing

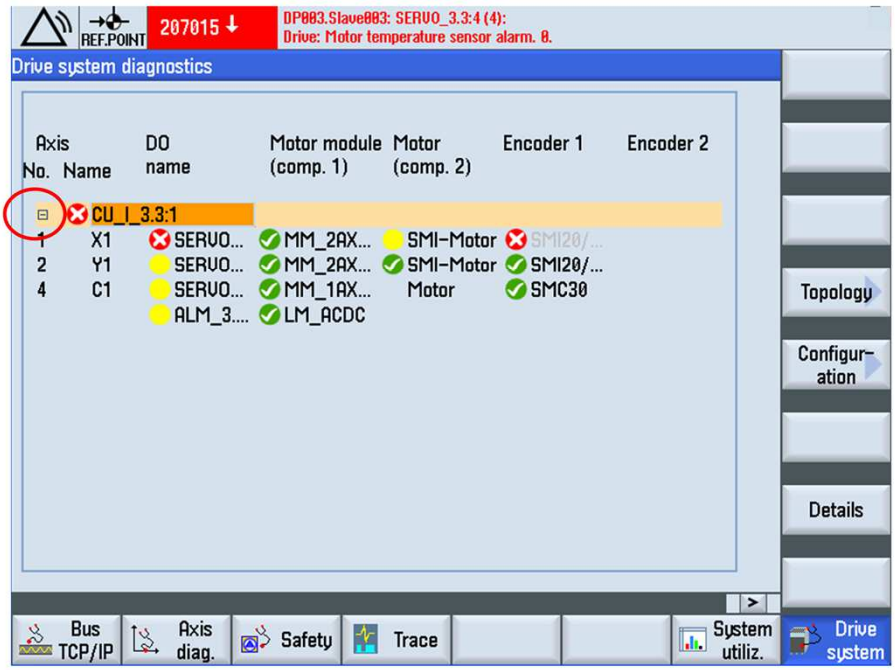
and the

- Time required for synchronized actions.

The CPU utilization is displayed in %.

CPU time share NC

The CPU time share of the NC+HMI processor for NC tasks is set in **MD 10185 NCK_PCOS_TIME_RATIO** default: 90 [%]
This value can be increased to improve NC performance.
However, this does slow down operation, and screens take longer to refresh.



 SITRAIN © Siemens AG 2017

Function

Information about the drives can be found in the Diagnostics operating area > Drive states.

All drive objects on the bus are listed if the list is opened by **pressing 'ENTER'** or **clicking '+'** with the mouse.

SK: Configuration

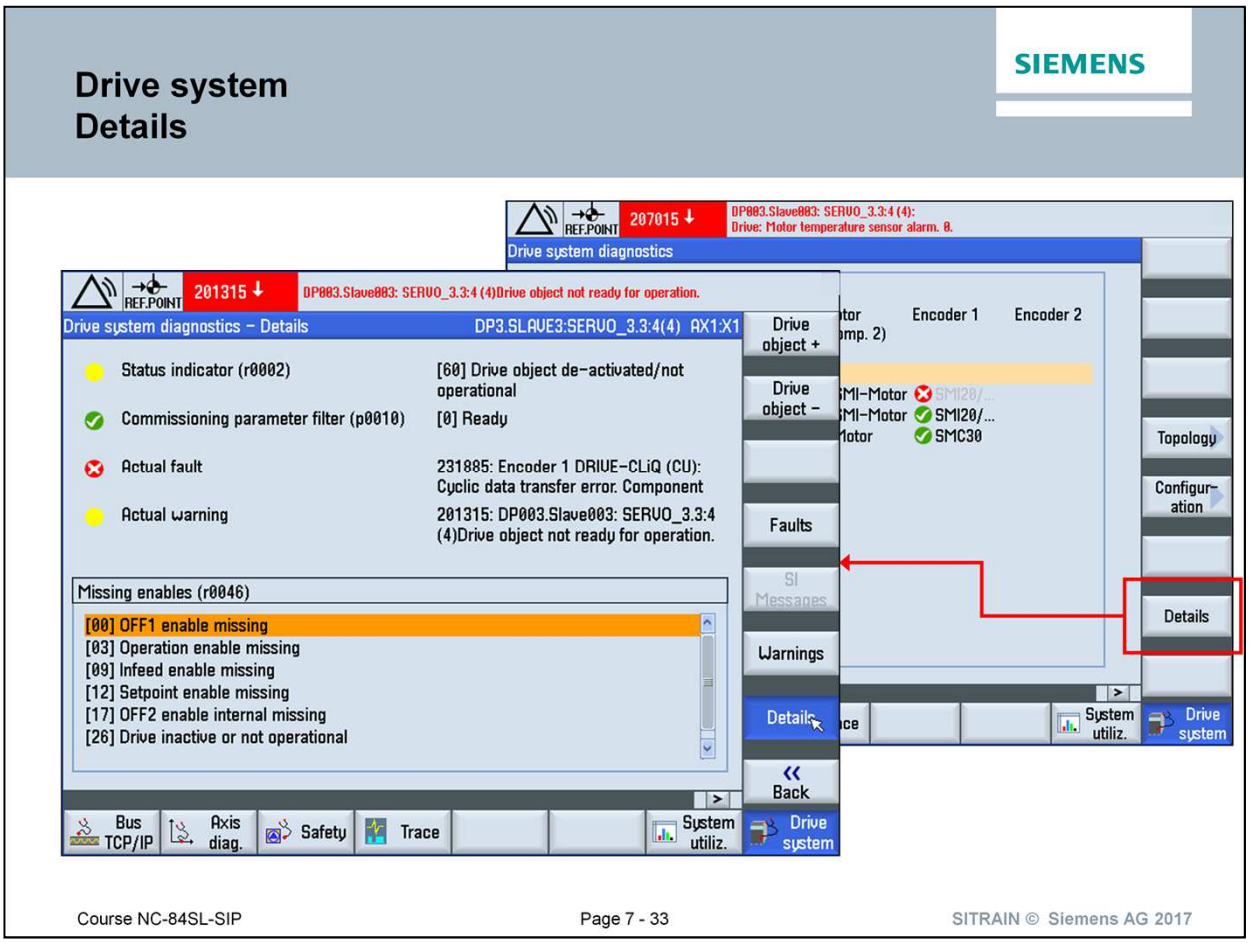
List of drive components, e.g. DRIVE-CLiQ components

SK: Topology

List of drive connections, e.g. DRIVE-CLiQ connections

SK: Details

More detailed information on the selected drive object



SK: Details

If an axis is in a fault condition, this is appropriately color-coded.

The following plain text displays simplify diagnostics:

r2 Operating display

p46 Missing enables

p10 Setup parameter filter

Current faults and alarms are also visible.

Drive system
Softkeys ‘Faults’ / ‘SI Messages’ / ‘Warnings’



 SITRAIN © Siemens AG 2017

Function

Here, drive alarms can be accessed as additional information, divided into the following categories 'Fault', 'SI message' and 'Warning' - without exiting the 'Drive system' menu.

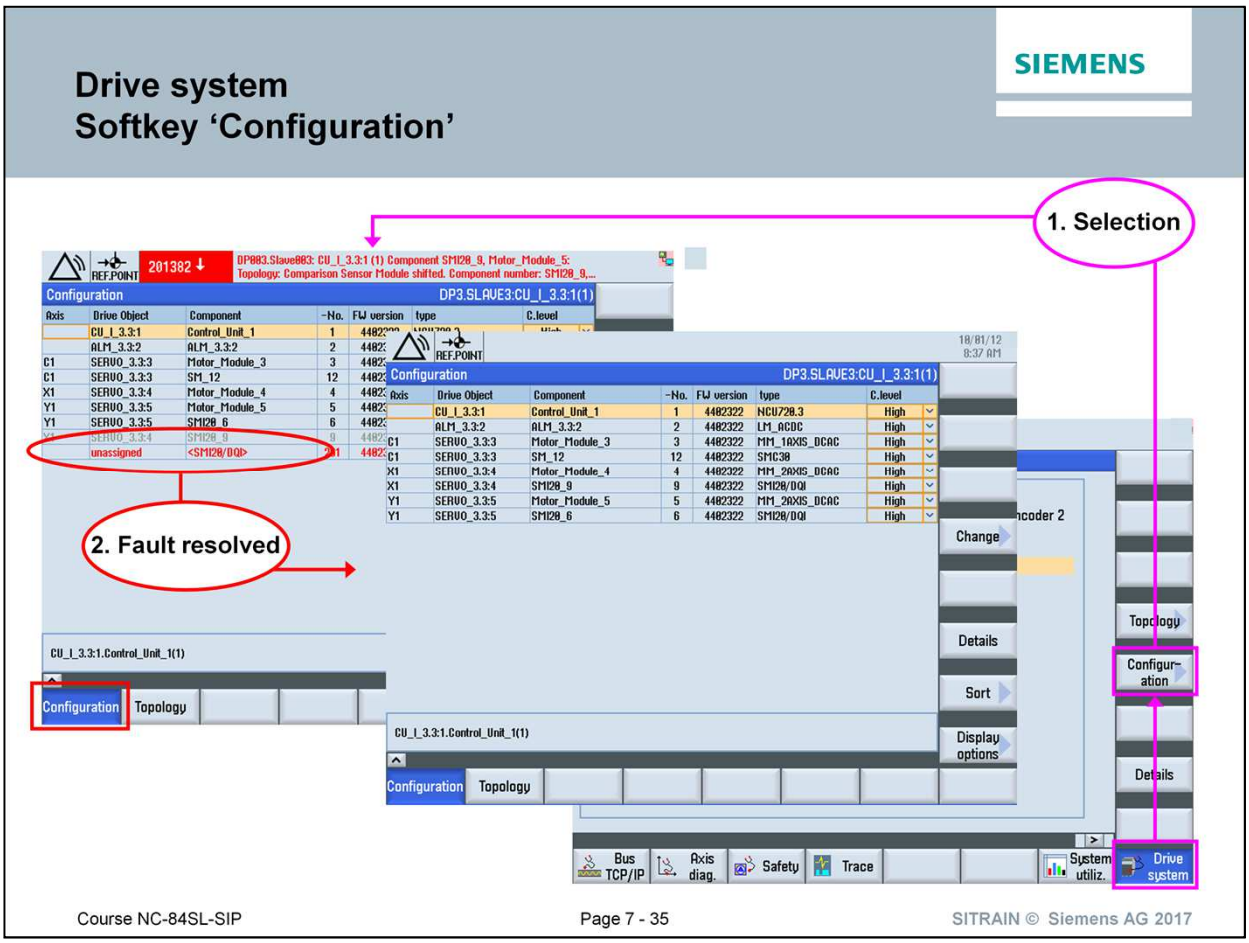
This is achieved using the following softkeys:

- **Faults**
- **SI Messages**
- **Warnings**

History

The history can also be called up under

1. Acknowledged fault
   …
8. Acknowledged fault



Function

The configured or found components are listed.
The target-actual comparison is standard (see the display options).

Components with faults are color-coded.

Gray

Gray means that a component is included in the target topology but missing from the actual topology.

Red

A component displayed in red has been found in the actual topology but does not appear in the target topology.

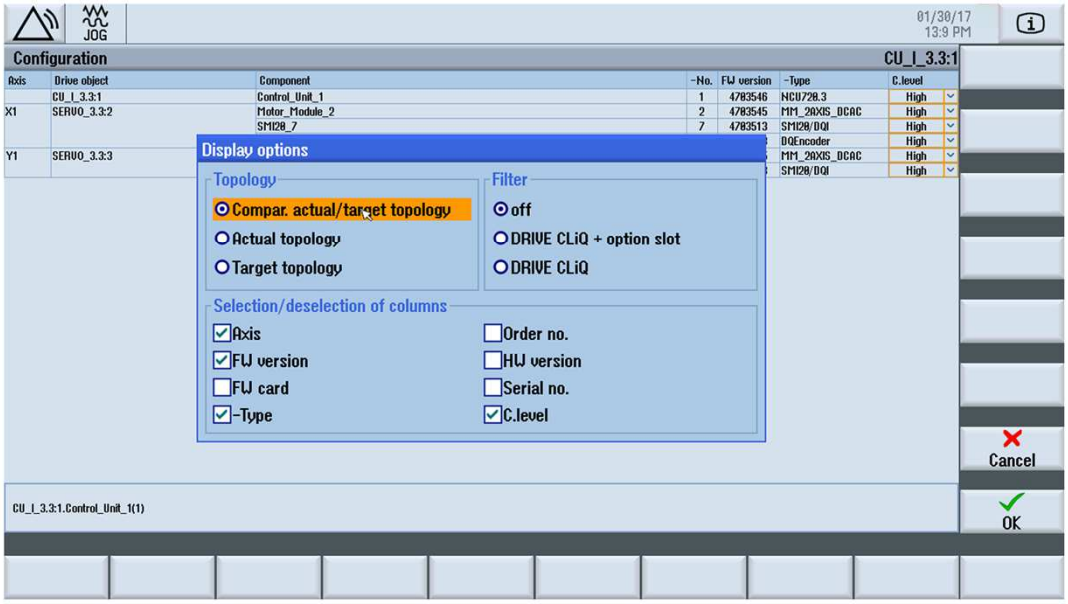
Example

The diagnostics on the screen form shown above, therefore, indicate that component 8 (SMI20) is missing in the actual topology.

It is possible that the DRIVE-CLiQ connector of the encoder is not plugged in.

The screen on the top right indicates the state after the fault has been resolved.

Drive system
Display options in the configuration screen form



 SITRAIN © Siemens AG 2017

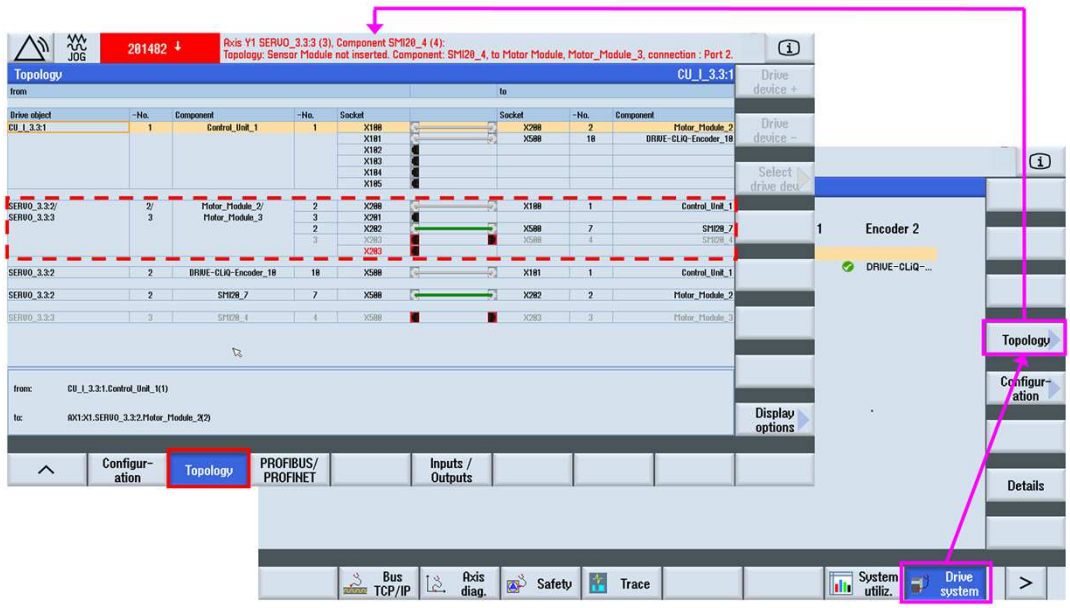


The Display options softkey opens a dialog box where the columns in the Configuration screen form can be selected.

The standard selection can be seen at the top of the screen, i.e. in particular:

- Compar. target/actual topology
- Display, DRIVE-CLiQ + option slot

Drive system
Softkey 'Topology'



 SITRAIN © Siemens AG 2017

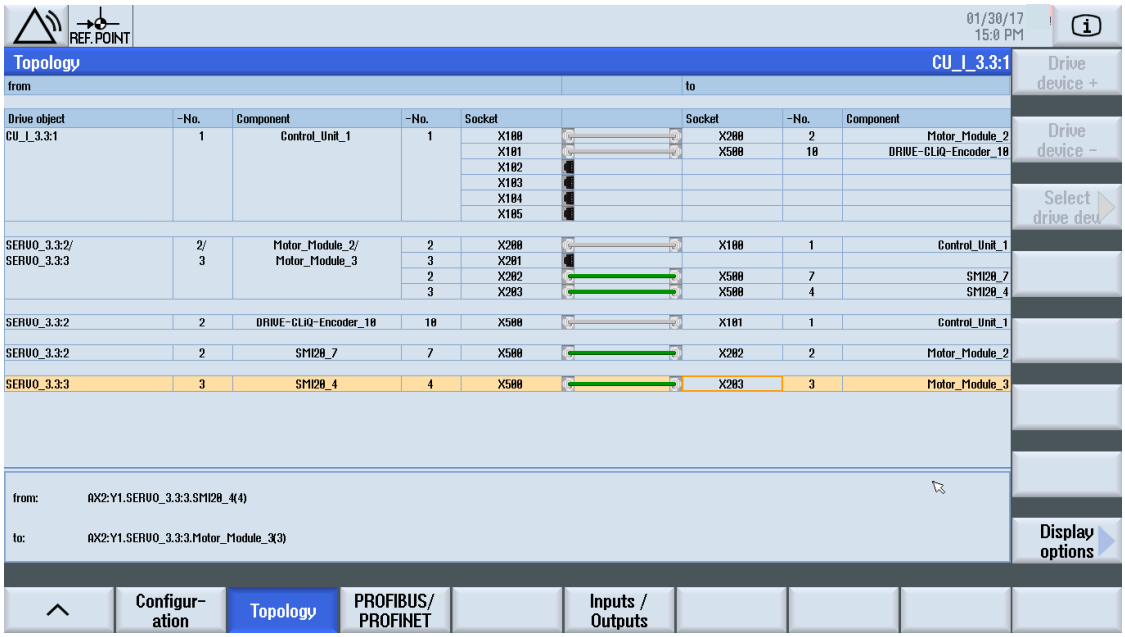
Function

The error display in the Topology screen form shows deviations between topologies in color.

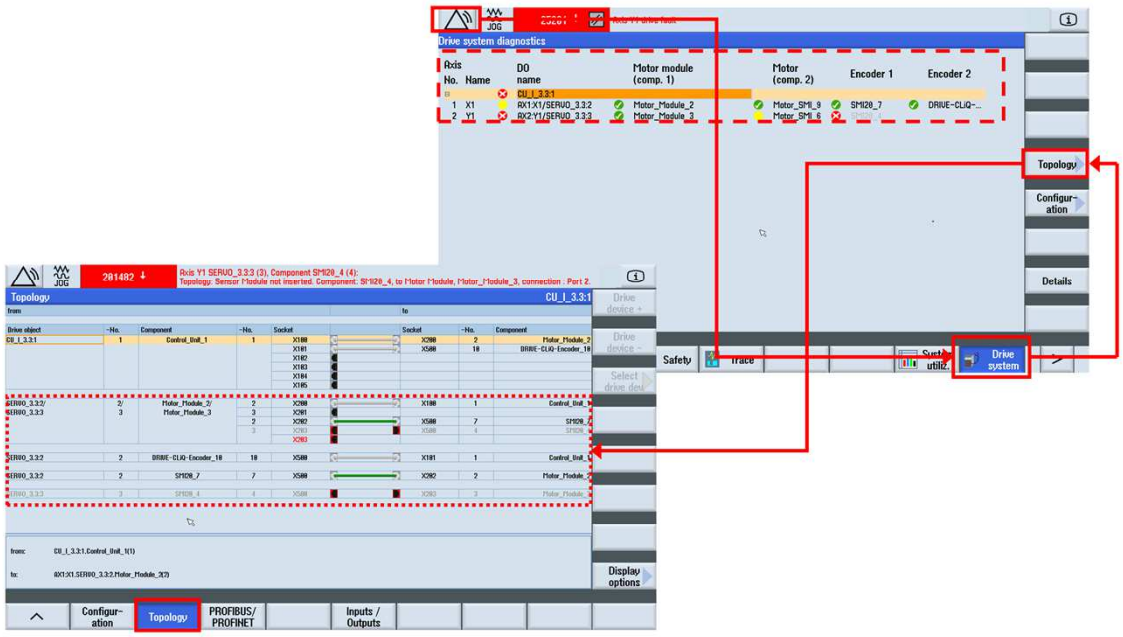
In the screen above, you can see that no component is expected at connection X201 on the Double Motor Module (gray), but an SMI20 has been found there (red).

On the other hand, at connection X202, an SMI20 is expected (gray), but has not been found (red)

After the error has been resolved (DRIVE-CLiQ plugged in incorrectly) this screen form no longer has any color-coded text and looks like this:



Diagnostics exercise: Topology



 SITRAIN © Siemens AG 2017

Objective

To get to know the topology view of the drive diagnostics for diagnosing DRIVE-CLiQ errors.

Steps

1. A colleague of yours makes a mistake with DRIVE-CLiQ (e.g. disconnects Motor Module X/Y...)
2. You open the Topology screen. You find this as follows: **> Operating Area: 'Diagnostics' > SK: 'Drive system' > SK: 'Topology'**
3. Attempt to evaluate the error using this screen only, without looking at the DRIVE-CLiQ wiring.

Color coding

The colors used to show the target-actual comparison have the following meaning:

Black OK	(target = actual)
Gray	Error (only in the target configuration)
Red	Error (only in the actual configuration)

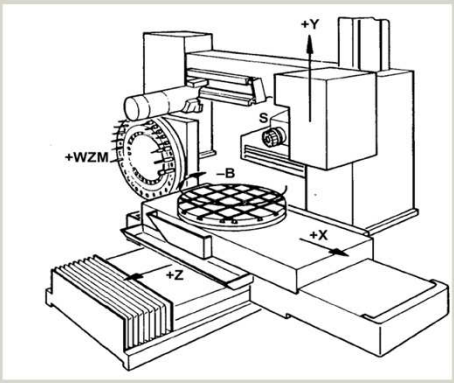
Example

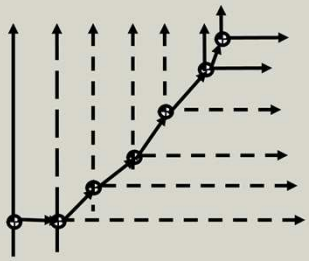
SERVO_3.3:4/	4/	Motor_Module_4/	4	X200	---	X201	3	Motor_Module_3
SERVO_3.3:5	5	Motor_Module_5	5	X201				
			4	X202	---	X500	9	SMI20_9
				X202				
			5	X203	---	X500	6	SMI20_6

Gray Error (connection to the SMI is missing in the actual configuration)

Red Error (empty connection missing in the target configuration)

SIEMENS





Chapter 8.1

Basic commissioning Kinematics

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Commissioning sequence 3
Coordinate systems 4
Frames 5
Axis types 6
Exercises: Commissioning kinematics 9
Exercise: 10
Exercise: Factory settings 11
Exercise: Axis configuration 12
Example: Configuration with 2nd channel 13
Machine data list 14
Machine data list 23

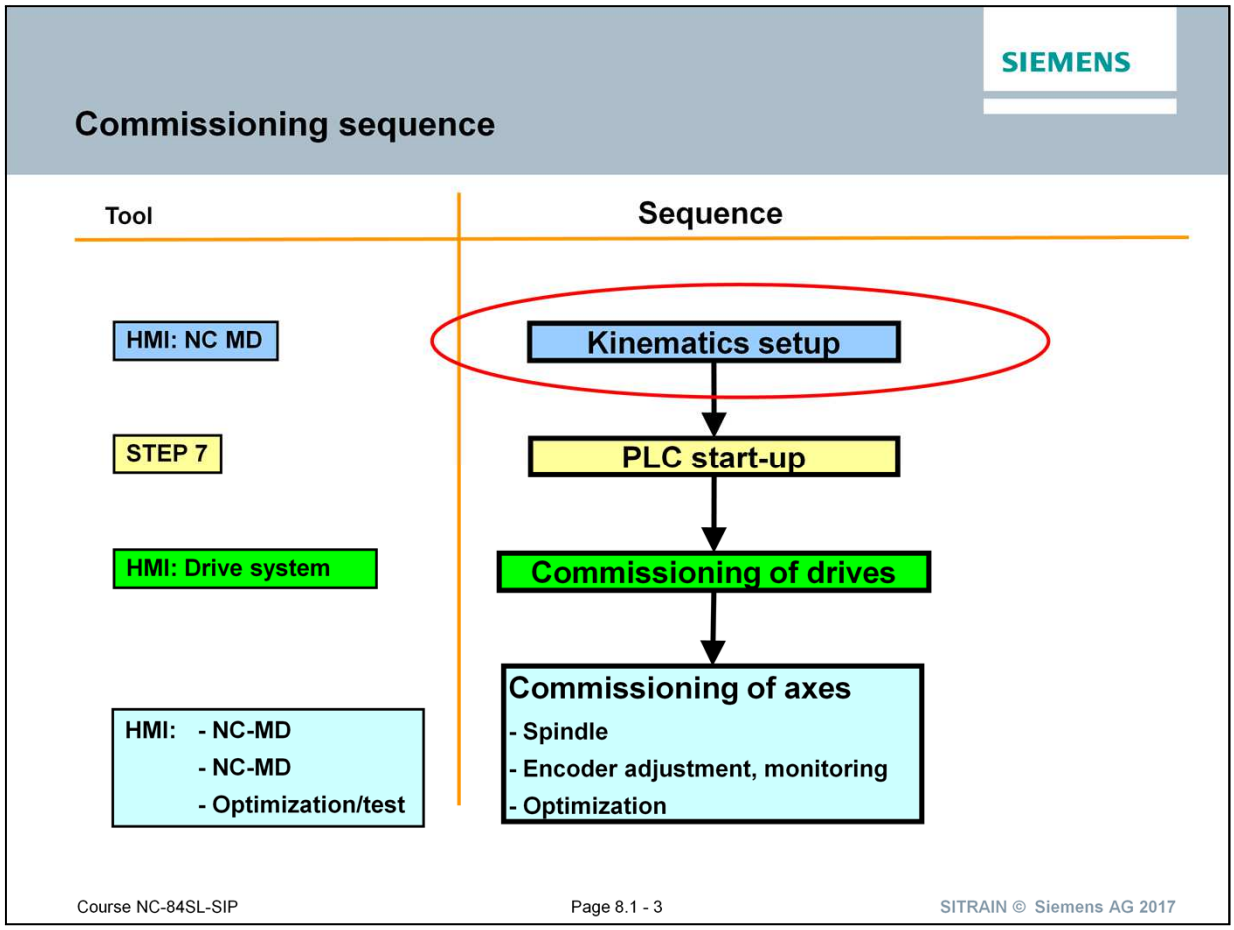
Learning objectives

- You will be familiar with coordinate systems and axis types
- You will be able to configure the axes



Course NC-84SL-SIP Page 8.1 - 2 SITRAIN © Siemens AG 2017

Notes

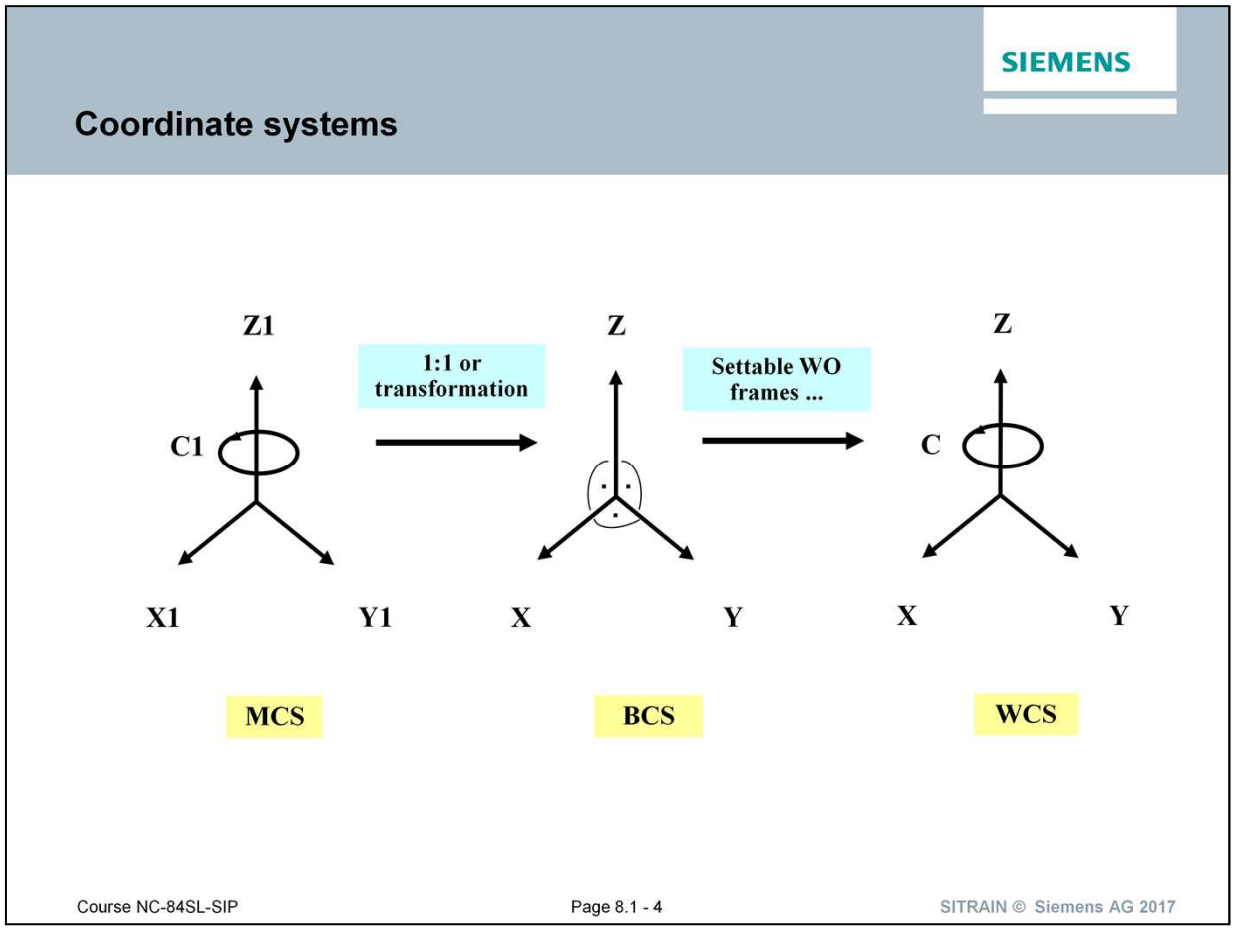


Commissioning part:

Kinematics

Start by identifying the axes on the machine and the axis groupings that are to be traversed with your own NC programs.

On the basis of this information, the axes are assigned to corresponding NC channels.
Auxiliary axes can also be traversed independently by the PLC, e.g. axes of a tool changer.



According to DIN 66217, clockwise, rectangular (Cartesian) coordinate systems are used in machine tools.

MCS The machine coordinate system (MCS) is formed by the machine axes. The machine axes can be perpendicular to one another, arranged in a Cartesian system, or arranged in any other way. The machine axes can be linear or rotary axes.

BCS The basic coordinate system BCS consists of 3 Cartesian axes (geometry axes) as well as other non-geometric axes (special axes). BCS and MCS coincide whenever the BCS can be mapped to the MCS without kinematic transformation (e.g. TRANSMIT/interfacial transformation, 5-axis transformation and max. 3 machine axes).

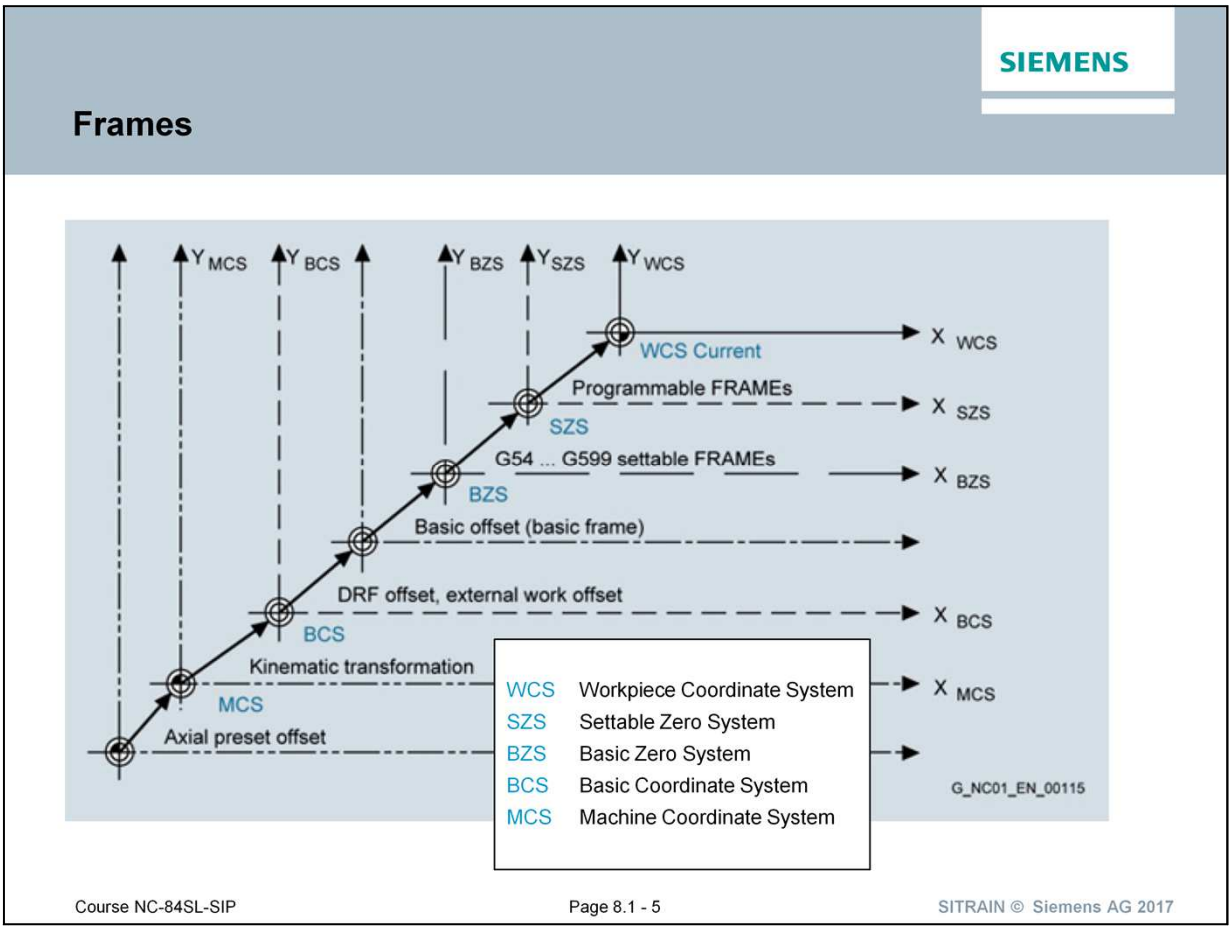
BZS Basic zero system BZS, DRF offsets, external work offsets and basic frames map the BCS on the BZS.

SZS Settable zero system SZS An activated settable work offset G54 to G599 transfers the BZS to the SZS.

WCS Workpiece coordinate system WCS. The programmable frame determines the WCS, which is the basis for programming. Work offsets are therefore used to transform the machine zero point into the workpiece zero point in order to simplify programming.

Conversion of coordinate systems

You can change between the coordinate systems stated above by conversion. The terminology used with SINUMERIK 840D sl makes a distinction between conversion between two rectangular (Cartesian) coordinate systems and conversion into a non-Cartesian coordinate system. From a mathematical point of view, the operation is always a coordinate transformation.

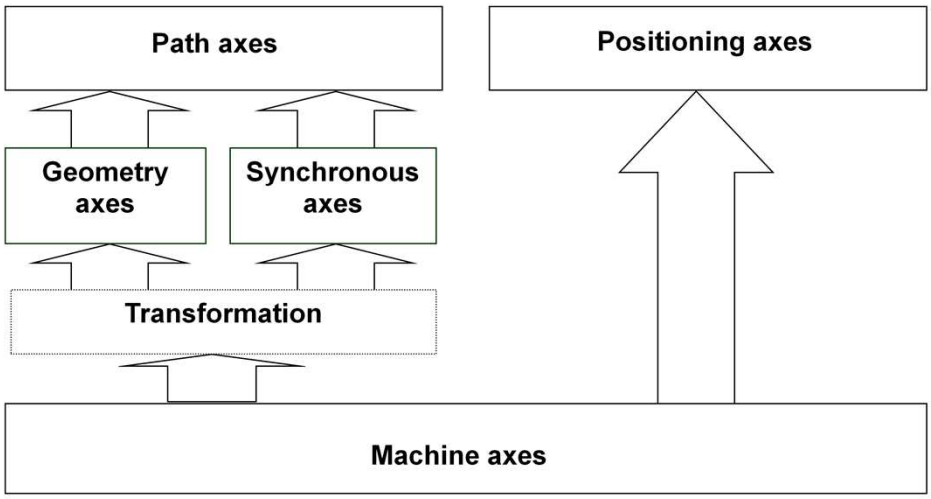


Work offsets

The following work offsets are possible:

- The basic area contains further subsections – such as global basic work offset, channel-specific basic work offset and configuration-dependent work offsets, such as rotary table reference or basic reference.
- Settable work offsets SZS: It is possible to enter up to 100 work offsets (G54 to G57, G505 to G599), offset coordinates, angles and scaling factors in order to call zero points program-wide for various fixtures or clamping operations, for example. The work offsets can be suppressed block by block.
- Programmable work offsets: Work offsets can be programmed with TRANS (substitution function, basis G54 to G599) or ATRANS (additive function). This allows you, for example, to work with different work offsets for repetitive machining operations at different positions on the workpiece. With G58/G59, previously programmed work offsets can be replaced in each axis.
- External work offsets:
  Axis-related linear work offsets can also be activated via the PLC user software (function blocks) with assignment to the system variable $AA_ETRANS [axis].

Axis types for part programming

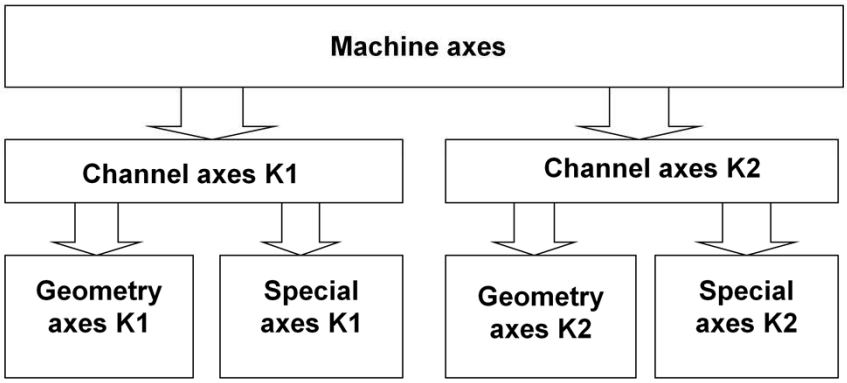


SITRAIN © Siemens AG 2017

Machine axes	Machine axes are the axes that actually exist on a machine tool.
Channel axes	In every channel, channel axes are defined for programming. They are a subset selected from the machine axes.
Geometry axes	The three geometry axes always form a fictitious rectangular coordinate system, the basic coordinate system (BCS). By using FRAMES (offset, rotation, scaling, mirroring), it is possible to image geometry axes of the workpiece coordinate system (WCS) on the BCS.
Special axes	Unlike geometry axes, no geometrical relationship is defined between special axes.
Path axes	Path axes are interpolated together (all the path axes of a channel have a common path interpolator). All the path axes of one channel have the same acceleration phase, constant travel phase, and deceleration phase. - Programming: G.. X.. Y.. F..
Positioning axes	Positioning axes are interpolated separately (each positioning axis has its own axis interpolator). Each positioning axis has its own position (POS), its own feedrate (FA), and an acceleration characteristic. - Programming: POS[X]=.. FA[X]=..
Synchronized axes	Synchronized axes are interpolated together with path axes (all path axes and synchronized axes of one channel have a common path interpolator). All path axes and all synchronized axes of a channel have the same acceleration phase, constant travel phase, and deceleration phase.

Axis types for axis configuration

SIEMENS



SITRAIN © Siemens AG 2017

Axis configuration

The assignment between the geometry axes, special axes, channel axes, and machine axes is made and the names of each axis type are defined in the machine data

MD 10000 AXCONF_MACHAX_NAME_TAB,

MD 20050 AXCONF_GEOAX_ASIGN_TAB,

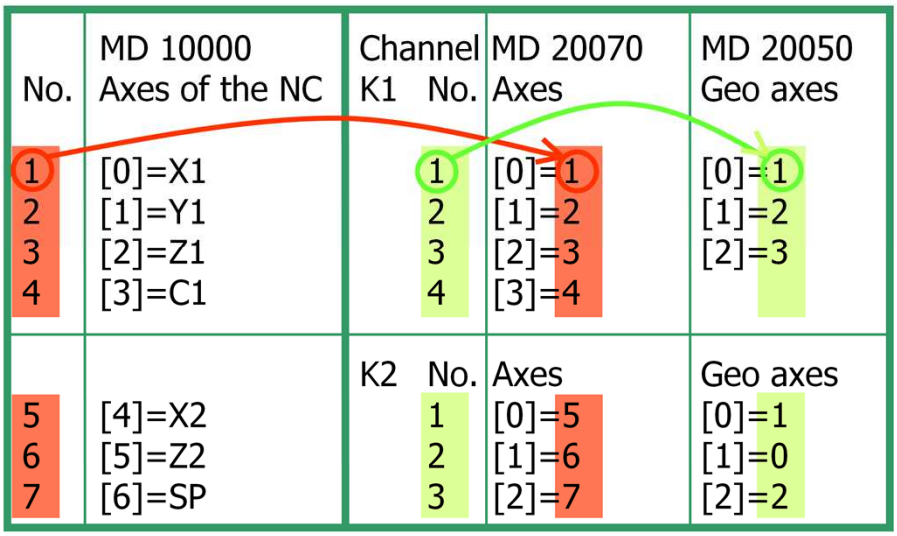
MD 20060 AXCONF_GEOAX_NAME_TAB,

MD 20070 AXCONF_MACHAX_USED,

MD 20080 AXCONF_CHANAX_NAME_TAB,

Axis types for axis configuration

SIEMENS



 SITRAIN © Siemens AG 2017

Machine axis name

MD 10000: AXCONF_MACHAX_NAME_TAB[m]
A name must be defined for each machine axis. This name is used for specific NC functions, e.g.:
- G74 Reference point approach from the part program
- G75 Fixed-point approach (MD 30600: Fixed-point positions)
- Measurement
- Display of axis-specific alarms, etc.

Geometry axis name

MD 20060: AXCONF_GEOAX_NAME_TAB[g]
Axis name in the part program, consisting of up to 16 characters. The geometry axes have a fixed reference to the workpiece coordinate system. The tool offset applies to the geometry axes.

Channel axis name

MD 20080: AXCONF_CHANAX_NAME TAB[k]
Axis name displayed when the WCS is selected. The first 3 channel axes are usually the 3 geometry axes. If there are additional axes in the channel (special axes), their names are defined in the part program via this MD.

Geometry axis assignment

MD 20050: AXCONF_GEOAX_ASSIGN_TAB[g]
Defines the order of the geometry axes in the channel. Enter the number of the channel axis that corresponds to this geometry axis. With turning machines, only 2 geometry axes are configured.
E.g.: Turning machine: MD 20050[0]=1 MD 20060[0]=X
MD 20050[1]=0 MD 20060[1]=
MD 20050[2]=2 MD 20060[2]=Z

Machine axis assignment

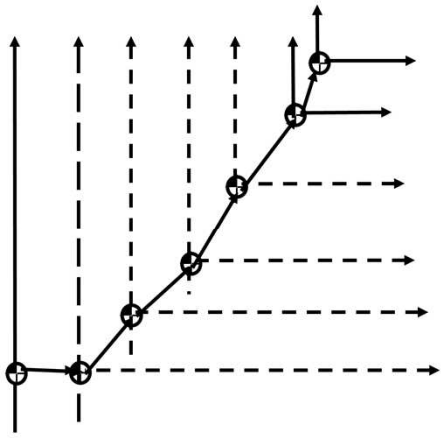
MD 20070: AXCONF_MACHAX_USED[k]
Defines the assignment of the channel axes to the machine axes. This MD assigns the machine axes to the individual channels. Enter the number of the machine axis that corresponds to this channel axis.

Indices

Index [g] is the geometry axis index (value range: 0 - 2).
Index [k] is the channel axis index (value range: 0 - 11).

Exercises: Commissioning kinematics



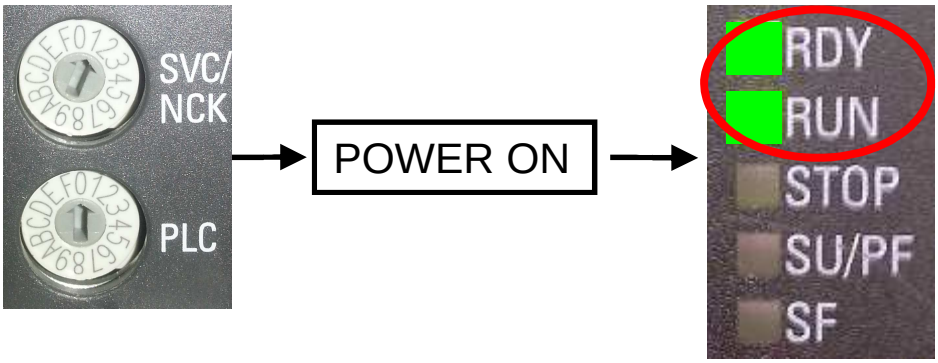


Course NC-84SL-SIP Page 8.1 - 9 SITRAIN © Siemens AG 2017

Notes

Memory reset of the NC

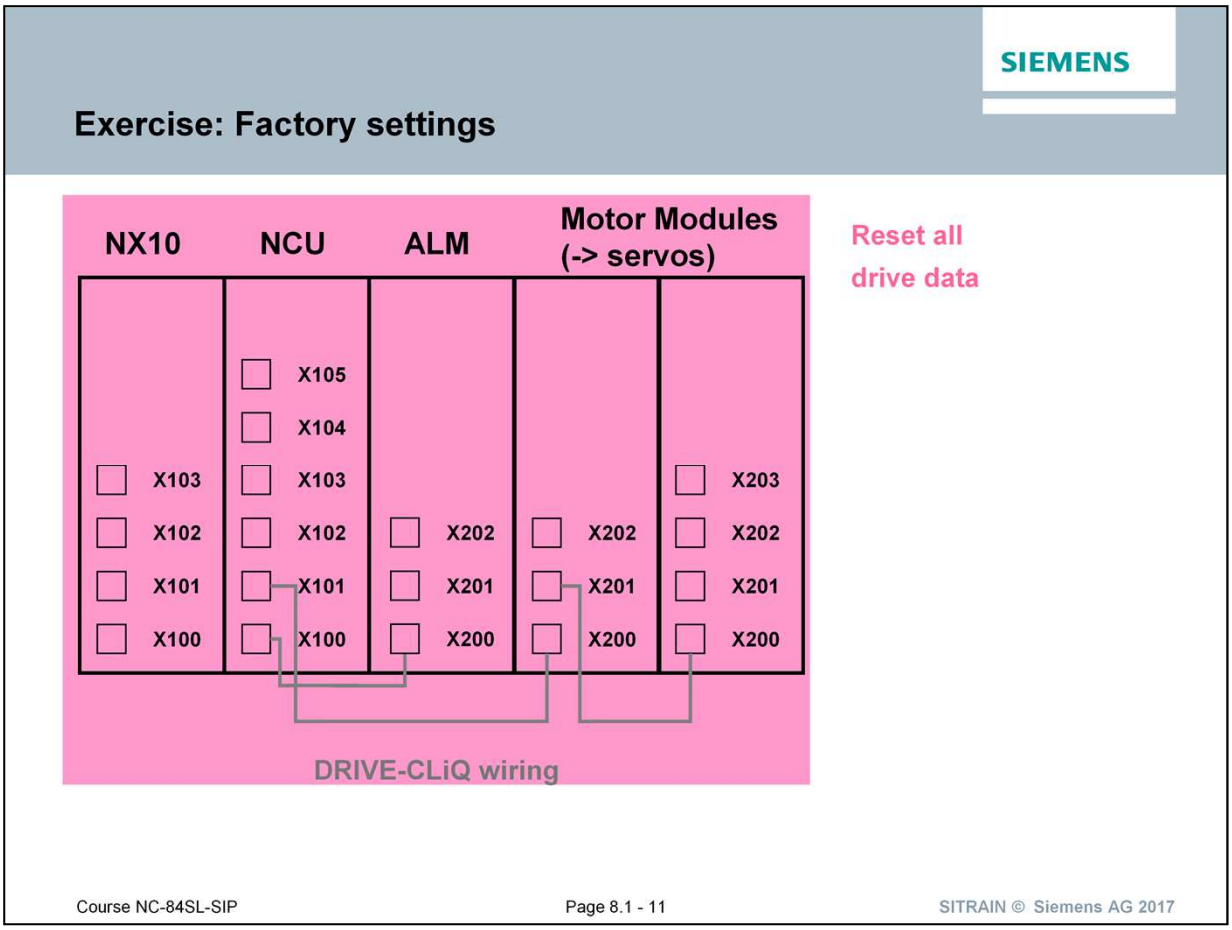
1. Switch off the control system.
2. Request NCK memory reset
   - Set the SVC/NCK rotary switch to position "1."
   - POWER ON (switch control off and back on again).



3. In the end, both switches will be set to "0."



The NC memory reset is now complete.



Why factory settings?

The drives are only reset in this case to avoid annoying error messages. The factory settings will only be needed in the next chapter.

Objective

To set the control to a defined initial state with the factory settings.
The factory settings must be restored before the device can be configured.

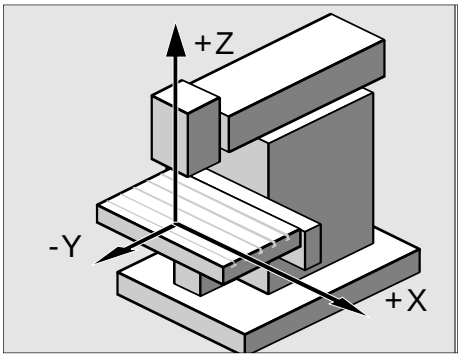
1. Select the factory settings

Select:
OA: Setup -> Softkey: Drive system ->
Softkey: Factory settings -> Softkey: Drive system/drive unit

2. Load factory settings

Confirm the prompt with
-> Softkey: OK
Wait for the prompt for automatic device configuration.
This will be described in the following exercise

Exercise: Axis configuration



Objective

Configure the kinematics of a milling machine with axes X, Y, Z, and C. Try to map the structure as presented in the theory section and implement it accordingly.

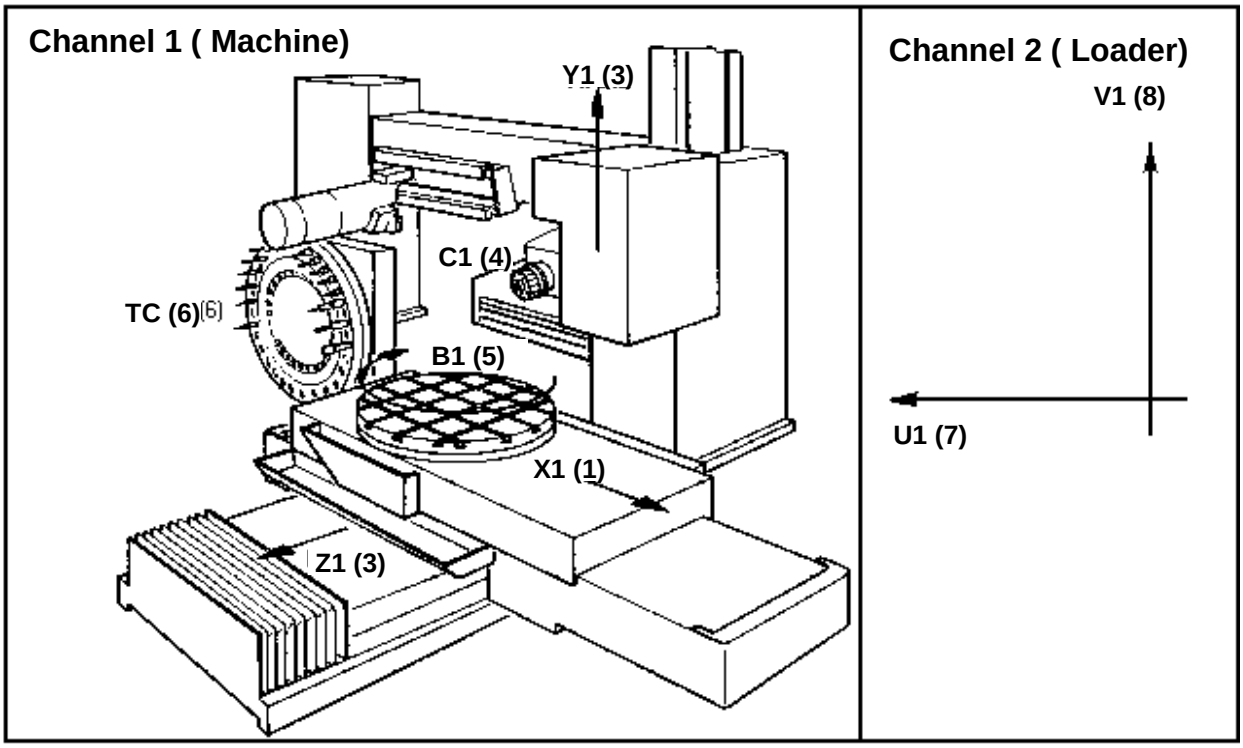
The corresponding machine data has been provided below for assistance. Next, enter the NC data. You should be able to use this data to traverse the simulated axes.

Machine axes	MD 10000[0]=X1	Machine axis name
	MD 10000[1]=Y1	
	MD 10000[2]=Z1	
	MD 10000[3]=C1	
Channel axes	MD 20070[0]=1	Assignment machine axis to channel
	MD 20070[1]=2	
	MD 20070[2]=3	
	MD 20070[3]=4	
	MD 20080[0]=X	Channel axis name
	MD 20080[1]=Y	
	MD 20080[2]=Z	
	MD 20080[3]=C	
Geometry axes	MD 20050[0]=1	Assignment channel axis - geometry axis
	MD 20050[1]=2	
	MD 20050[2]=3	
	MD 20060[0]=X	Geometry axis name
	MD 20060[1]=Y	
	MD 20060[2]=Z	

Alternatively, configure a turning machine with the axes X, Z, and C.

Machine axes	MD 10000[0]=X1	Machine axis name
	MD 10000[1]=Z1	
	MD 10000[2]=C1	
Channel axes	MD 20070[0]=1	Assignment machine axis to channel
	MD 20070[1]=2	
	MD 20070[2]=3	
	MD 20080[0]=X	Channel axis name
	MD 20080[1]=Z	
	MD 20080[2]=C	
Geometry axes	MD 20050[0]=1	Assignment channel axis - geometry axis
	MD 20050[1]=0	
	MD 20050[2]=2	
	MD 20060[0]=X	Geometry axis name
	MD 20060[1]=	
	MD 20060[2]=Z	

Example: Configuration with 2nd channel



							30310 30320
Linear axis	X1	1	X	X	1	0	0
Linear axis	Y1	2	Y	Y	2	0	0
Linear axis Sim.	Z1	3	Z	Z	3	0	0
Spindle	C1	4	C	-	-	1	1
Rotary axis Sim.	B1	5	B	-	-	0	1
Rotary axis Sim.	WZW	6	Q	-	-	0	1

Channel 2 (loader) comprising 2 axes

MD	10000	20070	20080	20060	20050	35000	30300 30310 30320
Linear axis Sim.	U1	7	U	X	1	0	0
Linear axis Sim.	V1	8	V	Y	2	0	0

Note

For this example, we must start with the following machine data assignments:

- Options (see softkey: Licenses):
  - 1 axis/spindle in addition **A00 -> 5**
  - 1 machining channel in addition **C10 -> 1** **-> Power On!**
- Configuration of 2nd channel in mode group 1:
  - MD 10010[0] = 1
  - **MD 10010[1] = 1** **-> Power On!**
- Alarm response CHAN_NOREADY instead of BAG_NOREADY:
  - MD 11412 = 1

In the NC/PLC interface, instead of signal DB11 DBX26.3 (mode group ready), the signal DB21, ... DBX36.5 (channel ready) is reset.

The alarm response remains limited to the channel in which the error is detected.

Machine data list milling

You should work through and configure the following machine data when you set up the milling machine.

MD	Identifier	Settings	Comment	Description
10260	$MN_CONVERT_SCALING_SYSTEM	1	Exact	Enable basic system conversion
10602	$MN_FRAME_GEOAX_CHANGE_MODE	1	Exact	Frames when changing geometry axes
10610	$MN_MIRROR_REF_AX	0	Exact	Reference axis for mirroring
10714	$MN_M_NO_FCT_EOP	32	Variable	M function for spindle active after reset
10722	$MN_AXCHANGE_MASK	Bit 2 = 1	Exact	Parameters for axis replacement behaviour
11410	$MN_SUPPRESS_ALARM_MASK	Bit 0 = 1 Bit 1 = 1 Bit 15 =1 Bit 20 =1	Variable	Bit 0: 15110: REORG not possible Bit 1: 10763 The path component of the block in the contour plane is zero Bit 15: 5000 Communication order cannot be executed Bit 20: 2900 Reboot is delayed
11450	$MN_SEARCH_RUN_MODE	Bit 0 = 1 Bit 1 = 1 Bit 2 = 1	Exact Exact Exact	Bit 0: Machining is stopped with the loading of the last action block after search run Bit 1: Automatic ASUP start after output of action blocks Bit 2: The output of the auxiliary functions in the action blocks is suppressed
11470	$MN_REPOS_MODE_MASK	Bit 3 = 1	Exact	Mask for support of special alarm outputs
17530	$MN_TOOL_DATA_CHANGE_COUNTER	1FH	Exact	Mark tool data change for HMI
18080	$MN_MM_TOOL_MANAGEMENT_MASK	Bit 0 = 1 Bit 1 = 1 Bit 3 = 1	Exact Exact Exact	Bit 0: Memory for TM-specific data is provided Bit 1: Memory for monitoring data (WZMO) is provided Bit 3: Memory to consider adjacent location is provided
19320	$ON_TECHNO_FUNCTION_MASK	Bit 4 = 1	Exact	Tool management with magazine data
20100	$MC_DIAMETER_AX_DEF	X	Variable	Geometry axis with transverse axis function
20110	$MC_RESET_MODE_MASK	Bit 0 = 1 Bit 6 = 1 Bit 14 =1 Bit 16 =0	Exact Exact Exact Variable	Bit 0 = 1 G codes acc. to MD20150 Bit 6 = 1 Current setting for active tool length offset is retained after reset/end-of-part-program Bit 14 = 1 The current setting of the basic frame is retained Bit 16 = 0 Initial setting for master spindle acc. to MD20090
20124	$MC_TOOL_MANAGEMENT_TOOLHOLDER	1	Exact	Tool holder number
20128	$MC_COLLECT_TOOL_CHANGE	0	Exact	No tool change commands after search run
20130	$MC_CUTTING_EDGE_RESET_VALUE	1	Exact	Tool and cutting edge remain activated
20150	$MC_G_CODE_RESET_VALUE	[5] = 2 [15] = 3 [21] = 2 [28] = 2 [48] = 3	Exact Variable Variable Exact Variable	Group 6: Plane selection 2 = 18 Group 16: Feedrate override 3 = CFIN Group 22: Tool offset type 2 = CUT2DF Group 29: Radius/diameter programming 2 = DIAMON Group 49: Point-to-point motion 3 = PTPG0
20152	$MC_G_CODE_RESET_MODE	[5] = 0 [7] = 1 [12] = 0 [14] = 1 [15] = 0 [21] = 0	Exact Exact Exact Exact Exact Variable	Group 6: Plane selection remains active after Reset Group 8: Zero point offset remains active after Reset Group 13: Work piece dimensioning inch/metric default after reset Group 15: Type of feed remains active after Reset Group 16: Feed correction remains active after Reset Group 22: Tool correction type default after reset
20202	$MC_WAB_MAXNUM_DUMMY_BLOCKS	10	Minimum	maximum number of blocks w/o traversing movement with SAR
20240	$MC_CUTCOM_MAXNUM_CHECK_BLOCKS	4	Minimum	Blocks for look-ahead contour calculation with TRC
20250	$MC_CUTCOM_MAXNUM_DUMMY_BLOCKS	5	Minimum	maximum number of blocks without traversing motion in TRC

MD	Identifier	Settings	Comment	Description
20310	$MC_TOOL_MANAGEMENT_MASK	Bit 0 = 1 Bit 1 = 1 Bit 3 = 1 Bit 14 =1 Bit 16 =1 Bit 23 =1	Exact Exact Exact Exact Exact Variable	Bit 0: Tool management active Bit 1: Tool monitoring function active Bit 3: Consider adjacent location active Bit 14: Automatic tool change during RESET and Start Bit 16: Programming T='Location number' Bit 23: On offset selection no synchronization with HL
20320	$MC_TOOL_TIME_MONITOR_MASK	Bit 0 = 1	Variable	Time monitoring for tool in tool holder 1
20360	$MC_TOOL_PARAMETER_DEF_ MASK	Bit 0 = 1	Variable	Definition of tool parameters
20734	$MC_EXTERN_FUNCTION_MASK	Bit 3 = 1	Variable	Errors in ISO scanner are not output, the block is transferred to the Siemens translator.
22550	$MC_TOOL_CHANGE_MODE	0	Exact	Tool change with T function
24006	$MC_CHSFRAME_RESET_MASK	Bit 0 = 1 Bit 5 = 0	Exact Exact	Active system frames after reset Bit 0 = 1 Actual value setting and scratching remains active Bit 5 = 0 System frame for cycles is deleted
24007	$MC_CHSFRAME_RESET_CLEAR_MASK	Bit 0 = 0 Bit 5 = 1	Exact Exact	Deletion of system frames after reset Bit 0 = 0 Actual value setting and scratching not delete Bit 5 = 1 System frame for cycles delete
24008	$MC_CHSFRAME_POWERON_MASK	Bit 0 = 0 Bit 5 = 0	Exact Exact	Reset channel system frames after power on Bit 0 = 0 Actual value setting and scratching not reset Bit 5 = 0 System frame for cycles not reset
24030	$MC_FRAME_ACS_SET	1	Exact	Adjustment of SZS coordinate system
24040	$MC_FRAME_ADAPT_MODE	7H	Exact	Adaptation of active frames
27860	$MC_PROCESSTIMER_MODE	Bit 0 = 1 Bit 1 = 1 Bit 4 = 1 Bit 5 = 1 Bit 6 = 1 Bit 8 = 1	Variable	Bit 0: Measurement of total operating time is active for all part programs Bit 1: Measurement of current program runtime is active Bit 4: Measurement also with active dry run feed Bit 5: Measurement also with program test Bit 6: $AC_CYCLE_TIME is not deleted on start by ASUP and PROG_EVENT Bit 8: AC_CYCLE_TIME will be deleted with GOTOS
27880	$MC_PART_COUNTER	Bit 0 = 1 Bit 8 = 1 Bit 11 =1	Variable	Bit 0: Counter $AC_REQUIRED_PARTS is activated Bit 8: Counter $AC_ACTUAL_PARTS is active Bit 11: Increment $AC_ACTUAL_PARTS with GOTOS
28000	$MC_MM_REORG_LOG_FILE_MEM	75	Minimum	Memory space for REORG (DRAM)
28010	$MC_MM_NUM_REORG_LUD_MODULES	20	Minimum	Number of blocks for local user variables in REORG (DRAM)
28082	$MC_MM_SYSTEM_FRAME_MASK	Bit 0 = 1 Bit 5 = 1 Bit 6 = 1	Exact Exact Exact	Setting up system frames (SRAM) Bit 0 = 1 System frame for actual value and scratching Bit 5 = 1 System frame for cycles Bit 6 = 1 System frame for transformations
28083	$MC_MM_SYSTEM_DATAFRAME_MASK	Bit 0 = 1 Bit 5 = 0	Exact Exact	Setting up data management system frames (SRAM) Bit 0 = 1 System frame for actual value and scratching Bit 5 = 0 System frame for cycles
28450	$MC_MM_TOOL_DATA_CHG_BUFF_SIZE	400	Exact	Buffer for tool data changes (DRAM)
35040	$MA_SPIND_ACTIV_AFTER_RESET	2	Exact	Own spindle RESET
42440	$SC_FRAME_OFFSET_INCR_PROG	0	Exact	Zero offsets in frames
42442	$SC_TOOL_OFFSET_INCR_PROG	0	Exact	Tool length offsets
42528	$SC_CUTCOM_DECEL_LIMIT	1	Variable	Feed lowering on circles with tool radius compensation
42940	$SC_TOOL_LENGTH_CONST	18	Exact	Change of tool length components with change of active plane
42950	$SC_TOOL_LENGTH_TYPE	2	Exact	Assignment of tool length compensation independent of tool type

MD	Identifier	Settings	Comment	Description
51024	$MNS_BLOCK_SEARCH_MODE_MASK_JS	Bit 0 = 1		Bit mask for available search modes (ShopMill/ShopTurn) Bit 0: Block search with calculation without approach Bit 1: Block search with calculation with approach Bit 3: Skip EXTCALL programs Bit 5: Block search with test run
51028	$MNS_BLOCK_SEARCH_MODE_MASK	Bit 0 = 1 Bit 1 = 1 Bit 4 = 1 Bit 5 = 1		Bit mask for available block search modes Bit 0: Block search with calculation without approach Bit 1: Block search with calculation with approach Bit 3: Skip EXTCALL programs Bit 4: Block search without calculation Bit 5: Block search with test run
51038	$MNS_SET_ACT_VALUE	1		Set actual value selection 0 = Set actual value is not offered. 1 = If a user frame (settable work offset e.g. G54) is active, it will be used. In G500 Set actual values is not offered (system frame is no longer used).
51039	$MNS_PROGRAM_CONTROL_MODE_MASK	Bit 0 = 1		Options for machine - program influence: Bit 0: Program test function available
51040	$MNS_SWITCH_TO_MACHINE_MASK	0H		Automatic operating area switchover to machine Bit 0: When selecting a program in the program manager, the operating area does not automatically switch over to Machine. Bit 1: When switching the type of operation via the MCP, the operating area is not automatically switched over to Machine. Bit 2: Do not switch automatically in Programs during program selection. Bit 3: No auto. Start of block search on selection / execution under Programs.
51226	$MNS_FUNCTION_MASK_SIM	0H		Function mask Simulation Bit 0: No automatic start on simulation selection Bit 1: Deactivate simulation Bit 4: Ignore frame components in X and Y (compatibility) Bit 5: Turning tool can rotate in the tool spindle Bit 6: Enable handwheel as simulation override Bit 7: Interpretation of handwheel values as absolute values Bit 10: Hide zero point symbol
51228	$MNS_FUNCTION_MASK_TECH	0H		Function mask Cross-technology Bit 0: G code programming without multi-channel data Bit 1: Enable print function of editor Bit 2: With external program selection, test program
52000	$MCS_DISP_COORDINATE_SYSTEM	34		0: Vertical lathe, right hand head 6: Vertical lathe, left hand head 33: Horizontal lathe with machining in front of the turning centre 34: Horizontal lathe with machining behind the turning centre
52005	$MCS_DISP_PLANE_MILL	0		Milling: 0: Plane selection on the operator panel
52006	$MCS_DISP_PLANE_TURN	18		Turning: Always G18
52200	$MCS_TECHNOLOGY	1	Exact	Technology 0: no specific configuration 1: turning 2: milling
52201	$MCS_TECHNOLOGY_EXTENSION	2		Extended technology 0: no specific configuration 1: turning: extended technology Turning for Milling 2: milling

MD	Identifier	Settings	Comment	Description
52206	$MCS_AXIS_USAGE	[2] = 3 [3] = 1		Meaning of the axes 0 = no special meaning; 1 = tool spindle (driven tool); 2 = C axis of the tool spindle (driven tool); 3 = main spindle (turning); 4 = C axis of the main spindle (turning); 5 = counter spindle (turning); 6 = C axis of the counter spindle (turning); 7 = linear axis of the counter spindle (turning); tailstock (turning); 9 = steady rest (turning); 10 = B axis (turning), 12 = B axis of the counter spindle (turning), 13 = Traverse travel X of the counter spindle (turning)
52207	$MCS_AXIS_USAGE_ATTRIB	0H		Axis attributes Bit 0: Rotates around the 1st geometry axis (applies to rotary axes) Bit 1: Rotates around the 2nd geometry axis (applies to rotary axes) Bit 2: Rotates around the 3rd geometry axis (applies to rotary axes) Bit 3: Positive direction of rotation is counter-clockwise (applies to rotary axes) Bit 4: Displayed direction of rotation for M3 is counterclockwise (applies to spindles) Bit 5: Direction of rotation M3 corresponds to rotary axis minus (applies to spindles) This bit must be set in the same way as a PLC bit. For 840D sl: PLC bit DBnn.DBX17.6, nn = 31 + machine axis index For 828D: PLC bit DB38xx.DBX2001.6, xx is machine axis index Bit 6: Show rotary axis as offset target for measuring Bit 7: Offer rotation axis in position pattern Bit 8: Offer rotation axis to accept blank (at milling machines) Bit 9: Spindle is not SPOS-capable Bit 10: Rotary axis rotates around 1st geometry axis (for position pattern only) Bit 11: Rotary axis rotates around 2nd geometry axis (for position pattern only) Bit 12: Rotary axis rotates around 3rd geometry axis (for position pattern only) Bit 13: Position of the axis for blank selection from active WO (for rotary axes)
52210	$MCS_FUNCTION_MASK_DISP	Bit 0 = 1 Bit 1 = 1 Bit 3 = 1		Function mask Display Bit 0: Measuring system for programs always in the base system Bit 1: Front view for turning in school coordinate system Bit 2: Hide “T,S,M” soft key in JOG area Bit 3: Generate automatic end-of-program in MDI (with the “Delete blocks” soft key) Bit 4: Show follow-on tool in T,F,S window Bit 5: Hide softkey “Actual machine values“ Bit 6: Hide tool radius/diameter in T,F,S window Bit 7: Hide tool length in T,F,S window Bit 8: Hide tool icon in T,F,S window Bit 9: Display selection ”Simple input” Bit 10: Do not offer channel in job lists Bit 11: For Work activate the display of the logical spindles Bit 12: Hide mold making view for G code Bit 13: Hide channel axes which are currently located in another channel Bit 14: Display overlay of DRF under machine Bit 15: Display overlay of tool ($AA_TOFF) under machine

MD	Identifier	Settings	Comment	Description
52211	$MCS_FUNCTION_MASK_DISP_ZOA	1FFFF5H		Function screen for displaying overview of work offsets Bit 0: Show machine position Bit 2: Display DRF offset Bit 3: Display $AA_OFF position offset Bit 4: Display $P_PARTFRAME Bit 5: Display $P_SETFRAME Bit 6: Display $P_EXTSFRAME Bit 7: Display $P_ISO1FRAME Bit 8: Display $P_ISO2FRAME Bit 9: Display $P_ISO3FRAME Bit10: Display $P_ACTBFRAME Bit 11: Display $P_IFRAME Bit 12: Display $P_TOOLFRAME Bit 13: Display $P_WPFRAME Bit 14: Display $P_TRAFRAME Bit 15: Display $P_PFRAME Bit 16: Display $P_ISO4FRAME Bit 17: Display $P_CYCFRAME Bit 18: Display sum of work offsets Bit 19: Display offset of active tool Bit 20: Display Work position Bit 21: Display BCS position Bit 22: Display $P_GFRAME Bit 23: Display TOFF Bit 24: Display overlay $AA_TOFF
52212	$MCS_FUNCTION_MASK_TECH	Bit 3 = 1	Exact	Function mask Cross-technology Bit 0: Enable swivel plane/align tool with tool carrier Bit 1: Optimized travel along software limit switches (ShopMill/ShopTurn) Bit 2: Special approach logic for step drill (ShopTurn) Bit 3: Call block search cycle for ShopMill/ShopTurn Bit 4: Approach logic through cycle (ShopTurn) Bit 5: Call block search cycle for SERUPRO Bit 6: Work offset ZV cannot be entered as an absolute value (ShopTurn) Bit 7: Detect the expiry of tool lives (ShopMill/ShopTurn) Bit 8: Manual machine (ShopMill/ShopTurn) Bit 9: Selection/deselection of work offset via softkey Bit 10: Enable swivel plane/align tool using kinematic transformation Bit 11: Switch off layer check for drilling and milling tools (ShopTurn) Bit 13: Approach tool change point Y in Machine (ShopTurn) Bit 14: With approach logic only check tool position with cutting edge (ShopTurn)
52214	$MCS_FUNCTION_MASK_MILL	0H		Function mask Milling Bit 0: Enable cylinder surface transformation (ShopMill) Bit 1: Offer fixed table for blank clamping (at milling machines) Bit 2: Special handling of slide mill/saw for polyhedron Bit 3: Enable inside/rear machining (ShopTurn) Bit 4: Enable spindle clamping (C axis) (ShopTurn) Bit 5: Enable spindle control of tool spindle via surface Bit 6: Enable spindle control of turning spindle via surface Bit 7: Offer fixed table for blank clamping (on boring mills)

MD	Identifier	Settings	Comment	Description
52216	$MCS_FUNCTION_MASK_DRILL	0H		Function mask Drilling Bit 0: CYCLE84 Unhide technology fields Bit 1: CYCLE840 Unhide technology fields Bit 2: Thread tapping also without encoder (ShopMill/ShopTurn) Bit 3: Constant cutting speed in rel. to diameter of centering (ShopMill/ShopTurn)
52218	$MCS_FUNCTION_MASK_TURN	0H		Function mask Turning Bit 0: Enable zoom under manual for tool measurement Bit 1: Enable parts gripper for cut-off Bit 2: Enable tailstock Bit 3: Enable spindle control of main spindle via surface Bit 4: Enable spindle control of counterspindle via surface Bit 5: Enable spindle control of tool spindle via surface Bit 6: Enable balance cutting for dual-channel stock removal Bit 7: Retraction during stock removal along contour with G1 Bit 8: Input spindle chuck data in the program Bit 9: Additional entry of tailstock data in the program Bit 10: Enable convex thread Bit 11: Reserved Bit 12: Deactivate thread synchronization Bit 13: Stock rem. contour with CYCLE95 (828D programGUIDE without Advanced Technology)
52229	$MCS_ENABLE_QUICK_M_CODES	0H		Enable fast M functions Bit 0: Coolant OFF Bit 1: Coolant 1 ON Bit 2: Coolant 2 ON Bit 3: Coolant 1 and 2 ON
52230	$MCS_M_CODE_ALL_COOLANTS_OFF	9		M code for all coolants OFF
52231	$MCS_M_CODE_COOLANT_1_ON	8		M code for coolant 1 ON
52232	$MCS_M_CODE_COOLANT_2_ON	7		M code for coolant 2 ON
52233	$MCS_M_CODE_COOLANT_1_AND_2_ON	-1		M code for coolant 1 + 2 ON
52250	$MCS_M_CODE_CHUCK_OPEN			[0]: M code for Open chuck with non-rotating main spindle [1]: M code for Open chuck with non-rotating counter spindle
52251	$MCS_M_CODE_CHUCK_OPEN_ROT			[0]: M code for Open chuck with rotating main spindle [1]: M code for Open chuck with rotating counter spindle
52252	$MCS_M_CODE_CHUCK_CLOSE			[0]: M code for Close chuck main spindle [1]: M code for Close chuck counter spindle

MD	Identifier	Settings	Comment	Description
52270	$MCS_TM_FUNCTION_MASK	0H		Function mask Tool management Bit 0: Do not permit Create tool on magazine location Bit 1: Load/unload disable, if machine is not in reset Bit 2: Load/unload disable for emergency stop Bit 3: Load/unload tool or relocate tool to/from spindle disabled Bit 4: Load directly in spindle Bit 5: Use grinding configuration file Bit 6: Permit relocating to/from spindle despite a block (Bit 3) Bit 7: Create tool using the T number Bit 8: Hide Relocate tool Bit 9: Hide Position magazine Bit 10: Reactivate tool using Position magazine Bit 11: Reactivate tool in all monitoring modes Bit 12: Fade out Reactivate tool
53220	$MAS_AXIS_MCS_POSITION	0		Position of axis in the Machine
53230	$MAS_SIM_START_POSITION	X=400 Z=0		Axis position at start of simulation Only adjust with geometry axes
53240	$MAS_SPINDLE_PARAMETER	0		[0]: Spindle chuck: Chuck dimension ZC1 [1]: Spindle chuck: Stop dimension ZS1 [2]: Spindle chuck: Jaw dimension ZE2 (counter spindle only) in the data set of the main spindle and the counter spindle
53241	$MAS_SPINDLE_CHUCK_TYPE	0		Spindle jaw type only for counter spindle = 0: Jaw type 1 = 1: Jaw type 2
53242	$MAS_TAILSTOCK_PARAMETER	0		[0] Tailstock diameter [1] Tailstock length
54215	$SNS_TM_FUNCTION_MASK_SET	Bit 0 = 1		Function mask Tool management Bit 0: Diameter display for rotating tools Bit 1: Default direction of rotation for all turning tools is M4 Bit 2: Create tool without name suggestion Bit 3: Input disable for tool name and tool type in loaded tools Bit 4: Input disable for loaded tools, if the channel is not in Reset Bit 5: Calculate tool wear inputs additively Bit 6: Numerical input of the tool ident. Bit 7: Fade out tool monitoring parameters Bit 8: Diameter display for transverse axis - Geometry Bit 9: Diameter display for transverse axis - Wear Bit 10: Enable loading/relocating of tools on buffer locations Bit 11: Creating of new tools in gripper locations is disabled Bit 12: Do not unload measuring tools if ”Unload all” function is executed Bit 13: Do not delete tool wear when inputting the geometry value Bit 14: Load/relocate tool. Search without pre-assignment of the magazine
55200	$SCS_MAX_INPUT_FEED_PER_REV	1		Feedrate input upper limit for mm/rev
55201	$SCS_MAX_INPUT_FEED_PER_TIME	10000		Feedrate input upper limit for mm/min
55202	$SCS_MAX_INPUT_FEED_PER_TOOTH	1		Feedrate input upper limit for mm/tooth

MD	Identifier	Settings	Comment	Description
55212	$SCS_FUNCTION_MASK_TECH_SET	Bit 1 = 1 Bit 2 = 1		Function mask Cross-technology Bit 0: Tool preselection active Bit 1: Calculate thread depth from metric thread lead Bit 2: Refer to the table for thread diameter and depth Bit 3: Delete programs generated by contour cycles
55214	$SCS_FUNCTION_MASK_MILL_SET	Bit 0 = 1 Bit 2 = 1		Function mask Milling Bit 0: Default setting - milling cycles with synchronous operation Bit 2: Depth calculation in milling cycles without parameter SC = 0: Depth calculation in milling cycles is carried out between reference plane + safety clearance and depth = 1: Depth calculation in milling cycles is carried out without inclusion of the safety clearance Note: Bit 2 is active with the following milling cycles: CYCLE61, CYCLE71, CYCLE76, CYCLE77, CYCLE79, CYCLE899, LONGHOLE, SLOT1, SLOT2, POCKET3, POCKET4
55216	$SCS_FUNCTION_MASK_DRILL_SET	Bit 3 = 1 Bit 4 = 1		Function mask Drilling Bit 0: Reverse direction of spindle rotation during tapping Bit 1: Consider rotation of the tool plane (CYCLE86) Bit 2: Consider swiveled table kinematics (CYCLE86) Bit 3: Monitoring spindle MD 31050/31060 (CYCLE84) Bit 4: Monitoring spindle MD 31050/31060 (CYCLE840) Bit 5: reserved Bit 6: Adapt spindle position to the tool orientation (CYCLE86) Bit 7: Adapt direction of retraction path in the plane to active mirroring (CYCLE86)
55218	$SCS_FUNCTION_MASK_TURN_SET	Bit 0 = 1		Function mask Turning Bit 0: New thread table during thread cutting Bit 1: Reserved (CYCLE93) Bit 2: Reserved (CYCLE93) Bit 3: Chamfer in CYCLE930 as chamfer length (CHF) Bit 4: Retraction from the inside machining (CYCLE951)
55300	$SCS_EASY_SAFETY_CLEARANCE	1		Simple input: Safety clearance
55301	$SCS_EASY_DWELL_TIME	0.6		Simple input: dwell time
55305	$SCS_EASY_DRILL_DEEP_FD1	90		Simple input: percentage 1st federate deep - hole drilling
55306	$SCS_EASY_DRILL_DEEP_DF	90		Simple input: percentage infeed deep - hole drilling
55307	$SCS_EASY_DRILL_DEEP_V1	1.2		Simple input: min. depth infeed deep - hole drilling
55308	$SCS_EASY_DRILL_DEEP_V2	1.4		Simple input: absolute retraction value deep - hole drilling
55309	$SCS_EASY_THREAD_RETURN_DIST	2		Simple input: return distance thread tapping
55400	$SCS_MILL_ENGRAVE_POINT_RAD	0		Engraving cycle CYCLE60: Circ. path radius for generating the char. ”point”

MD	Identifier	Settings	Comment	Description
55500	$SCS_TURN_FIN_FEED_PERCENT	100		Only for ShopTurn cycles with complete machining roughing and finishing - Grooving cycles : all - Undercut cycles : “Thread DIN” and “Thread” - Thread cycles : all
55505	$SCS_TURN_ROUGH_O_RELEASE_DIST	1		Return distance stock removal for external machining
55506	$SCS_TURN_ROUGH_I_RELEASE_DIST	0,5		Return distance stock removal for internal machining
55510	$SCS_TURN_GROOVE_DWELL_TIME	-1		negative value in spindle revolutions positive value in seconds
55540	$SCS_TURN_PART_OFF_CTRL_DIST	0,1		Path for cut-off check
55541	$SCS_TURN_PART_OFF_CTRL_FEED	0		Feedrate for cut-off check
55542	$SCS_TURN_PART_OFF_CTRL_FORCE	10		Force in percent for cut-off check, in %
55580	$SCS_TURN_CONT_RELEASE_ANGLE	45		Contour turning: retraction angle
55581	$SCS_TURN_CONT_RELEASE_DIST	1		Contour turning: retraction value
55582	$SCS_TURN_CONT_TRACE_ANGLE	5		Contour turning: minimum angle for rounding along contour
55583	$SCS_TURN_CONT_VARIABLE_DEPTH	20		Contour turning: percentage for variable cutting depth
55584	$SCS_TURN_CONT_BLANK_OFFSET	1		Contour turning: blank allowance
55585	$SCS_TURN_CONT_INTERRUPT_TIME	-1		Contour turning: feed interrupt time negative value in spindle revolutions positive value in seconds
55586	$SCS_TURN_CONT_INTER_RETRACTION	1		Contour turning: retraction path after feed interrupt
55587	$SCS_TURN_CONT_MIN_REST_MAT_AX1	50		Contour turning: minimum difference dimension residual machining axis 1
55588	$SCS_TURN_CONT_MIN_REST_MAT_AX2	50		Contour turning: minimum difference dimension residual machining axis 2
55595	$SCS_TURN_CONT_TOOL_BEND_RETR	0,1		Contour plunge turning: retraction path due to tool bending
55596	$SCS_TURN_CONT_TURN_RETRACTION	0,1		Contour plunge turning: retraction depth prior to turning

Machine data list turning

You should work through and configure the following machine data when you set up the turning machine.

MD	Identifier	Settings	Comment	Description
10260	$MN_CONVERT_SCALING_SYSTEM	1	Exact	Enable basic system conversion
10602	$MN_FRAME_GEOAX_CHANGE_MODE	1	Exact	Frames when changing geometry axes
10610	$MN_MIRROR_REF_AX	0	Exact	Reference axis for mirroring
10714	$MN_M_NO_FCT_EOP	32	Variable	M function for spindle active after reset
10722	$MN_AXCHANGE_MASK	Bit 2 = 1	Exact	Parameters for axis replacement behaviour
11410	$MN_SUPPRESS_ALARM_MASK	Bit 0 = 1 Bit 1 = 1 Bit 15 =1 Bit 20 =1	Variable	Bit 0: 15110: REORG not possible Bit 1: 10763 The path component of the block in the contour plane is zero Bit 15: 5000 Communication order cannot be executed Bit 20: 2900 Reboot is delayed
11450	$MN_SEARCH_RUN_MODE	Bit 0 = 1 Bit 1 = 1 Bit 2 = 1	Exact Exact Exact	Bit 0: Machining is stopped with the loading of the last action block after search run Bit 1: Automatic ASUP start after output of action blocks Bit 2: The output of the auxiliary functions in the action blocks is suppressed
11470	$MN_REPOS_MODE_MASK	Bit 3 = 1	Exact	Mask for support of special alarm outputs
17530	$MN_TOOL_DATA_CHANGE_COUNTER	1FH	Exact	Mark tool data change for HMI
18080	$MN_MM_TOOL_MANAGEMENT_MASK	Bit 0 = 1 Bit 1 = 1 Bit 3 = 1	Exact Exact Exact	Bit 0: Memory for TM-specific data is provided Bit 1: Memory for monitoring data (WZMO) is provided Bit 3: Memory to consider adjacent location is provided
19320	$ON_TECHNO_FUNCTION_MASK	Bit 4 = 1	Exact	Tool management with magazine data
20100	$MC_DIAMETER_AX_DEF	X	Variable	Geometry axis with transverse axis function
20110	$MC_RESET_MODE_MASK	Bit 0 = 1 Bit 6 = 1 Bit 14 =1 Bit 16 =0	Exact Exact Exact Variable	Bit 0 = 1 G codes acc. to MD20150 Bit 6 = 1 Current setting for active tool length offset is retained after reset/end-of-part-program Bit 14 = 1 The current setting of the basic frame is retained Bit 16 = 0 Initial setting for master spindle acc. to MD20090
20124	$MC_TOOL_MANAGEMENT_TOOLHOLDER	1	Exact	Tool holder number
20128	$MC_COLLECT_TOOL_CHANGE	0	Exact	No tool change commands after search run
20130	$MC_CUTTING_EDGE_RESET_VALUE	1	Exact	Tool and cutting edge remain activated
20150	$MC_G_CODE_RESET_VALUE	[5] = 2 [15] = 3 [21] = 2 [28] = 2 [48] = 3	Exact Variable Variable Exact Variable	Group 6: Plane selection 2 = 18 Group 16: Feedrate override 3 = CFIN Group 22: Tool offset type 2 = CUT2DF Group 29: Radius/diameter programming 2 = DIAMON Group 49: Point-to-point motion 3 = PTPG0
20152	$MC_G_CODE_RESET_MODE	[5] = 0 [7] = 1 [12] = 0 [14] = 1 [15] = 0 [21] = 0	Exact Exact Exact Exact Exact Variable	Group 6: Plane selection remains active after Reset Group 8: Zero point offset remains active after Reset Group 13: Work piece dimensioning inch/metric default after reset Group 15: Type of feed remains active after Reset Group 16: Feed correction remains active after Reset Group 22: Tool correction type default after reset
20202	$MC_WAB_MAXNUM_DUMMY_BLOCKS	10	Minimum	maximum number of blocks w/o traversing movement with SAR
20240	$MC_CUTCOM_MAXNUM_CHECK_BLOCKS	4	Minimum	Blocks for look-ahead contour calculation with TRC
20250	$MC_CUTCOM_MAXNUM_DUMMY_BLOCKS	5	Minimum	maximum number of blocks without traversing motion in TRC

MD	Identifier	Settings	Comment	Description
20310	$MC_TOOL_MANAGEMENT_MASK	Bit 0 = 1 Bit 1 = 1 Bit 3 = 1 Bit 14 =1 Bit 16 =1 Bit 23 =1	Exact Exact Exact Exact Exact Variable	Bit 0: Tool management active Bit 1: Tool monitoring function active Bit 3: Consider adjacent location active Bit 14: Automatic tool change during RESET and Start Bit 16: Programming T='Location number' Bit 23: On offset selection no synchronization with HL
20320	$MC_TOOL_TIME_MONITOR_MASK	Bit 0 = 1	Variable	Time monitoring for tool in tool holder 1
20360	$MC_TOOL_PARAMETER_DEF_ MASK	Bit 0 = 1	Variable	Definition of tool parameters
20734	$MC_EXTERN_FUNCTION_MASK	Bit 3 = 1	Variable	Errors in ISO scanner are not output, the block is transferred to the Siemens translator.
22550	$MC_TOOL_CHANGE_MODE	0	Exact	Tool change with T function
24006	$MC_CHSFRAME_RESET_MASK	Bit 0 = 1 Bit 5 = 0	Exact Exact	Active system frames after reset Bit 0 = 1 Actual value setting and scratching remains active Bit 5 = 0 System frame for cycles is deleted
24007	$MC_CHSFRAME_RESET_CLEAR_MASK	Bit 0 = 0 Bit 5 = 1	Exact Exact	Deletion of system frames after reset Bit 0 = 0 Actual value setting and scratching not delete Bit 5 = 1 System frame for cycles delete
24008	$MC_CHSFRAME_POWERON_MASK	Bit 0 = 0 Bit 5 = 0	Exact Exact	Reset channel system frames after power on Bit 0 = 0 Actual value setting and scratching not reset Bit 5 = 0 System frame for cycles not reset
24030	$MC_FRAME_ACS_SET	1	Exact	Adjustment of SZS coordinate system
24040	$MC_FRAME_ADAPT_MODE	7H	Exact	Adaptation of active frames
27860	$MC_PROCESSTIMER_MODE	Bit 0 = 1 Bit 1 = 1 Bit 4 = 1 Bit 5 = 1 Bit 6 = 1 Bit 8 = 1	Variable	Bit 0: Measurement of total operating time is active for all part programs Bit 1: Measurement of current program runtime is active Bit 4: Measurement also with active dry run feed Bit 5: Measurement also with program test Bit 6: $AC_CYCLE_TIME is not deleted on start by ASUP and PROG_EVENT Bit 8: AC_CYCLE_TIME will be deleted with GOTOS
27880	$MC_PART_COUNTER	Bit 0 = 1 Bit 8 = 1 Bit 11 =1	Variable	Bit 0: Counter $AC_REQUIRED_PARTS is activated Bit 8: Counter $AC_ACTUAL_PARTS is active Bit 11: Increment $AC_ACTUAL_PARTS with GOTOS
28000	$MC_MM_REORG_LOG_FILE_MEM	75	Minimum	Memory space for REORG (DRAM)
28010	$MC_MM_NUM_REORG_LUD_MODULES	20	Minimum	Number of blocks for local user variables in REORG (DRAM)
28082	$MC_MM_SYSTEM_FRAME_MASK	Bit 0 = 1 Bit 5 = 1 Bit 6 = 1	Exact Exact Exact	Setting up system frames (SRAM) Bit 0 = 1 System frame for actual value and scratching Bit 5 = 1 System frame for cycles Bit 6 = 1 System frame for transformations
28083	$MC_MM_SYSTEM_DATAFRAME_MASK	Bit 0 = 1 Bit 5 = 0	Exact Exact	Setting up data management system frames (SRAM) Bit 0 = 1 System frame for actual value and scratching Bit 5 = 0 System frame for cycles
28450	$MC_MM_TOOL_DATA_CHG_BUFF_SIZE	400	Exact	Buffer for tool data changes (DRAM)
35040	$MA_SPIND_ACTIV_AFTER_RESET	2	Exact	Own spindle RESET
42440	$SC_FRAME_OFFSET_INCR_PROG	0	Exact	Zero offsets in frames
42442	$SC_TOOL_OFFSET_INCR_PROG	0	Exact	Tool length offsets
42528	$SC_CUTCOM_DECEL_LIMIT	1	Variable	Feed lowering on circles with tool radius compensation
42940	$SC_TOOL_LENGTH_CONST	18	Exact	Change of tool length components with change of active plane
42950	$SC_TOOL_LENGTH_TYPE	2	Exact	Assignment of tool length compensation independent of tool type

MD	Identifier	Settings	Comment	Description
51024	$MNS_BLOCK_SEARCH_MODE_MASK_JS	Bit 0 = 1		Bit mask for available search modes (ShopMill/ShopTurn) Bit 0: Block search with calculation without approach Bit 1: Block search with calculation with approach Bit 3: Skip EXTCALL programs Bit 5: Block search with test run
51028	$MNS_BLOCK_SEARCH_MODE_MASK	Bit 0 = 1 Bit 1 = 1 Bit 4 = 1 Bit 5 = 1		Bit mask for available block search modes Bit 0: Block search with calculation without approach Bit 1: Block search with calculation with approach Bit 3: Skip EXTCALL programs Bit 4: Block search without calculation Bit 5: Block search with test run
51038	$MNS_SET_ACT_VALUE	1		Set actual value selection 0 = Set actual value is not offered. 1 = If a user frame (settable work offset e.g. G54) is active, it will be used. In G500 Set actual values is not offered (system frame is no longer used).
51039	$MNS_PROGRAM_CONTROL_MODE_MASK	Bit 0 = 1		Options for machine - program influence: Bit 0: Program test function available
51040	$MNS_SWITCH_TO_MACHINE_MASK	0H		Automatic operating area switchover to machine Bit 0: When selecting a program in the program manager, the operating area does not automatically switch over to Machine. Bit 1: When switching the type of operation via the MCP, the operating area is not automatically switched over to Machine. Bit 2: Do not switch automatically in Programs during program selection. Bit 3: No auto. Start of block search on selection / execution under Programs.
51226	$MNS_FUNCTION_MASK_SIM	0H		Function mask Simulation Bit 0: No automatic start on simulation selection Bit 1: Deactivate simulation Bit 4: Ignore frame components in X and Y (compatibility) Bit 5: Turning tool can rotate in the tool spindle Bit 6: Enable handwheel as simulation override Bit 7: Interpretation of handwheel values as absolute values Bit 10: Hide zero point symbol
51228	$MNS_FUNCTION_MASK_TECH	0H		Function mask Cross-technology Bit 0: G code programming without multi-channel data Bit 1: Enable print function of editor Bit 2: With external program selection, test program
52000	$MCS_DISP_COORDINATE_SYSTEM	34		0: Vertical lathe, right hand head 6: Vertical lathe, left hand head 33: Horizontal lathe with machining in front of the turning centre 34: Horizontal lathe with machining behind the turning centre
52005	$MCS_DISP_PLANE_MILL	0		Milling: 0: Plane selection on the operator panel
52006	$MCS_DISP_PLANE_TURN	18		Turning: Always G18
52200	$MCS_TECHNOLOGY	1	Exact	Technology 0: no specific configuration 1: turning 2: milling
52201	$MCS_TECHNOLOGY_EXTENSION	2		Extended technology 0: no specific configuration 1: turning: extended technology Turning for Milling 2: milling

MD	Identifier	Settings	Comment	Description
52206	$MCS_AXIS_USAGE	[2] = 3 [3] = 1		Meaning of the axes 0 = no special meaning; 1 = tool spindle (driven tool); 2 = C axis of the tool spindle (driven tool); 3 = main spindle (turning); 4 = C axis of the main spindle (turning); 5 = counter spindle (turning); 6 = C axis of the counter spindle (turning); 7 = linear axis of the counter spindle (turning); tailstock (turning); 9 = steady rest (turning); 10 = B axis (turning), 12 = B axis of the counter spindle (turning), 13 = Traverse travel X of the counter spindle (turning)
52207	$MCS_AXIS_USAGE_ATTRIB	0H		Axis attributes Bit 0: Rotates around the 1st geometry axis (applies to rotary axes) Bit 1: Rotates around the 2nd geometry axis (applies to rotary axes) Bit 2: Rotates around the 3rd geometry axis (applies to rotary axes) Bit 3: Positive direction of rotation is counter-clockwise (applies to rotary axes) Bit 4: Displayed direction of rotation for M3 is counterclockwise (applies to spindles) Bit 5: Direction of rotation M3 corresponds to rotary axis minus (applies to spindles) This bit must be set in the same way as a PLC bit. For 840D sl: PLC bit DBnn.DBX17.6, nn = 31 + machine axis index For 828D: PLC bit DB38xx.DBX2001.6, xx is machine axis index Bit 6: Show rotary axis as offset target for measuring Bit 7: Offer rotation axis in position pattern Bit 8: Offer rotation axis to accept blank (at milling machines) Bit 9: Spindle is not SPOS-capable Bit 10: Rotary axis rotates around 1st geometry axis (for position pattern only) Bit 11: Rotary axis rotates around 2nd geometry axis (for position pattern only) Bit 12: Rotary axis rotates around 3rd geometry axis (for position pattern only) Bit 13: Position of the axis for blank selection from active WO (for rotary axes)
52210	$MCS_FUNCTION_MASK_DISP	Bit 0 = 1 Bit 1 = 1 Bit 3 = 1		Function mask Display Bit 0: Measuring system for programs always in the base system Bit 1: Front view for turning in school coordinate system Bit 2: Hide “T,S,M” soft key in JOG area Bit 3: Generate automatic end-of-program in MDI (with the “Delete blocks” soft key) Bit 4: Show follow-on tool in T,F,S window Bit 5: Hide softkey “Actual machine values“ Bit 6: Hide tool radius/diameter in T,F,S window Bit 7: Hide tool length in T,F,S window Bit 8: Hide tool icon in T,F,S window Bit 9: Display selection ”Simple input” Bit 10: Do not offer channel in job lists Bit 11: For Work activate the display of the logical spindles Bit 12: Hide mold making view for G code Bit 13: Hide channel axes which are currently located in another channel Bit 14: Display overlay of DRF under machine Bit 15: Display overlay of tool ($AA_TOFF) under machine

MD	Identifier	Settings	Comment	Description
52211	$MCS_FUNCTION_MASK_DISP_ZOA	1FFFF5H		Function screen for displaying overview of work offsets Bit 0: Show machine position Bit 2: Display DRF offset Bit 3: Display $AA_OFF position offset Bit 4: Display $P_PARTFRAME Bit 5: Display $P_SETFRAME Bit 6: Display $P_EXTSFRAME Bit 7: Display $P_ISO1FRAME Bit 8: Display $P_ISO2FRAME Bit 9: Display $P_ISO3FRAME Bit10: Display $P_ACTBFRAME Bit 11: Display $P_IFRAME Bit 12: Display $P_TOOLFRAME Bit 13: Display $P_WPFRAME Bit 14: Display $P_TRAFRAME Bit 15: Display $P_PFRAME Bit 16: Display $P_ISO4FRAME Bit 17: Display $P_CYCFRAME Bit 18: Display sum of work offsets Bit 19: Display offset of active tool Bit 20: Display Work position Bit 21: Display BCS position Bit 22: Display $P_GFRAME Bit 23: Display TOFF Bit 24: Display overlay $AA_TOFF
52212	$MCS_FUNCTION_MASK_TECH	Bit 3 = 1	Exact	Function mask Cross-technology Bit 0: Enable swivel plane/align tool with tool carrier Bit 1: Optimized travel along software limit switches (ShopMill/ShopTurn) Bit 2: Special approach logic for step drill (ShopTurn) Bit 3: Call block search cycle for ShopMill/ShopTurn Bit 4: Approach logic through cycle (ShopTurn) Bit 5: Call block search cycle for SERUPRO Bit 6: Work offset ZV cannot be entered as an absolute value (ShopTurn) Bit 7: Detect the expiry of tool lives (ShopMill/ShopTurn) Bit 8: Manual machine (ShopMill/ShopTurn) Bit 9: Selection/deselection of work offset via softkey Bit 10: Enable swivel plane/align tool using kinematic transformation Bit 11: Switch off layer check for drilling and milling tools (ShopTurn) Bit 13: Approach tool change point Y in Machine (ShopTurn) Bit 14: With approach logic only check tool position with cutting edge (ShopTurn)
52214	$MCS_FUNCTION_MASK_MILL	0H		Function mask Milling Bit 0: Enable cylinder surface transformation (ShopMill) Bit 1: Offer fixed table for blank clamping (at milling machines) Bit 2: Special handling of slide mill/saw for polyhedron Bit 3: Enable inside/rear machining (ShopTurn) Bit 4: Enable spindle clamping (C axis) (ShopTurn) Bit 5: Enable spindle control of tool spindle via surface Bit 6: Enable spindle control of turning spindle via surface Bit 7: Offer fixed table for blank clamping (on boring mills)

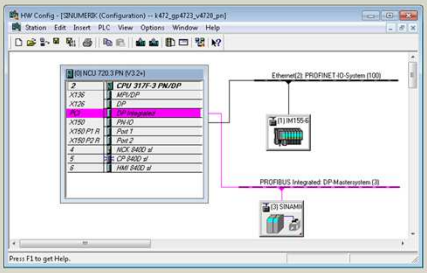
MD	Identifier	Settings	Comment	Description
52216	$MCS_FUNCTION_MASK_DRILL	0H		Function mask Drilling Bit 0: CYCLE84 Unhide technology fields Bit 1: CYCLE840 Unhide technology fields Bit 2: Thread tapping also without encoder (ShopMill/ShopTurn) Bit 3: Constant cutting speed in rel. to diameter of centering (ShopMill/ShopTurn)
52218	$MCS_FUNCTION_MASK_TURN	0H		Function mask Turning Bit 0: Enable zoom under manual for tool measurement Bit 1: Enable parts gripper for cut-off Bit 2: Enable tailstock Bit 3: Enable spindle control of main spindle via surface Bit 4: Enable spindle control of counterspindle via surface Bit 5: Enable spindle control of tool spindle via surface Bit 6: Enable balance cutting for dual-channel stock removal Bit 7: Retraction during stock removal along contour with G1 Bit 8: Input spindle chuck data in the program Bit 9: Additional entry of tailstock data in the program Bit 10: Enable convex thread Bit 11: Reserved Bit 12: Deactivate thread synchronization Bit 13: Stock rem. contour with CYCLE95 (828D programGUIDE without Advanced Technology)
52229	$MCS_ENABLE_QUICK_M_CODES	0H		Enable fast M functions Bit 0: Coolant OFF Bit 1: Coolant 1 ON Bit 2: Coolant 2 ON Bit 3: Coolant 1 and 2 ON
52230	$MCS_M_CODE_ALL_COOLANTS_OFF	9		M code for all coolants OFF
52231	$MCS_M_CODE_COOLANT_1_ON	8		M code for coolant 1 ON
52232	$MCS_M_CODE_COOLANT_2_ON	7		M code for coolant 2 ON
52233	$MCS_M_CODE_COOLANT_1_AND_2_ON	-1		M code for coolant 1 + 2 ON
52250	$MCS_M_CODE_CHUCK_OPEN			[0]: M code for Open chuck with non-rotating main spindle [1]: M code for Open chuck with non-rotating counter spindle
52251	$MCS_M_CODE_CHUCK_OPEN_ROT			[0]: M code for Open chuck with rotating main spindle [1]: M code for Open chuck with rotating counter spindle
52252	$MCS_M_CODE_CHUCK_CLOSE			[0]: M code for Close chuck main spindle [1]: M code for Close chuck counter spindle

MD	Identifier	Settings	Comment	Description
52270	$MCS_TM_FUNCTION_MASK	0H		Function mask Tool management Bit 0: Do not permit Create tool on magazine location Bit 1: Load/unload disable, if machine is not in reset Bit 2: Load/unload disable for emergency stop Bit 3: Load/unload tool or relocate tool to/from spindle disabled Bit 4: Load directly in spindle Bit 5: Use grinding configuration file Bit 6: Permit relocating to/from spindle despite a block (Bit 3) Bit 7: Create tool using the T number Bit 8: Hide Relocate tool Bit 9: Hide Position magazine Bit 10: Reactivate tool using Position magazine Bit 11: Reactivate tool in all monitoring modes Bit 12: Fade out Reactivate tool
53220	$MAS_AXIS_MCS_POSITION	0		Position of axis in the Machine
53230	$MAS_SIM_START_POSITION	X=400 Z=0		Axis position at start of simulation Only adjust with geometry axes
53240	$MAS_SPINDLE_PARAMETER	0		[0]: Spindle chuck: Chuck dimension ZC1 [1]: Spindle chuck: Stop dimension ZS1 [2]: Spindle chuck: Jaw dimension ZE2 (counter spindle only) in the data set of the main spindle and the counter spindle
53241	$MAS_SPINDLE_CHUCK_TYPE	0		Spindle jaw type only for counter spindle = 0: Jaw type 1 = 1: Jaw type 2
53242	$MAS_TAILSTOCK_PARAMETER	0		[0] Tailstock diameter [1] Tailstock length
54215	$SNS_TM_FUNCTION_MASK_SET	Bit 0 = 1		Function mask Tool management Bit 0: Diameter display for rotating tools Bit 1: Default direction of rotation for all turning tools is M4 Bit 2: Create tool without name suggestion Bit 3: Input disable for tool name and tool type in loaded tools Bit 4: Input disable for loaded tools, if the channel is not in Reset Bit 5: Calculate tool wear inputs additively Bit 6: Numerical input of the tool ident. Bit 7: Fade out tool monitoring parameters Bit 8: Diameter display for transverse axis - Geometry Bit 9: Diameter display for transverse axis - Wear Bit 10: Enable loading/relocating of tools on buffer locations Bit 11: Creating of new tools in gripper locations is disabled Bit 12: Do not unload measuring tools if ”Unload all” function is executed Bit 13: Do not delete tool wear when inputting the geometry value Bit 14: Load/relocate tool. Search without pre-assignment of the magazine
55200	$SCS_MAX_INPUT_FEED_PER_REV	1		Feedrate input upper limit for mm/rev
55201	$SCS_MAX_INPUT_FEED_PER_TIME	10000		Feedrate input upper limit for mm/min
55202	$SCS_MAX_INPUT_FEED_PER_TOOTH	1		Feedrate input upper limit for mm/tooth

MD	Identifier	Settings	Comment	Description
55212	$SCS_FUNCTION_MASK_TECH_SET	Bit 1 = 1 Bit 2 = 1		Function mask Cross-technology Bit 0: Tool preselection active Bit 1: Calculate thread depth from metric thread lead Bit 2: Refer to the table for thread diameter and depth Bit 3: Delete programs generated by contour cycles
55214	$SCS_FUNCTION_MASK_MILL_SET	Bit 0 = 1 Bit 2 = 1		Function mask Milling Bit 0: Default setting - milling cycles with synchronous operation Bit 2: Depth calculation in milling cycles without parameter SC = 0: Depth calculation in milling cycles is carried out between reference plane + safety clearance and depth = 1: Depth calculation in milling cycles is carried out without inclusion of the safety clearance Note: Bit 2 is active with the following milling cycles: CYCLE61, CYCLE71, CYCLE76, CYCLE77, CYCLE79, CYCLE899, LONGHOLE, SLOT1, SLOT2, POCKET3, POCKET4
55216	$SCS_FUNCTION_MASK_DRILL_SET	Bit 3 = 1 Bit 4 = 1		Function mask Drilling Bit 0: Reverse direction of spindle rotation during tapping Bit 1: Consider rotation of the tool plane (CYCLE86) Bit 2: Consider swiveled table kinematics (CYCLE86) Bit 3: Monitoring spindle MD 31050/31060 (CYCLE84) Bit 4: Monitoring spindle MD 31050/31060 (CYCLE840) Bit 5: reserved Bit 6: Adapt spindle position to the tool orientation (CYCLE86) Bit 7: Adapt direction of retraction path in the plane to active mirroring (CYCLE86)
55218	$SCS_FUNCTION_MASK_TURN_SET	Bit 0 = 1		Function mask Turning Bit 0: New thread table during thread cutting Bit 1: Reserved (CYCLE93) Bit 2: Reserved (CYCLE93) Bit 3: Chamfer in CYCLE930 as chamfer length (CHF) Bit 4: Retraction from the inside machining (CYCLE951)
55300	$SCS_EASY_SAFETY_CLEARANCE	1		Simple input: Safety clearance
55301	$SCS_EASY_DWELL_TIME	0.6		Simple input: dwell time
55305	$SCS_EASY_DRILL_DEEP_FD1	90		Simple input: percentage 1st federate deep - hole drilling
55306	$SCS_EASY_DRILL_DEEP_DF	90		Simple input: percentage infeed deep - hole drilling
55307	$SCS_EASY_DRILL_DEEP_V1	1.2		Simple input: min. depth infeed deep - hole drilling
55308	$SCS_EASY_DRILL_DEEP_V2	1.4		Simple input: absolute retraction value deep - hole drilling
55309	$SCS_EASY_THREAD_RETURN_DIST	2		Simple input: return distance thread tapping
55400	$SCS_MILL_ENGRAVE_POINT_RAD	0		Engraving cycle CYCLE60: Circ. path radius for generating the char. ”point”

MD	Identifier	Settings	Comment	Description
55500	$SCS_TURN_FIN_FEED_PERCENT	100		Only for ShopTurn cycles with complete machining roughing and finishing - Grooving cycles : all - Undercut cycles : “Thread DIN” and “Thread” - Thread cycles : all
55505	$SCS_TURN_ROUGH_O_RELEASE_DIST	1		Return distance stock removal for external machining
55506	$SCS_TURN_ROUGH_I_RELEASE_DIST	0,5		Return distance stock removal for internal machining
55510	$SCS_TURN_GROOVE_DWELL_TIME	-1		negative value in spindle revolutions positive value in seconds
55540	$SCS_TURN_PART_OFF_CTRL_DIST	0,1		Path for cut-off check
55541	$SCS_TURN_PART_OFF_CTRL_FEED	0		Feedrate for cut-off check
55542	$SCS_TURN_PART_OFF_CTRL_FORCE	10		Force in percent for cut-off check, in %
55580	$SCS_TURN_CONT_RELEASE_ANGLE	45		Contour turning: retraction angle
55581	$SCS_TURN_CONT_RELEASE_DIST	1		Contour turning: retraction value
55582	$SCS_TURN_CONT_TRACE_ANGLE	5		Contour turning: minimum angle for rounding along contour
55583	$SCS_TURN_CONT_VARIABLE_DEPTH	20		Contour turning: percentage for variable cutting depth
55584	$SCS_TURN_CONT_BLANK_OFFSET	1		Contour turning: blank allowance
55585	$SCS_TURN_CONT_INTERRUPT_TIME	-1		Contour turning: feed interrupt time negative value in spindle revolutions positive value in seconds
55586	$SCS_TURN_CONT_INTER_RETRACTION	1		Contour turning: retraction path after feed interrupt
55587	$SCS_TURN_CONT_MIN_REST_MAT_AX1	50		Contour turning: minimum difference dimension residual machining axis 1
55588	$SCS_TURN_CONT_MIN_REST_MAT_AX2	50		Contour turning: minimum difference dimension residual machining axis 2
55595	$SCS_TURN_CONT_TOOL_BEND_RETR	0,1		Contour plunge turning: retraction path due to tool bending
55596	$SCS_TURN_CONT_TURN_RETRACTION	0,1		Contour plunge turning: retraction depth prior to turning

SIEMENS





Chapter 8.2

Basic commissioning of the NC, PLC, drives

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Commissioning sequence 3
S7 project: Overview 4
S7 project: HW Config 5
S7 project: PLC program 11
Testing the PLC start-up 12
SINAMICS commissioning 13
SINAMICS commissioning: Topology 19
SINAMICS commissioning: Axis assignment 21
Setpoint and actual value configuration 24
Exercises: Commissioning of NC, PLC, and drives 25
Exercise: NC and PLC memory reset 26
Exercise: Creating an S7 project 27
Exercises for PLC hardware configuration: 29
Exercises: ET200 SP (PROFINET) 30
Notes on PROFINET 50
Exercises: ET200 S (PROFIBUS) 52
Exercises: Basic program 67
Exercise: Factory settings 72
Exercise: SINAMICS device configuration 73
Exercise: ALM commissioning 77
Exercise: Commissioning of Servo_3.3:3 with the Drive Wizard 81
Exercise: Assigning axes 86
Appendix 90

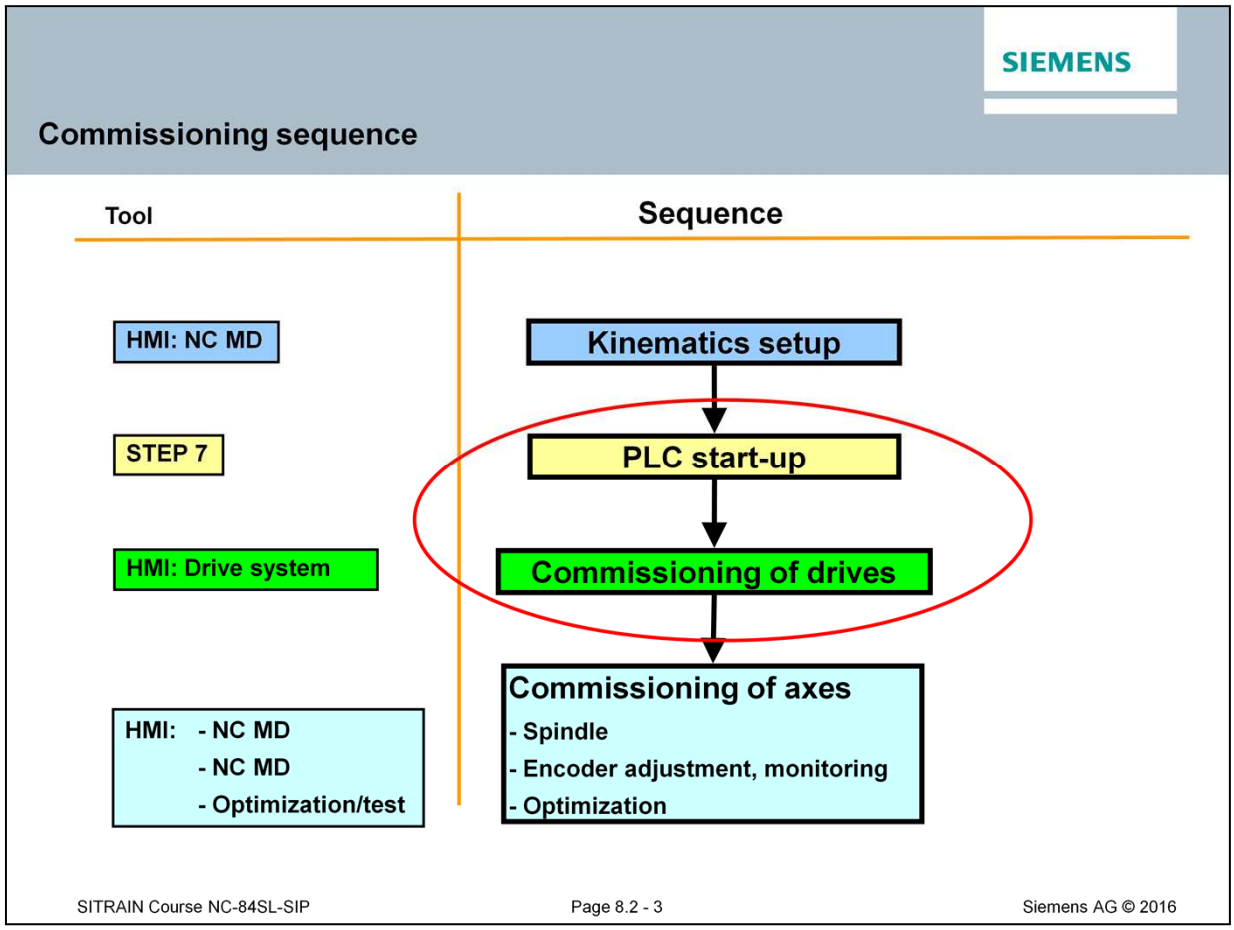
Learning objectives

- You will master basic commissioning of the NC, PLC, and the drives
- You will be able to perform the PLC basic commissioning (HW Config and basic program)
- You will be able to perform the device commissioning of the SINAMICS
- You will be able to parameterize drives and motors manually in the SINAMICS
- You will be able to perform axis assignment



 Siemens AG © 2016

Notes



Commissioning part:

PLC

The PLC start-up procedure comprises the following steps:

- HW Config
- PLC program for supplying MCP, enables, etc.

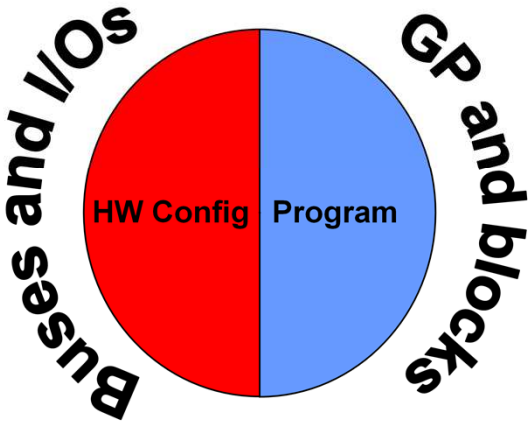
Drives

The drive commissioning procedure comprises the following steps:

- Device configuration (detection of DRIVE-CLiQ wiring = topology)
- Manual commissioning of drives not detected completely; second encoder, etc.
- Assignment of axes

SIEMENS

S7 project: Overview

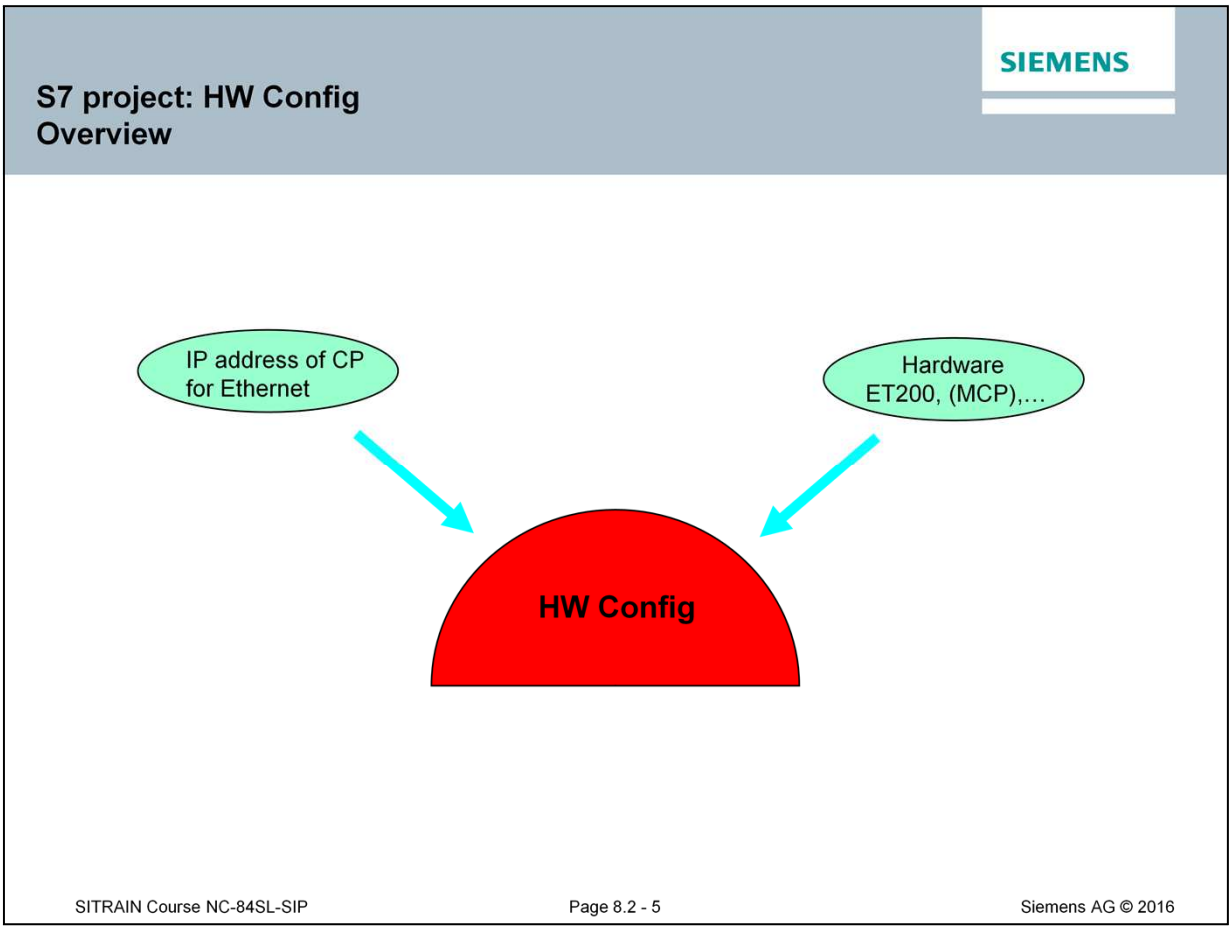


SITRAIN Course NC-84SL-SIP Page 8.2 - 4 Siemens AG © 2016

Requirements

Installation of a suitable Toolbox.
This installs the basic program and the current hardware is available in the hardware configuration.
For further components, the necessary GSD files have to be installed.

The S7 project provides the basis for setting up communication between the drives and the NCU, since the drives are always connected to the internal PROFIBUS of the PLC. If S7 is not configured, communication with the drives will not be possible.
The DP cycles (integrated isochronous PROFIBUS) correspond to the position control cycles.



NCU

First a suitable NCU is inserted from the hardware catalog. Now up to three PROFIBUS segments are created, PROFIBUS 1 and 2 (external) and PROFIBUS 3 (internal), as well as a PROFINET segment.

PROFINET

This interface has to be activated (networked) if a corresponding I/O is present (e.g. ET200 SP PROFINET).

External PROFIBUS

This interface has to be activated if a corresponding I/O is present (e.g. ET200 S PROFIBUS).

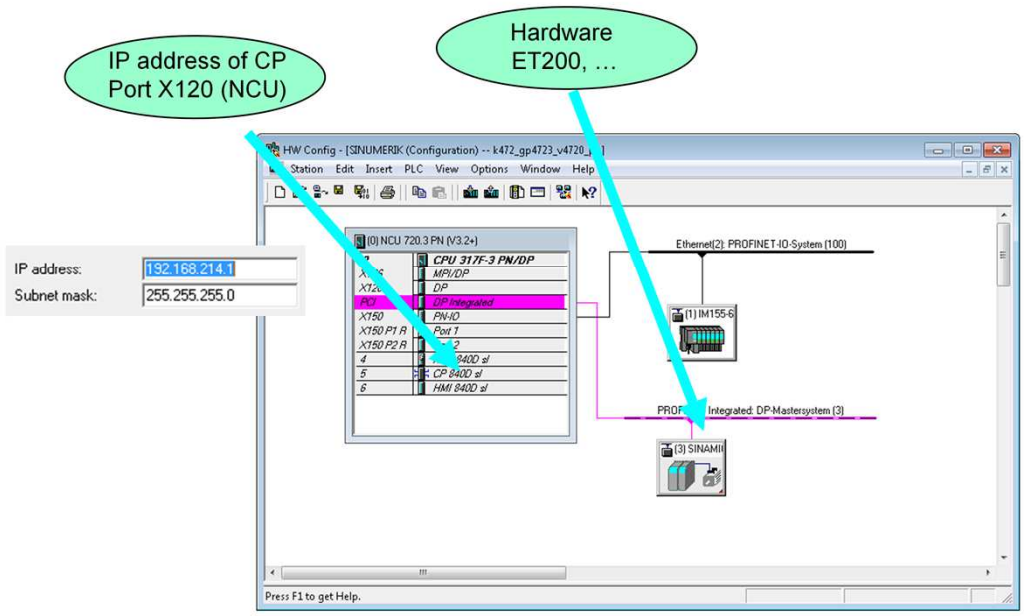
PROFINET / PROFIBUS

In both cases, the existing modules such as the interface module IM, digital input/output module DI/DO, etc. have to be be selected and parameterized.

IP address CP

The IP address of the CP must be set to the IP address of the NCU port used. This setting is used for online access from the SIMATIC Manager to the PLC (download HW Config, diagnostics HW Config online, download blocks, etc.)

S7 project: HW Config Details (IP address CP)



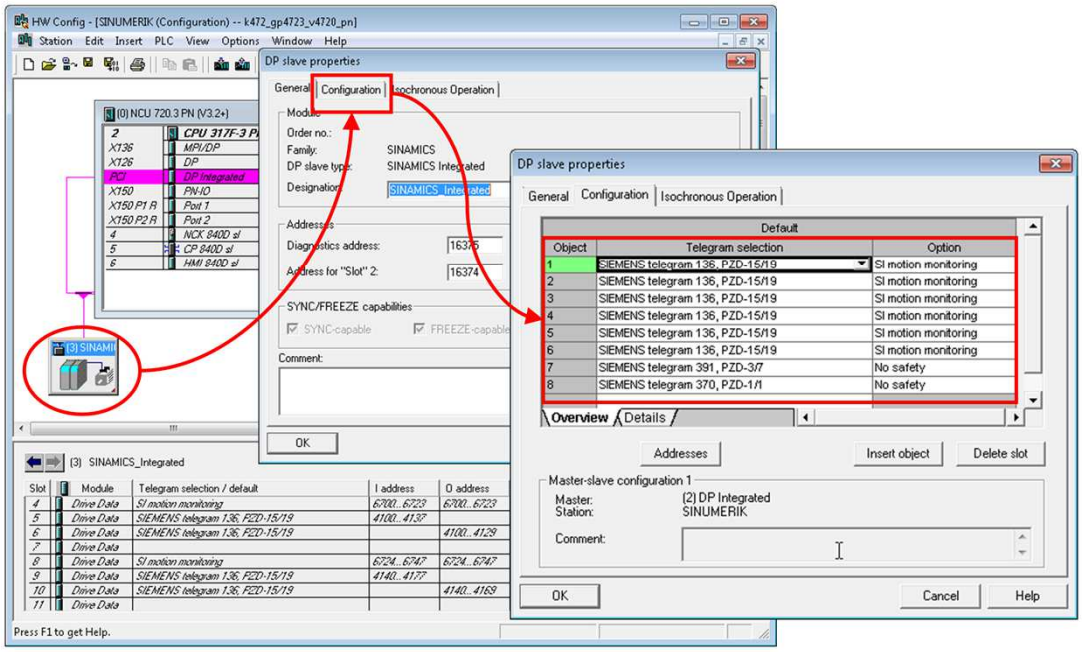
IP address of CP

To regulate communication between the user interface and the NC/PLC, the IP address and the subnet mask must be set for the CP; these settings must be identical to those in the Linux system.

By default, the connection is made as follows:

X120: IP addr. = 192.168.214.1 Subnet mask = 255.255.255.0

X127: IP addr. = 192.168.215.1 Subnet mask = 255.255.255.224



HW Config

We can also check the PROFIBUS configuration for the newly created station in the HW Config tool.

Telegrams

The following command displays an overview of telegrams:

-> SINAMICS (double-click -> Object properties)

-> 'Configuration' tab

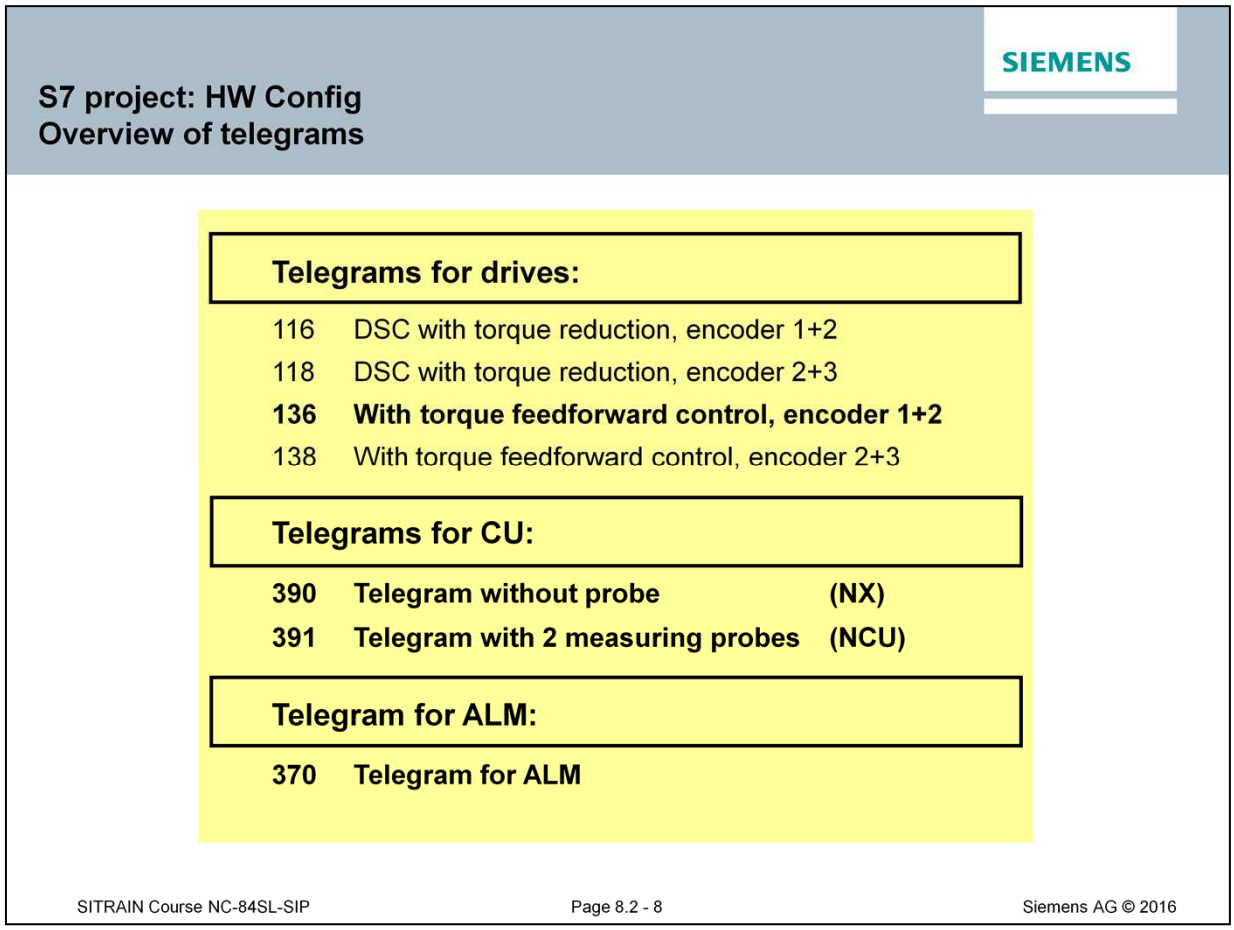
The telegrams listed here affect some of the machine data in the NC and drive.

PLC objects

The individual PLC objects are:

Object No.	Telegrams		Object type
	NCU 7x0.2	NCU 7x0.3	
1	116	136	1st drive
2	116	136	2nd drive
3	116	136	3rd drive
4	116	136	4th drive
5	116	136	5th drive
6	116	136	6th drive
7	391	391	CU
8	370	370	ALM

6 drives are always entered here, regardless of the actual number of drives.

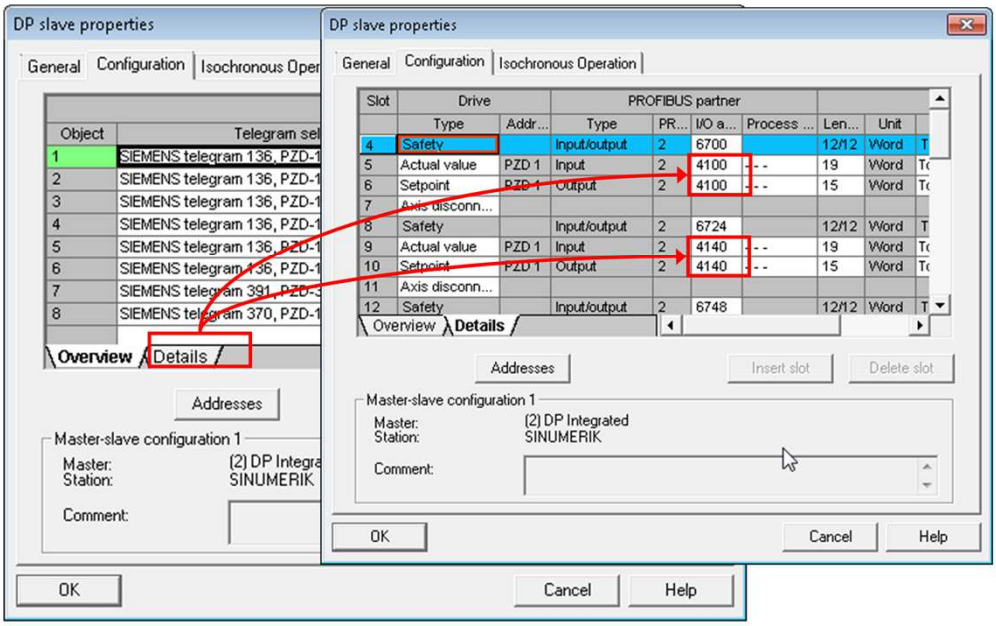


Telegrams with 840D sl

The telegrams used with SINUMERIK 840D sl are listed above.

Bold font indicates the telegrams according to the hardware configuration (NCU 7x0.3).

S7 project: HW Config
I/O addresses of the drives



Details Here you can see the I/O addresses of the communication partners.

Drive addresses The addresses are assigned according to the following rules:

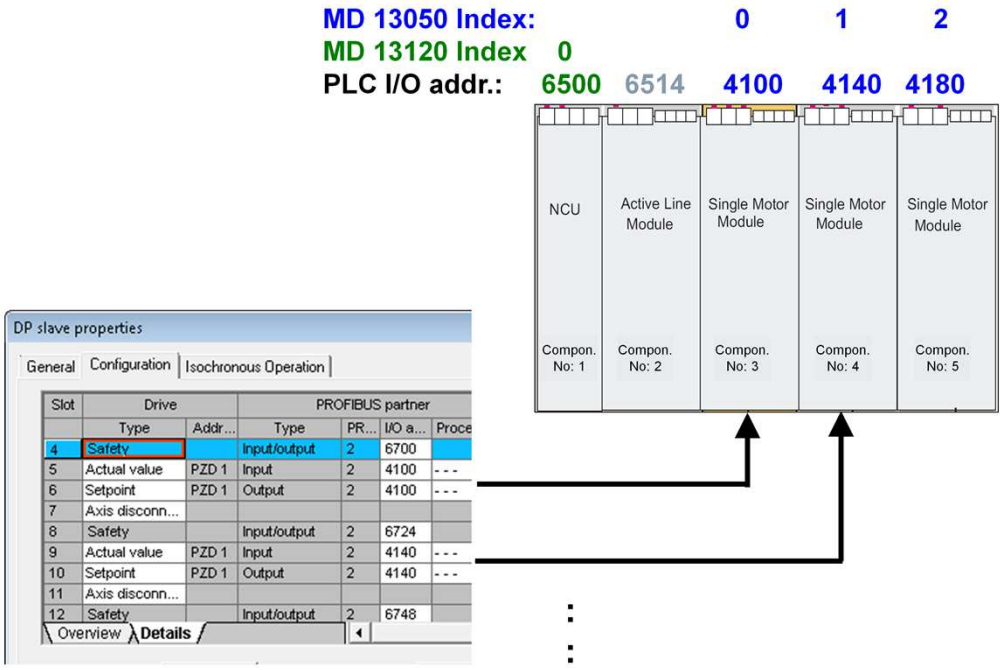
Drives: Start address = 4100, PZD 15/19 -> increment 40 bytes
-> Addresses = 4100, 4140, 4180, ...

These addresses correspond to the default setting of MD 13060

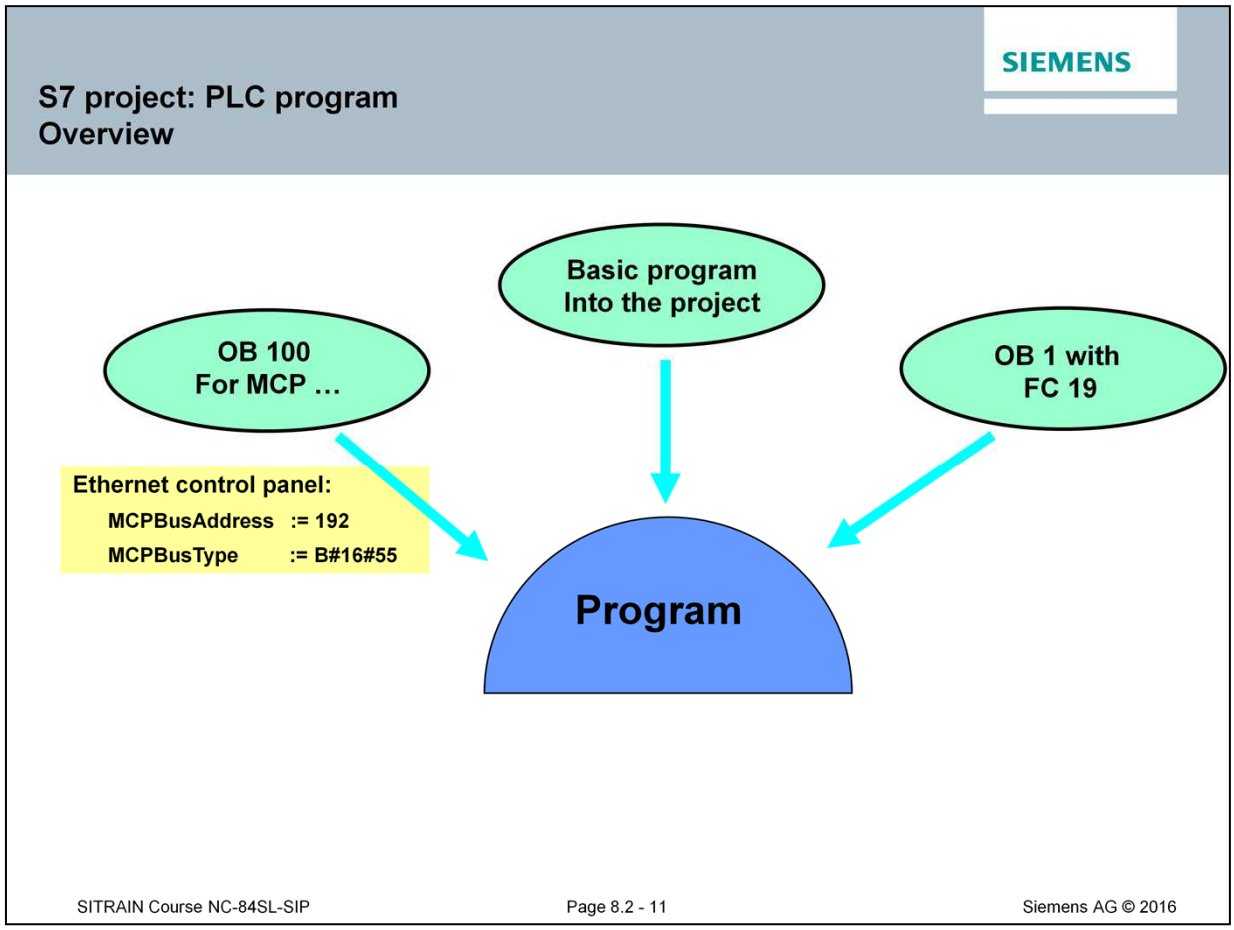
Safety: Start address = 6700, 12 words -> 24 bytes
-> Addresses = 6700, 6724, 6748,

Why control? The telegram types and I/O addresses must be specified in NC MD and in SINAMICS parameters.

S7 project: HW Config
PLC I/O address - Drive module



As well as the assignment of a PLC I/O address to a drive (this assignment is not fixed and can be changed), a setting must be made indicating which axis data set is used for which Motor Module. For this purpose, the logical drive number is set to point to the corresponding index of data item 13050.



PLC program

The basic program serves as the starting point for the PLC program.

Basic program

The basic program is the block library of the Toolbox.
It is copied to the project.

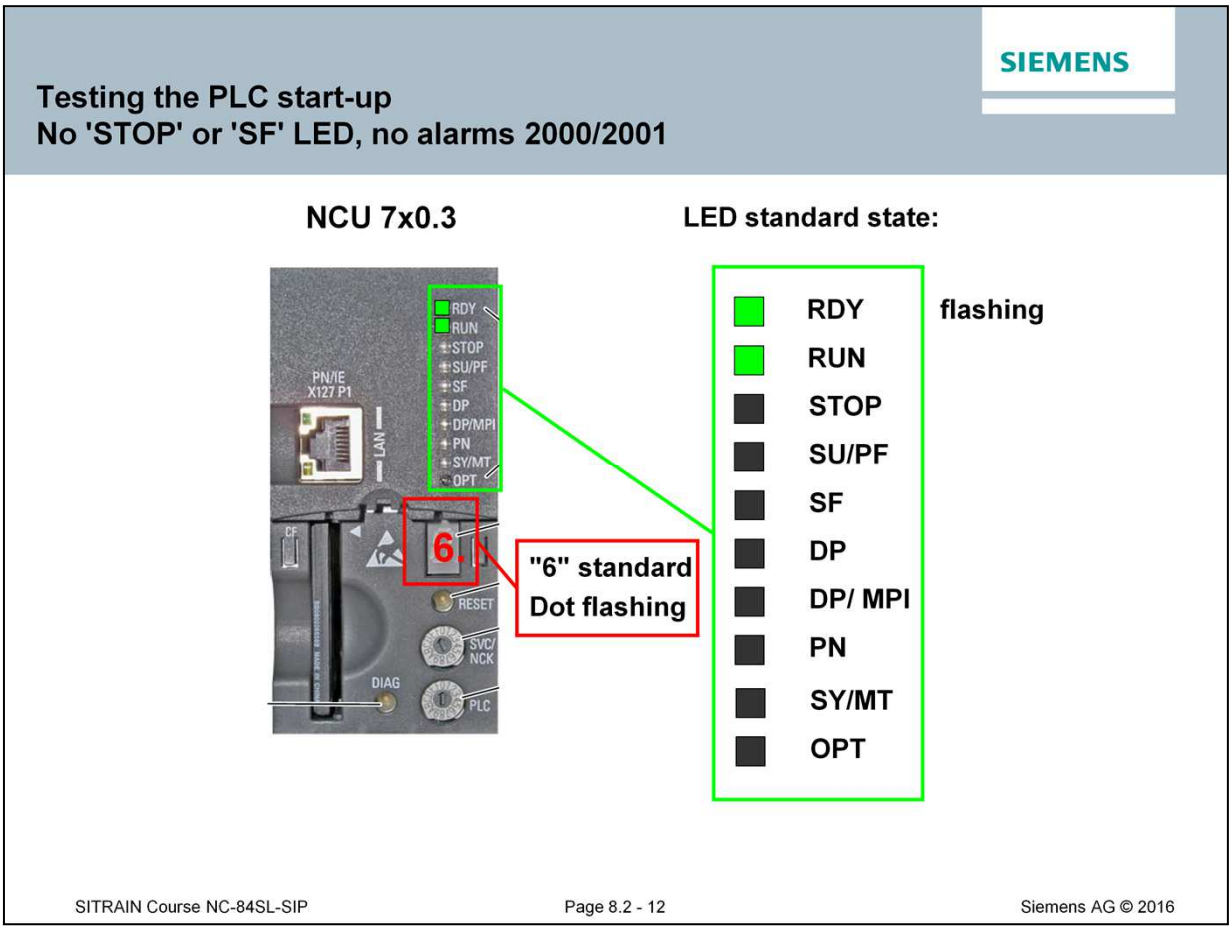
OB 100

OB 100 is the commissioning OB. Basic settings for the MCP, for example, are made in its FB1 call.

OB 1

OB 1 is the cyclic block. The functions programmed here include:

- FC 19 or similar for MCP
- FC 10 for alarms/messages
- Enables/interlocks



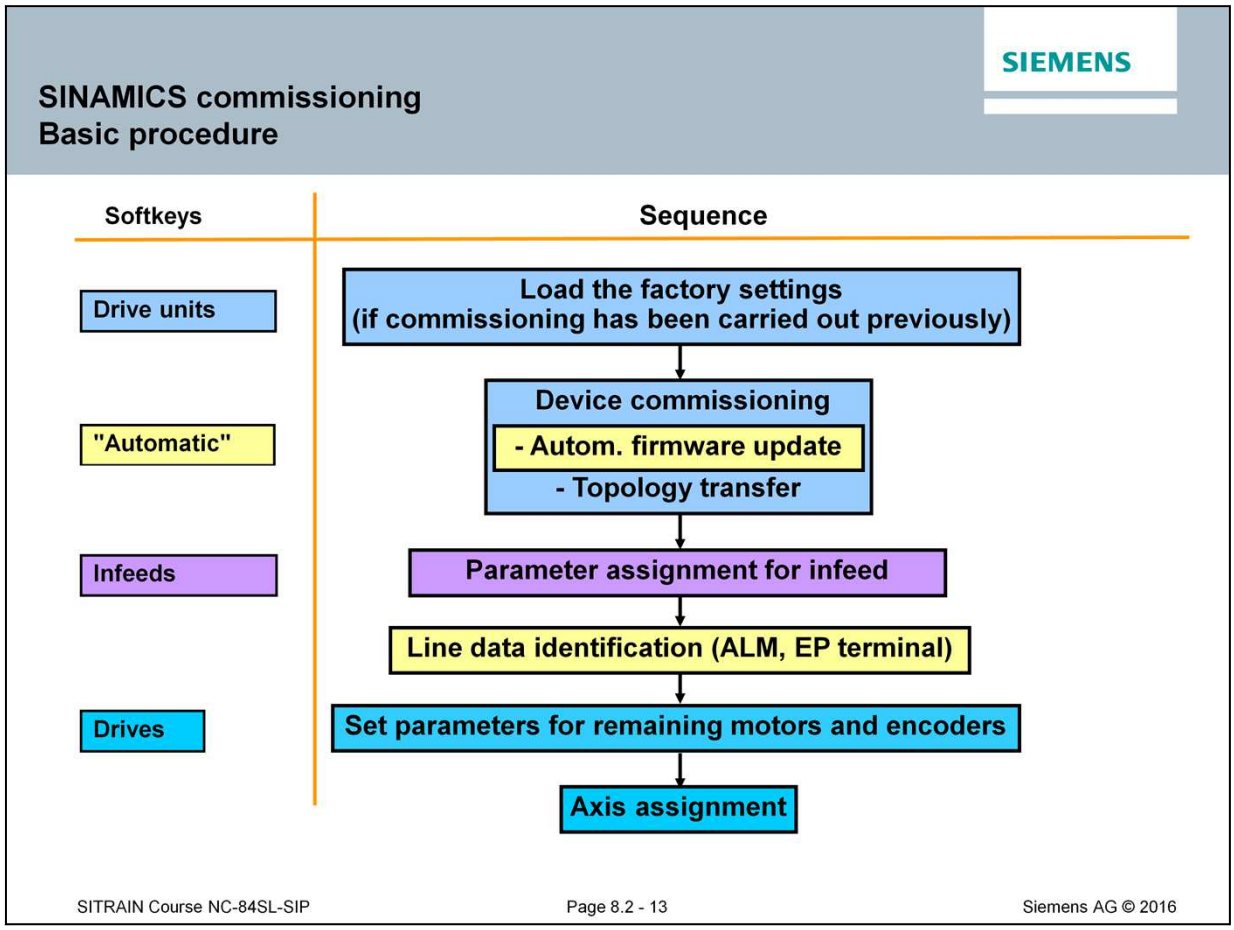
Check PLC start-up: Check the following points after start-up:

- **No PLC STOP or errors:**
  - LED 'RUN' and 'RDY' green
  - LED 'STOP' off (if yellow: Rotary switch 'PLC' to 0? check diagnostic buffer)
  - LED 'SF' off (if red: check PB/PN hardware)
  - LED 'DP' off (if red: check PB connection)
  - LED 'DP/MPI' off "
  - LED 'PN' off (if red: check PN hardware)
  - LED 'OPT' off (if red: check PN connection; (if red and flashing: PN device failed?)
- **No alarms:**
  - 2000 'sign-of-life monitoring PLC'
  - 2001 'PLC start-up failed'
- **MCP running** (no LEDs flashing)

Diagnostics of SINAMICS

To view corresponding SINAMICS alarms in the HMI, set

- **MD 13150= 3F3F h** (general MD)



SINAMICS commissioning sequence

Essentially, the SINAMICS commissioning sequence comprises the following steps:

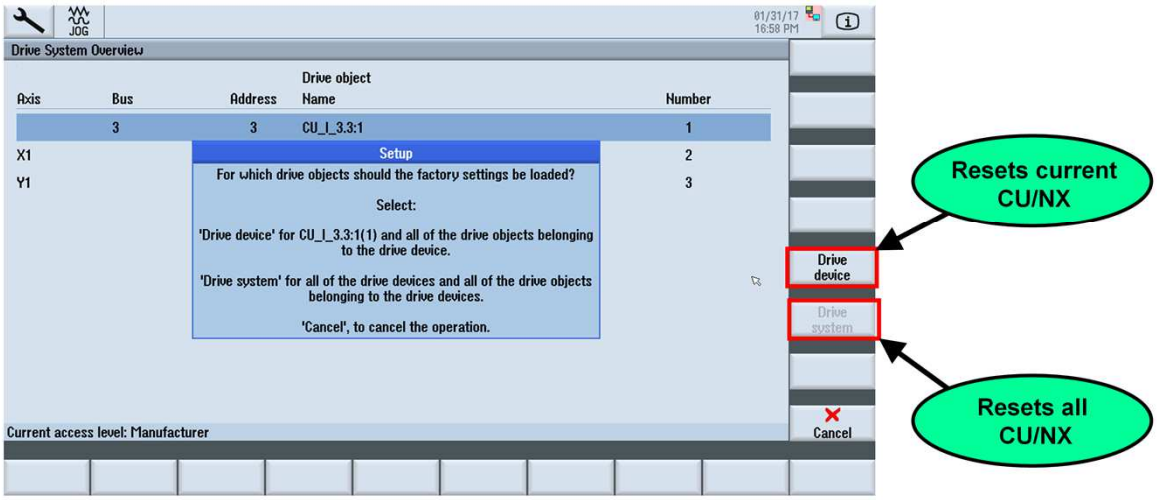
- **Load factory settings (if commissioning has been carried out previously)**
- **Device commissioning (incl. automatic firmware update)**
- **Parameter assignment of infeed**
  (not required if infeed without DRIVE- CLiQ)
- **Set parameters for remaining motors and encoders**
- **Network identification** (not required if supply without DRIVE-CLiQ)
- **Assigning axes**

Further steps

Further actions such as drive optimization cannot be carried out until after axis assignment.

SINAMICS commissioning
Loading the factory settings

SIEMENS



 Siemens AG © 2016

Objective of the factory settings

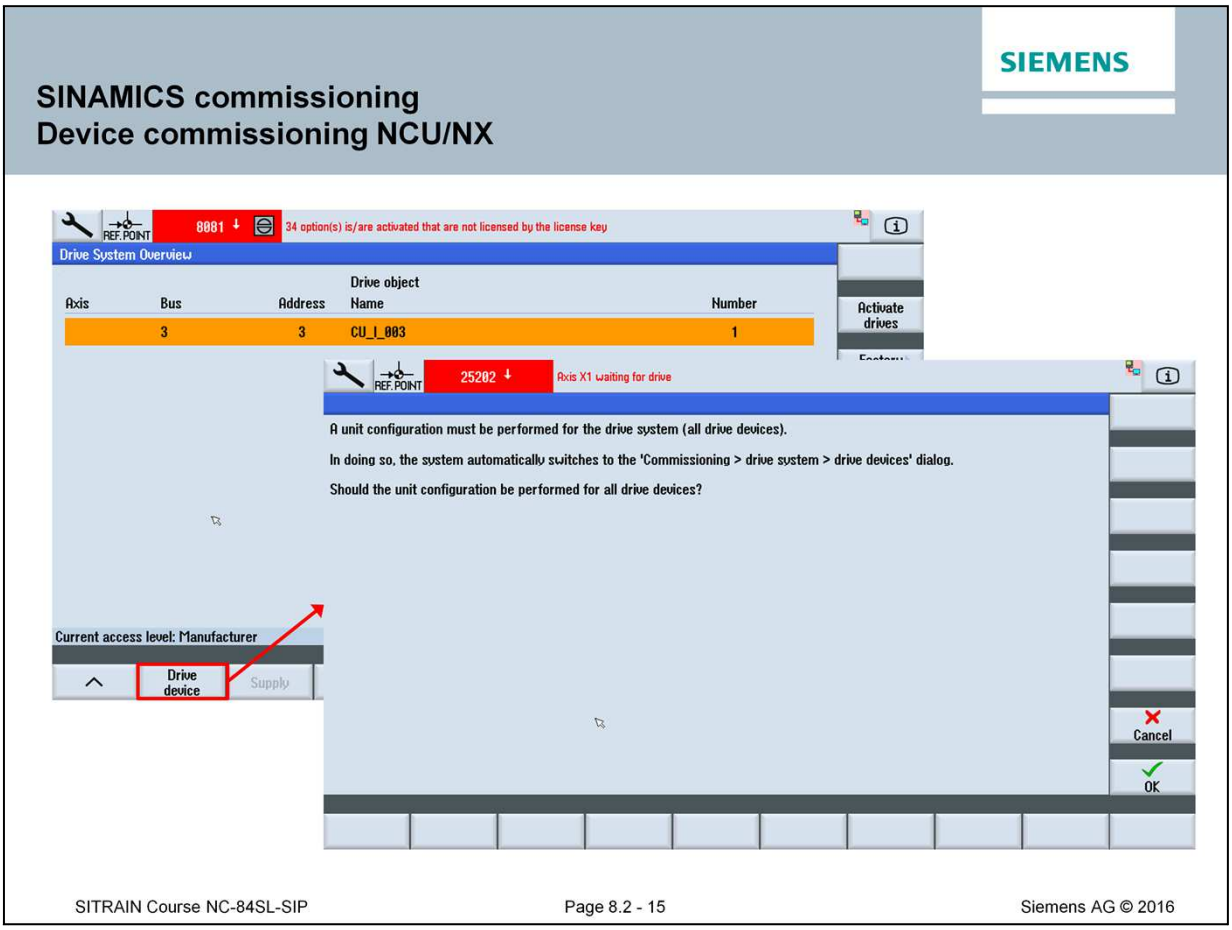
The factory settings are the default data for SINAMICS. They restore the control to a known initial state.

Factory settings

- Drive system This selection causes a complete memory reset of all drive data.
If this point cannot be selected as in the figure above, there is only **one** drive unit; the device and system are therefore identical.

- Drive unit Resets only the data of the currently selected CU and its servos (and the Line Module).

- **Current DO** Resets only the data of the currently selected drive object (DO).



Purpose of device commissioning = Automatic device configuration

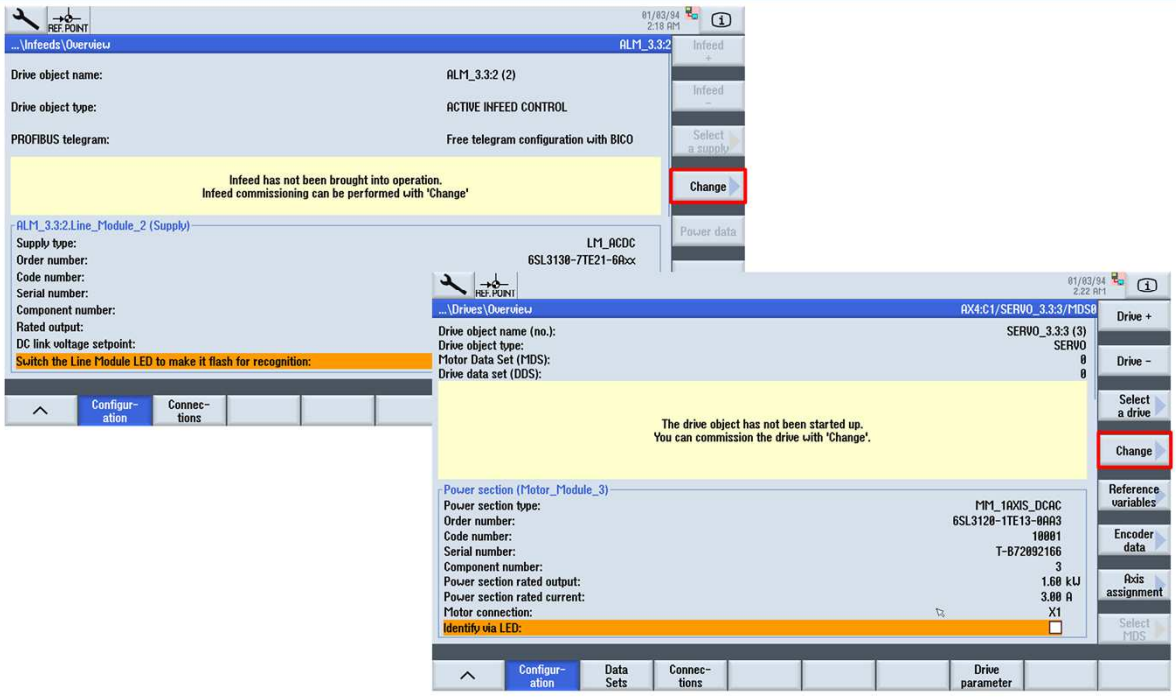
The device commissioning of the SINAMICS drives permits the following:

- Transfer of the actual topology to the drive target topology
  (i.e. to the SINAMICS parameters of the central CU and NX modules)
  - With the transfer of the topology, all components connected on the DRIVE-CLiQ are detected and the drive-internal data traffic is initialized.
- Drive object assignment for the PROFIBUS connection.
  - The PROFIBUS connection via the relevant telegrams has been specified in the HW Config.
- Drive object assignment for drive parameters (p978).

Requirement Establish the factory setting

Sequence of operator actions

1. The cursor is positioned in the **"Drive system"** menu.
2. Press the **"Drive units"** softkey.
   - The HMI detects that no device commissioning settings or data have been transferred yet.
3. Press the **"OK"** softkey to configure the entire drive system.

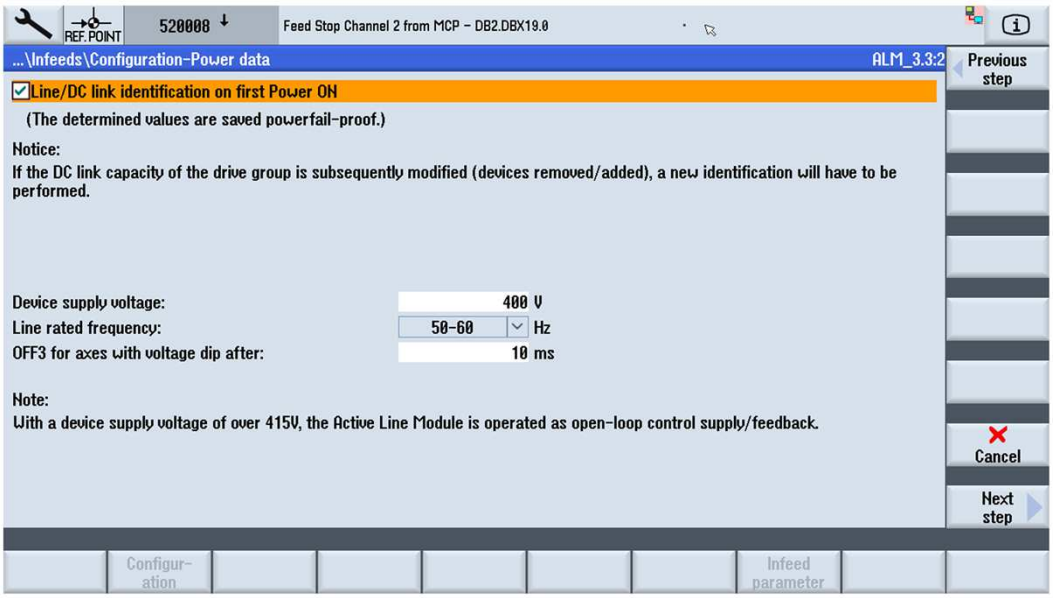


 Siemens AG © 2016

Drive system

From the Drive system overview screen, you go to the data sets of infeeds and drives, where you are prompted to enter any missing data.

The details of the procedure are described in an exercise.



Line identification

During automatic drive commissioning (topology detection), a line supply with DRIVE-CLiQ connection is detected as the drive object and put into operation.

A message appears informing the user that the process will continue with the infeed or PROFIBUS configuration once device configuration with commissioning is complete.

During configuration of an ALM, the supply voltage, type of interconnection etc. are queried.

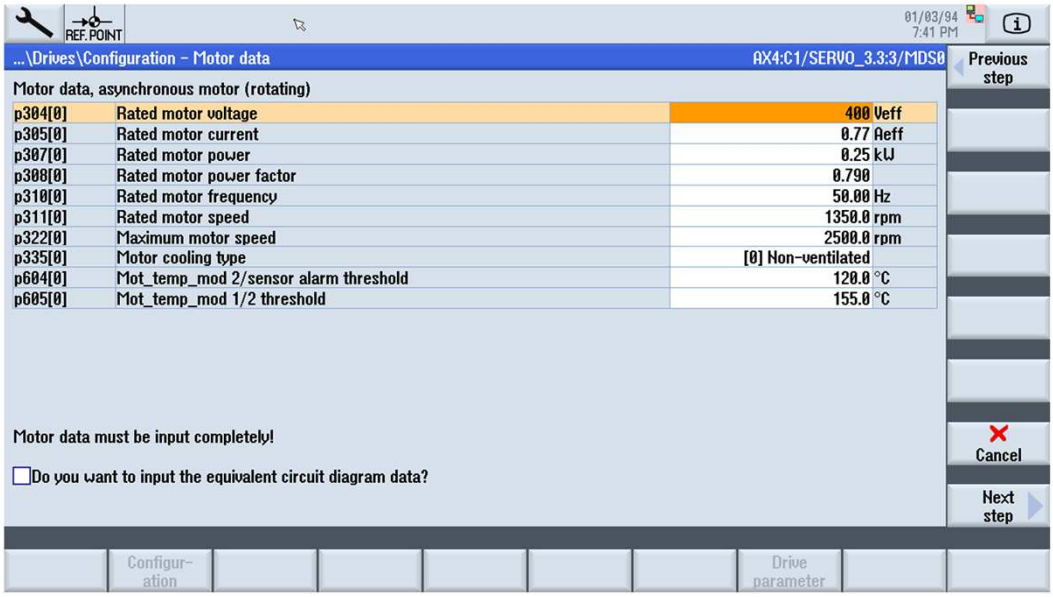
The following line identification serves to determine the properties of the line power supply, characterized by resistance, capacitance and inductance. Their aim is to optimize power recovery to the system.

Line identification starts automatically following commissioning and first-time activation of ON/OFF1 on the ALM, as well as pulse enable (EP terminal).

If line data detection of this nature is required again, you can request it repeatedly via the "Line data" softkey, e.g. when restarting the machine in a different place of installation.
Before a machine is shipped to a destination country where a different supply voltage is used, the line supply data must be set accordingly and line/DC-link identification activated.

Internal SINAMICS line contactor control

This defines a BiCo output and input that switches an external contactor or receives a feedback signal about the switching. A missing feedback signal produces an error message.



Commissioning Wizard

This tool for drive commissioning is integrated into the user interface and is used if not all components have been automatically detected or if changes are to be made.

Start

To start the Commissioning Wizard, proceed as follows:

> OA: 'Setup' > Softkey: 'Drive system'

> Softkey: 'Drive' > Softkey: 'Change'

Functions

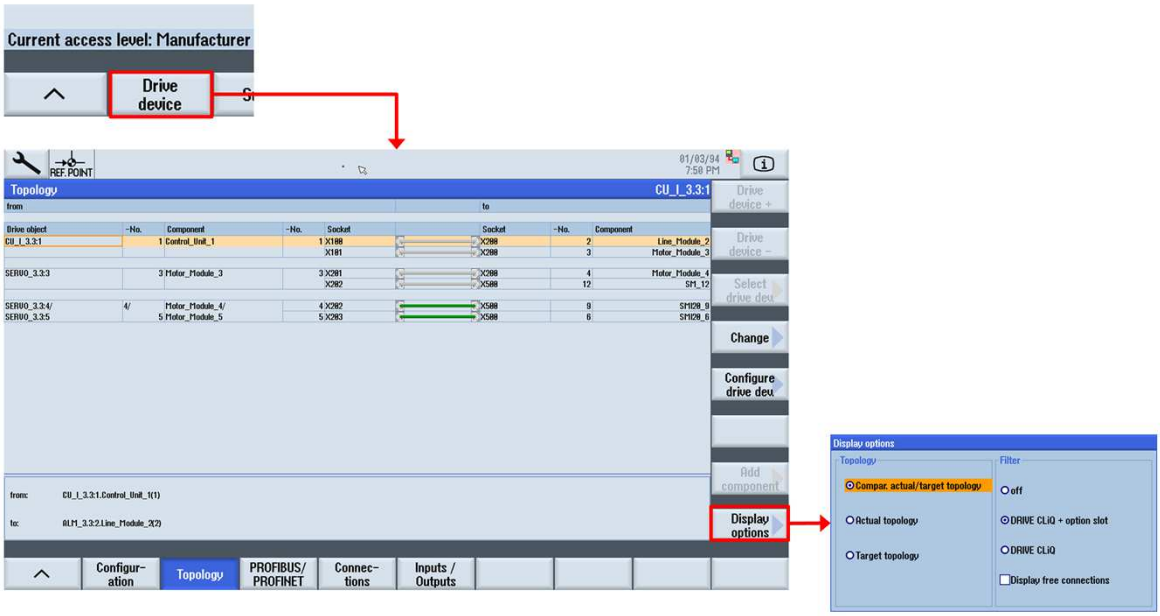
The following functions are available:

- Select the function modules
- Select the motor type; assign motor parameters (select from list and/or manual input)
- Activate brake control
- Input data for the equivalent circuit diagram
- Assign encoders
- Assign encoder parameters (select from list and/or manual input)
- Select the controller mode (e.g. speed controller with encoder) and the telegram.

End

At the end of the sequence (which is described in detail in an exercise), save the data with softkey:

> Softkey: 'Completed' > Softkey: 'Yes'



Show configuration

The following operating sequence displays the configuration, i.e. the list of detected and configured components:

> OA: 'Setup' > Softkey: 'Drive system' -> Softkey: 'Drive unit'

Display options

The display in both screens also depends on the

> Softkey: Display options

A meaningful selection is

- **Topology = compar. actual/target topology**
- **Filter = DRIVE-CLiQ + option slot**

Comparison

Now you can check that:

- All intended DRIVE-CLiQ interconnections are present
- All the DRIVE-CLiQ interconnections are correct

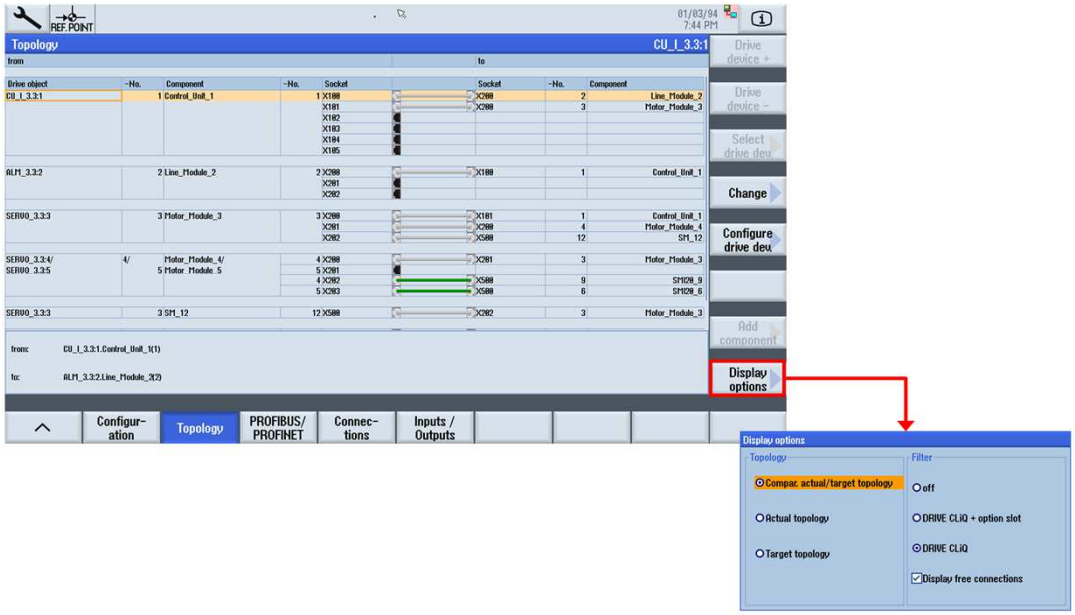
The errors are highlighted in red (missing in target topology) or gray (missing in actual topology).

Motors, encoders

To display these objects too, set

- **Filter = off**

SINAMICS commissioning: Topology - All components



Display options

- **Compar. actual/target topology**
- **Filter = off**
- **Display free connections = no**

Components shown gray in the actual/target topology comparison are missing or are not connected via DRIVE-CLiQ, as in the screenshot above.

Explanation

- **Control_Unit_1** CU on the NCU
- **ALM_3_3_2** Active Line Module on port X100
- **Motor_Module_3** Motor Module = MoMo= drive on CU, port X101

The appended numbers are component numbers.

- **SM_12** Sensor Module on port X202 of the MoMo 3
- **Motor_14** Motor at the Power Output 1 = PO 1 of the MoMo 3
- **Encoder_13** Encoder on the SM_12, connection AN1 (analog)

Therefore:

- **Motor_SMI_11** is the name of SMI motor, component no. 11 (motor data on SMI_9), for example

2-axis module

Ensure that the DRIVE-CLiQ ports and motor connections are permanently assigned on the Double Motor Module:

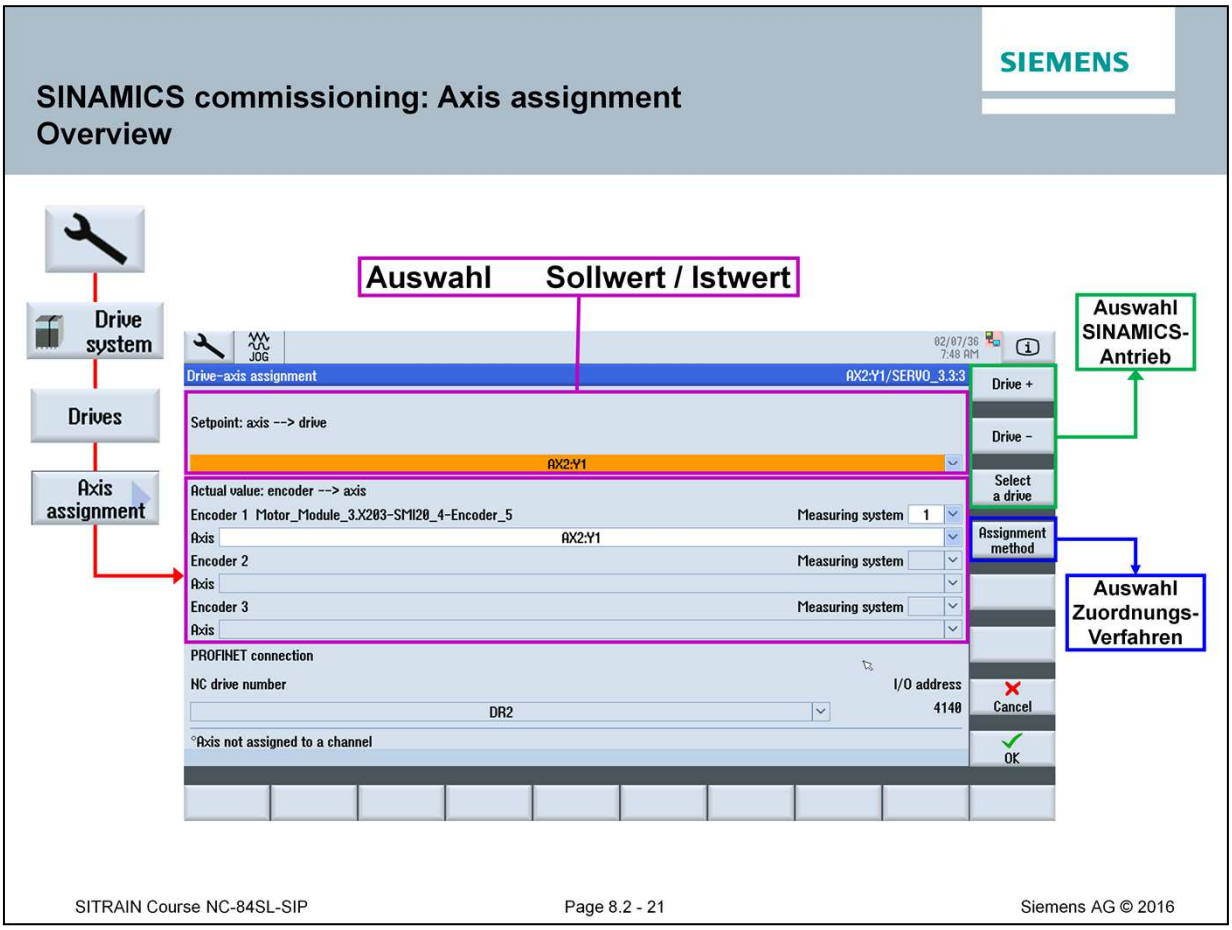
	Motor encoder port	**Motor connection**	
1st drive	**X202**	**X1, at front:**	**"PO1"**
2nd drive	**X203**	**X2, at rear:**	**"PO2"**

The numbering depends on the sequence on the DRIVE-CLiQ bus.

Notice

The numbers assigned to the components have various meanings; the **last number** is the **component number**, this can be preceded by the *component type*; examples:

Designation	Type	Component number	
*SMI20*_**6**	SMI20	**6**	(of the SMI)
*Motor_SMI*_**11**	SMI motor	**11**	(of the motor)



Assign axis

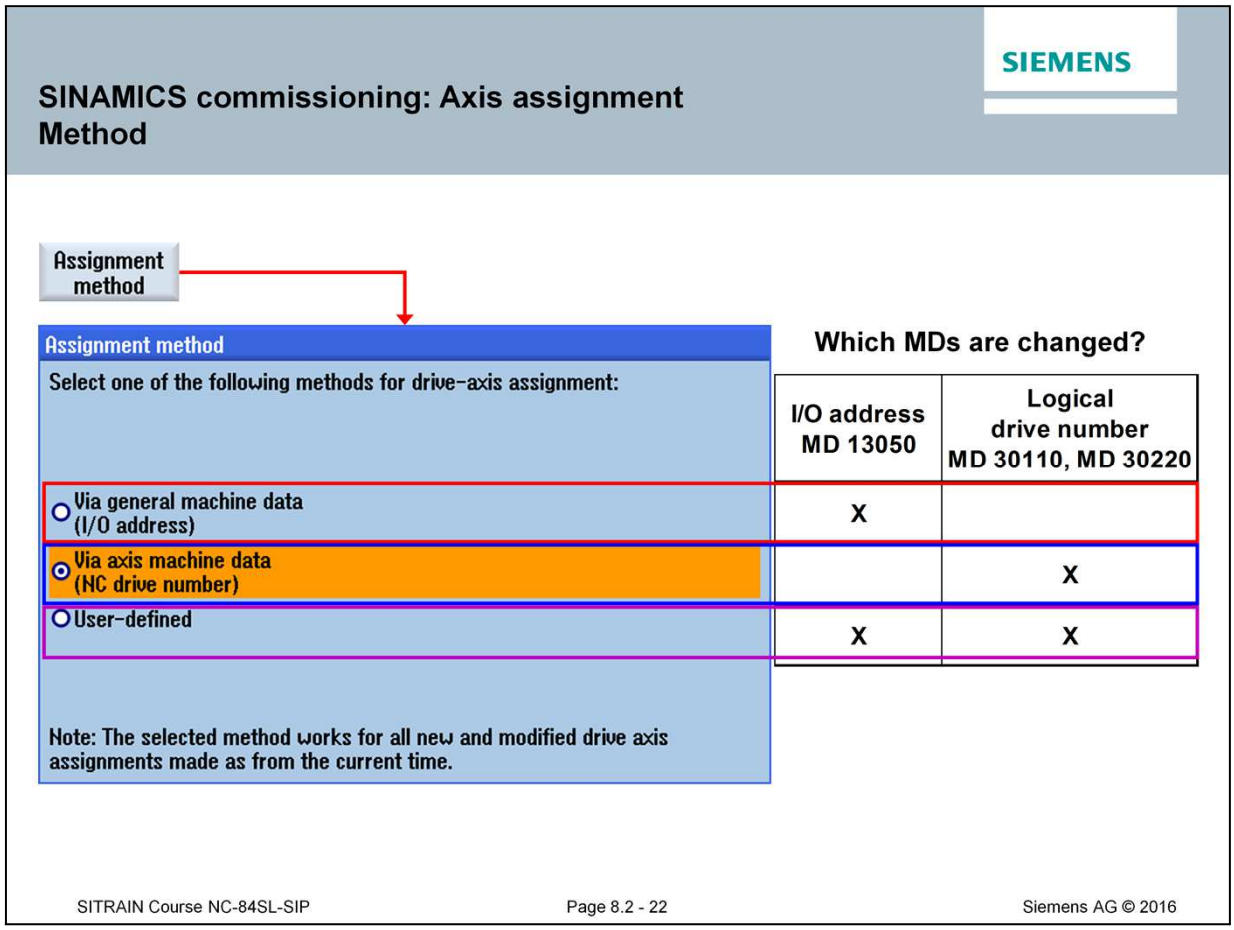
In the commissioning tool, there is an 'Assign axis' softkey.
This function is used to assign a DO to an axis defined via general and channel-specific MDs.

Operator action

- Menu selection:
  > **OA: 'Setup' > Softkey: 'Drive system'**
  > **Softkey: 'Drive' > Softkey: 'Axis assignment'**
- Select the drive with the softkey:
  > **Softkey: 'Drive +' / 'Drive –'**
- Select an axis in the list boxes
  > **'Setpoint axis --> Drive'**

  and
  > **'Actual value encoder --> Axis'**

Displayed on the following pages: Select axis, encoder, and PROFIBUS connection.



Assignment method

The following methods are available:

- **Via general machine data (I/O address)** — That is the standard; it changes MD 13050
- **Via axis machine data (NC drive number)** — That is simpler to control; it changes MD 30110 and MD 30220

  This logical drive number then corresponds to the slot number:

  NCU No. 1 – 6

  1. NX No. 7 – 12

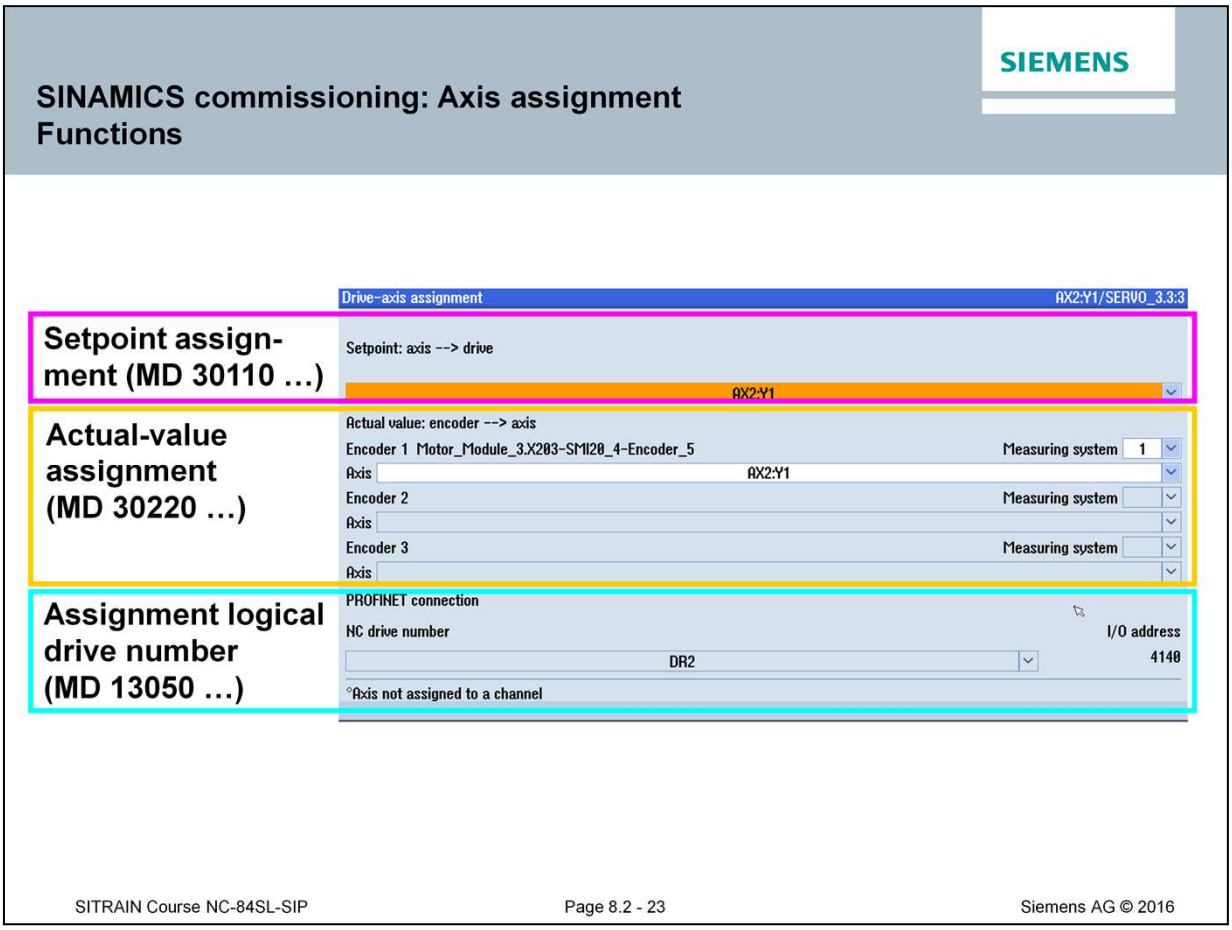
  etc.
- **User-defined** — This permits free assignment of the logical drive number;

  it changes all three MDs: 13050, 30110, 30220

Of course, further MDs are changed in all cases (number of pulses, encoder, etc)

Notes

Once you have freely assigned logical drives numbers with the selection 'user-defined', these changed numbers are retained (even if the variant 'Gen. MD' or 'Axis MD' is selected afterward)!

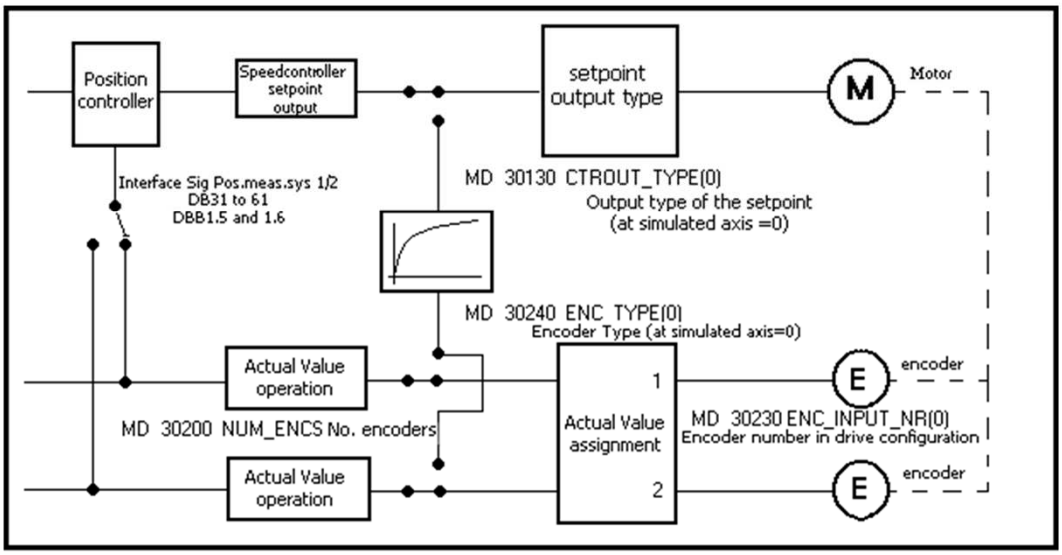


Functions

The following can be selected for each drive:

- **Setpoint:** Selection of the SINUMERIK axis.
- **Actual value:** Selection of the SINUMERIK encoder
- **PROFINET connection:** Selection of the logical drive number **n** with **DRn** (n=1, .., 31) (only with 'user-defined' method)

Setpoint and actual value configuration



These data are automatically correctly preassigned by the function "Assign axis".

Setpoint assignment MD 30110 CTRLOUT_MODULE_NR Logical drive number (setpoint).
Used as a pointer to the I/O address table in MD13050.
If MD 13050 has not been resorted, this will be the same as the slot no., e.g.:
- 6 drives after the CU or CU/ALM: 1 ... 6
- 3 drives of a NX10, then: 7 ... 9

Setpoint type MD 30130 CTRLOUT_TYPE
1 Output of setpoint
0 Simulation

Number of measuring systems MD 30200 NUM_ENCS
1 One position measuring system
2 Two position measuring systems
Measuring system 1 or 2 is selected with NST DB31.DBX1.5 / 1.6

Index[n] The measuring system machine data has an index [0] or [1].
[0] Values for the 1st measuring system
[1] Values for the 2nd measuring system

Actual value assignment
MD 30220 ENC_MODULE_NR[n] Logical drive number (actual value).
Numbering: See MD 30110!

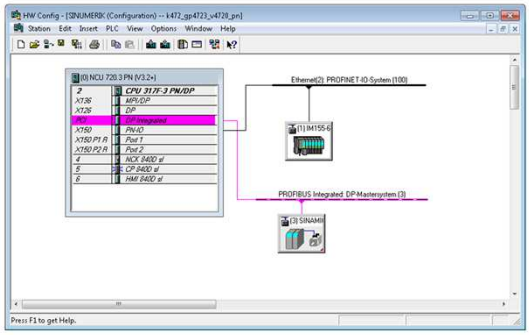
Actual value input MD 30230 ENC_INPUT_NR[0] SINAMICS encoder number
(values 1/2/3 mean first/second/third SINAMICS encoder)

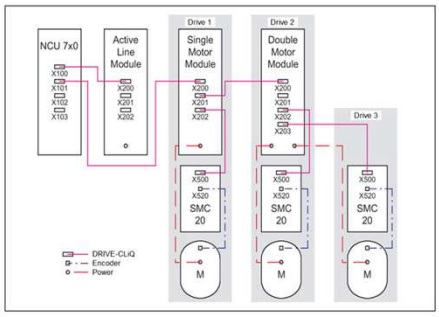
Measuring system type MD 30240 ENC_TYPE[n]
1 Incremental measuring system (and MD34200=1)
4 Absolute measuring system (and MD34200=0)
0 Simulation

Polarity actual value MD 32110 ENC_FEEDBACK_POL [n]
0/1 Default value
-1 Changing the polarity

Direction reversal MD 32100 AX_MOTION_DIR
0/1 Default value
-1 Direction reversal




HW Config - [SINUMERIK (Configuration) -- k472_gp4723_v4728_pn]
Station Edit Insert PLC View Options Window Help
(0) NCU 720.3 PN (V3.2+)
CPU 317F-3 PN/DP
MPI/DP
DP
PN-IO
Port 1
Port 2
NCK 840D sl
CP 840D sl
HMI 840D sl
Ethernet(2): PROFINET-IO-System (100)
(1) IM155-6
PROFIBUS: Integrated_DP-Mastersystem (3)
(3) SINAMI
Press F1 to get Help.


NCU 7x0
X100
X101
X102
X103
Active Line Module
X200
X201
X202
Drive 1
Single Motor Module
X200
X201
X202
Drive 2
Double Motor Module
X200
X201
X202
Drive 3
X500
X520
SMC 20
M
DRIVE-CLiQ
Encoder
Power

Notes

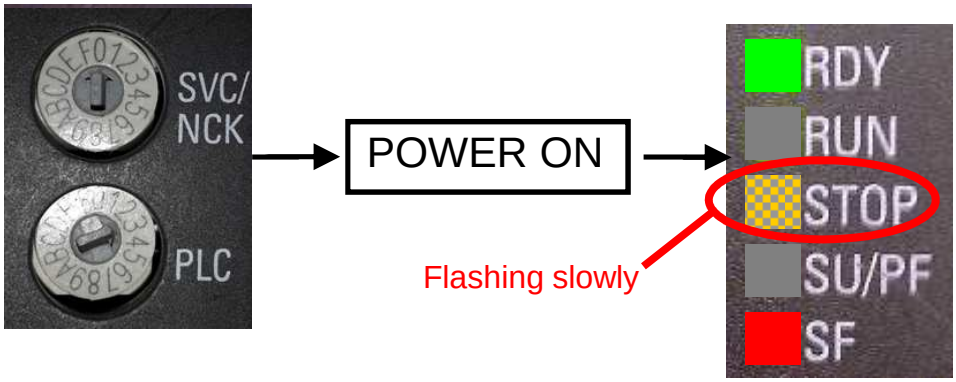
Exercise: PLC memory reset.

PLC memory reset

Turn the PLC rotary switch to position "3."

POWER ON (switch control off and back on again).

-> The 'STOP' LED on the PLC flashes slowly (memory reset request detected).



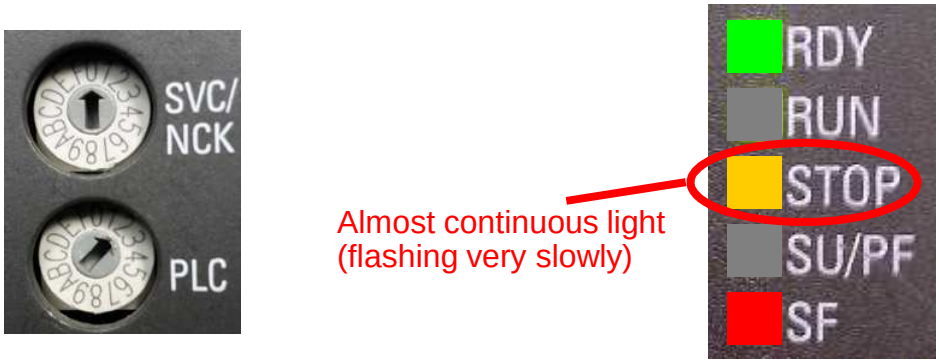
Turn the PLC rotary switch to position "2" and back to position "3".

-> The 'STOP' LED on the PLC flashes quickly several times (memory reset active).



At the end, the switch is set as follows:

- PLC switch in position "2" (STOP). -> The STOP LED is (almost) continuously lit.



The PLC has undergone a memory reset. (All blocks except SFB/SFC, FB15, and DB1 deleted), it is in the stop state (switch PLC in position 2).

Exercise: Creating an S7 project

Objective of the exercises in the PLC section

To communicate with the PLC components, we must put together and load a valid hardware configuration.

In addition, the following is performed in the basic program:

- Basic settings are made in OB 100 at FB 1
- FC19 is called in OB 1 for the MCP
- The axis enables are set in OB 1.

New project

In the SIMATIC Manager, create a new project with

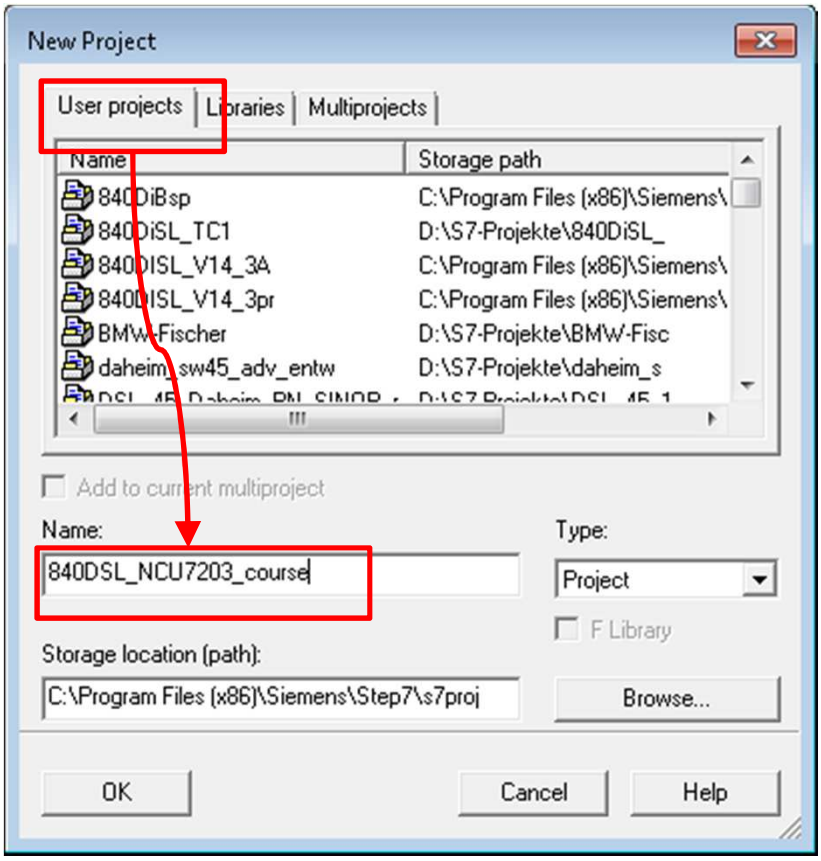
> Menu: 'File' -> 'New'

Project name

Select the tab

- **User projects**

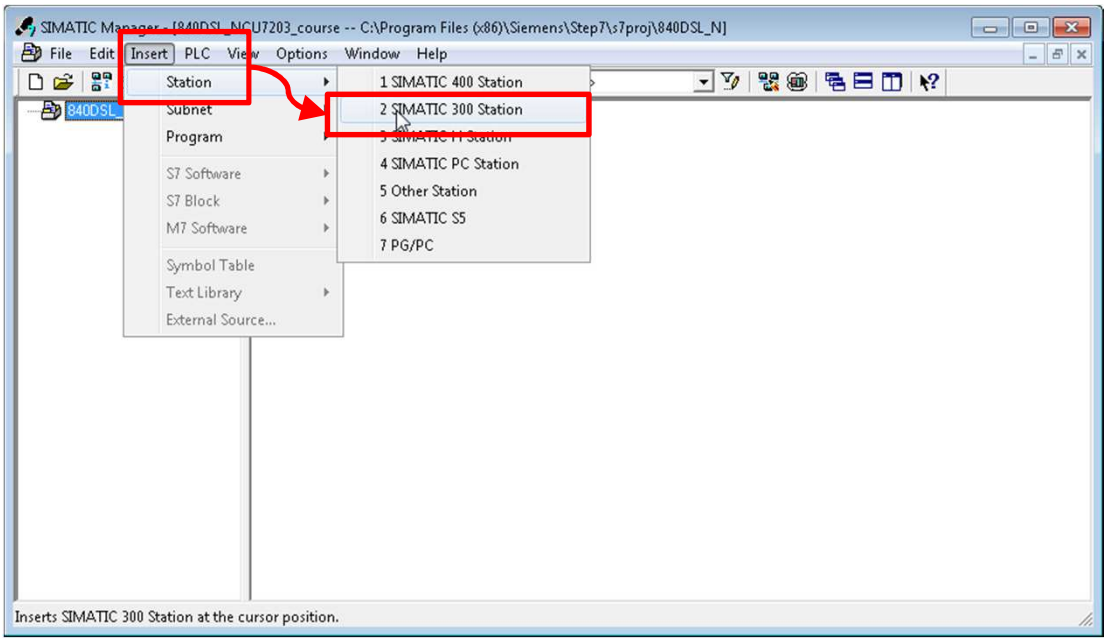
and assign a project name, see screenshot.



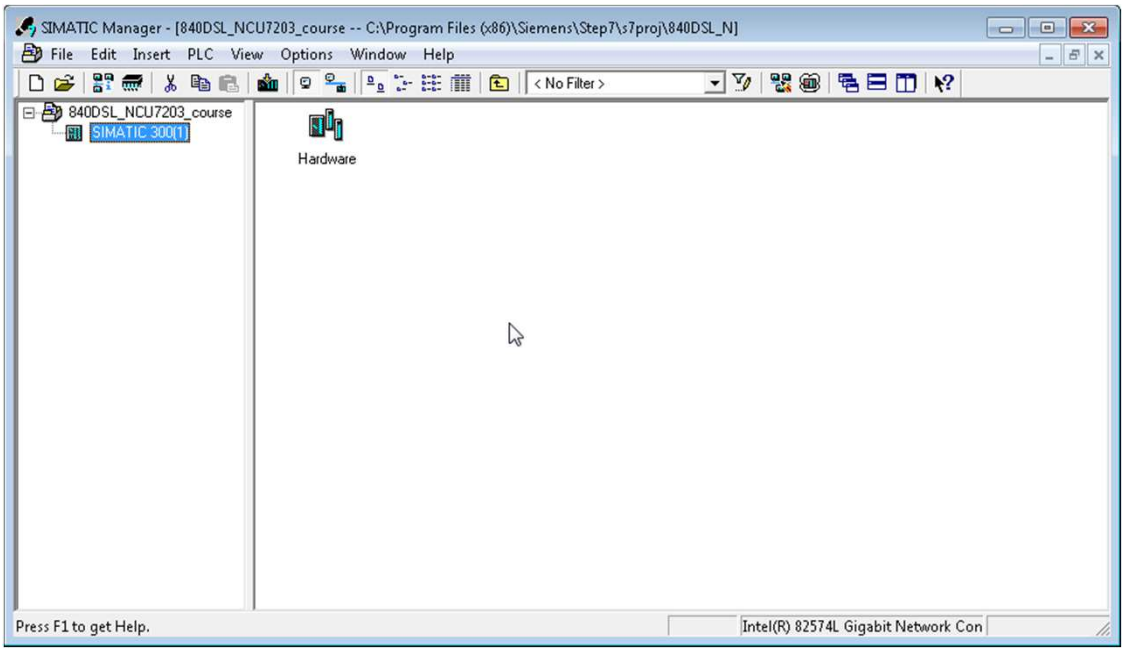
Insert S7-300 station

Insert an S7-300 station by selecting

> Insert > Station > SIMATIC 300 station



Check Your S7 project should now look like this:



Exercises for PLC hardware configuration:
- Depends on I/Os!

SIEMENS

What PLC I/Os are installed in the training rack?



ET200 SP (PROFINET) -> Exercises:
ET200 SP (PROFINET)



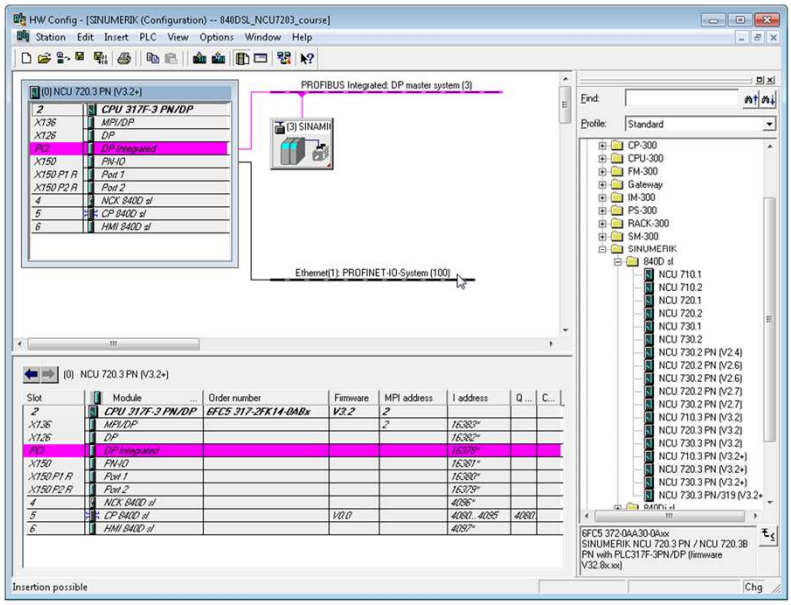
ET200 S (PROFIBUS) -> Exercises:
ET200 S (PROFIBUS)

Siemens AG © 2016

Notes

Exercises: ET200 SP (PROFINET)
Create HW Config with NCU7x0

SIEMENS





Objective

To create a hardware configuration with a suitable NCU.

This activate the PROFINET interfaces to then be able to configure the ET200 SP (PN) I/Os.

Steps

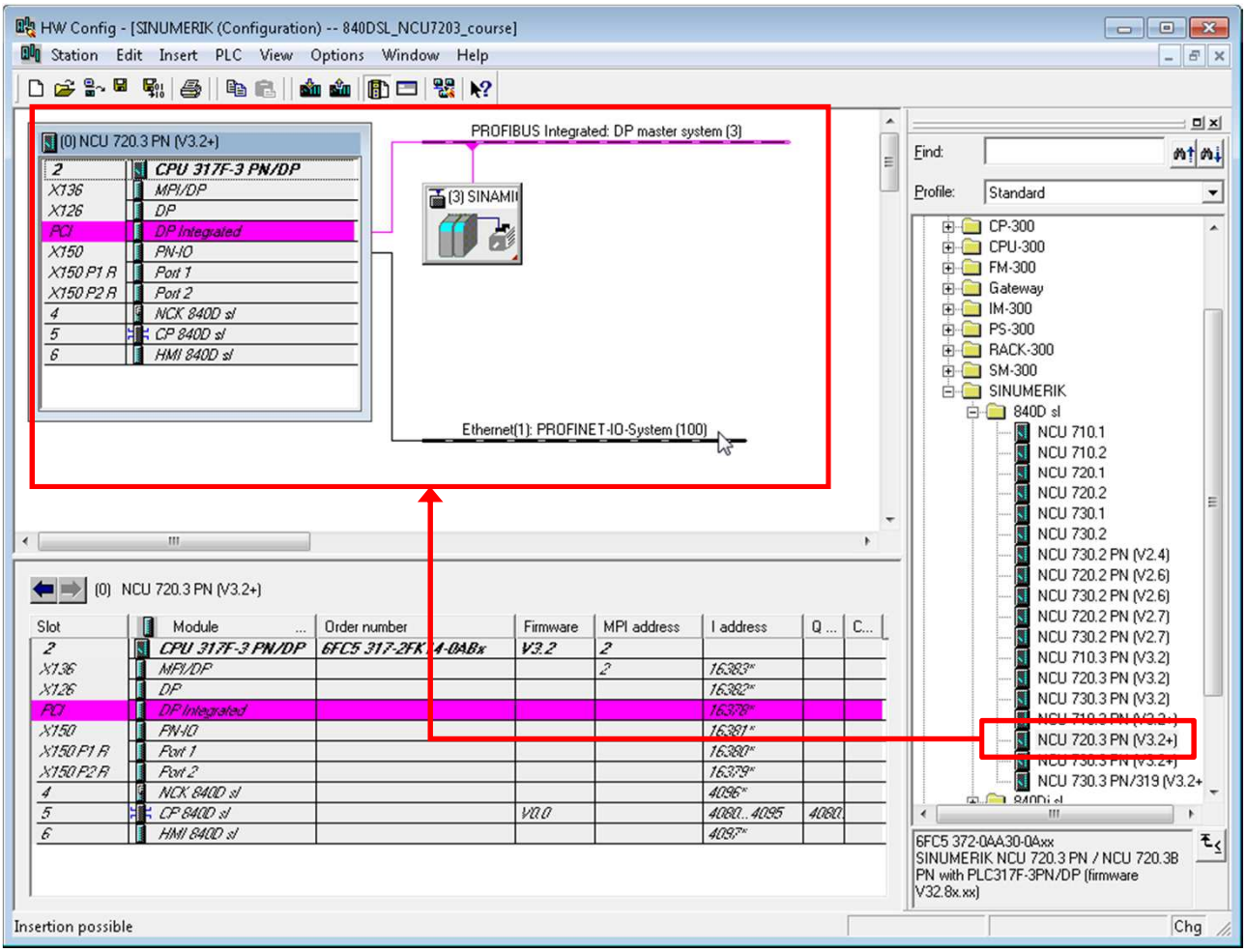
1. Start HW Config
2. Select an NCU
3. Activate PROFINET-IO
4. Leave PROFIBUS inactive
5. Set the IP address of the CP

1. Start HW Config From the SIMATIC Manager, launch the HW Config tool for the station you have just created.

2. Select NCU In the module catalog, select a suitable NCU 7x0 from the path

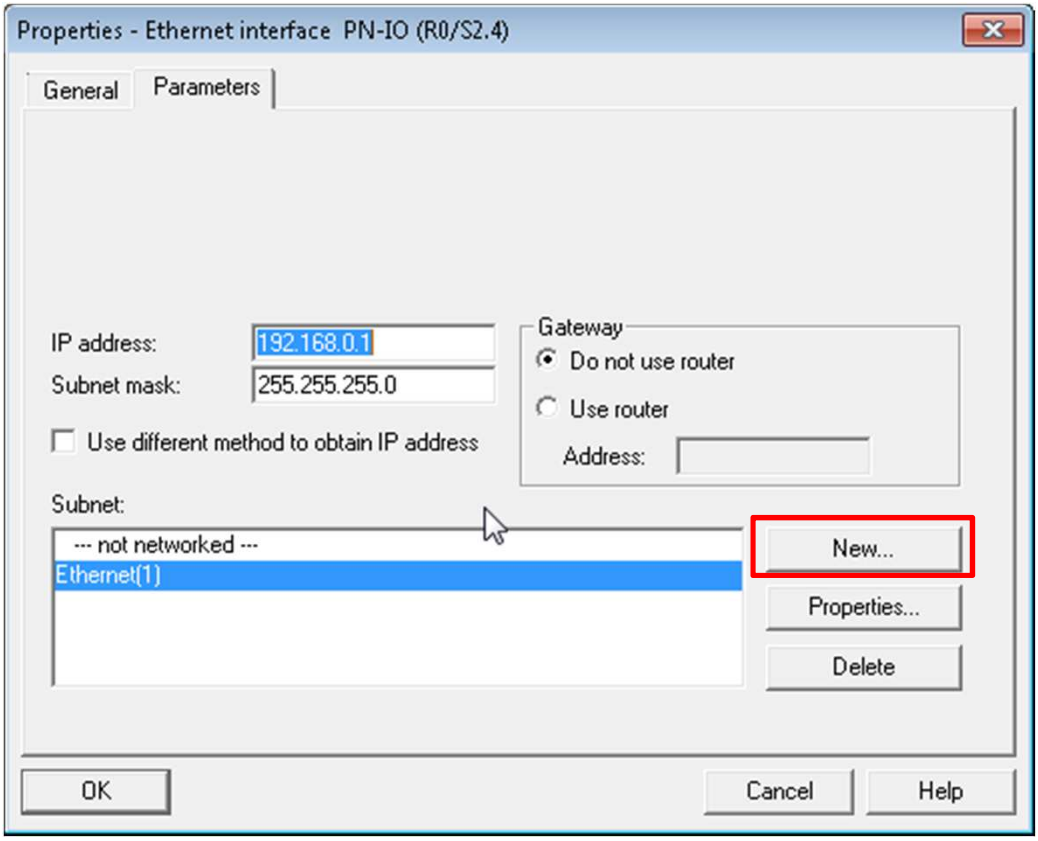
> SIMATIC 300 > SINUMERIK > 840D sl

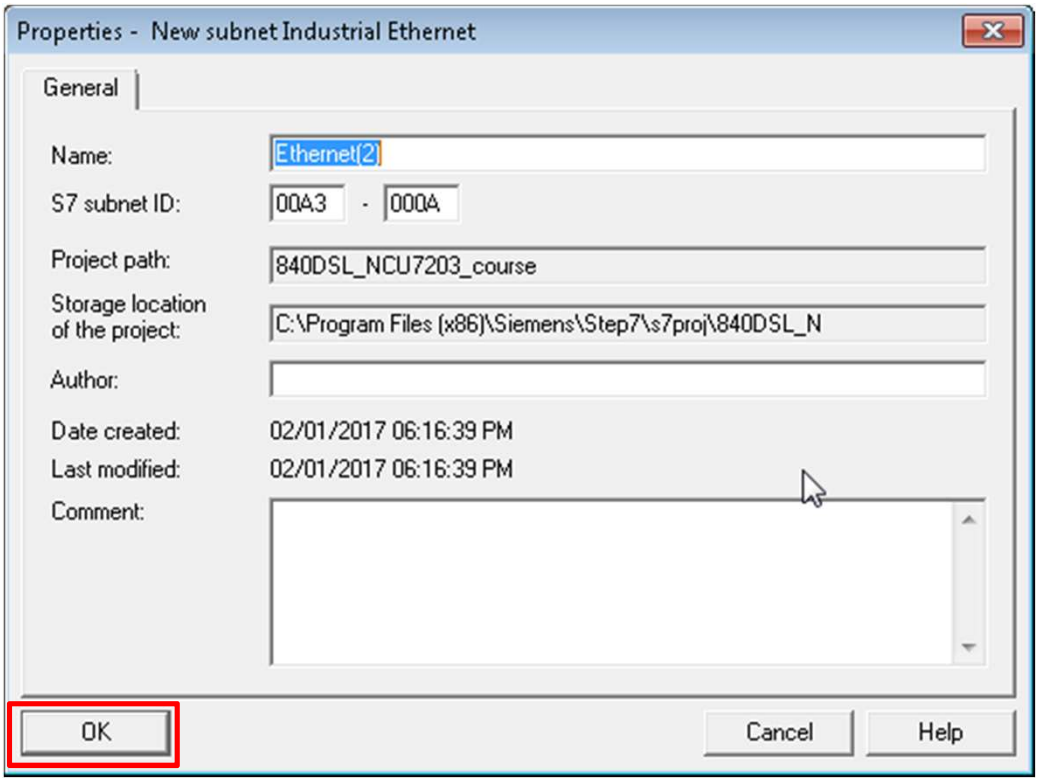
> NCU 720.3 (V3.2+)



3. Network PROFINET

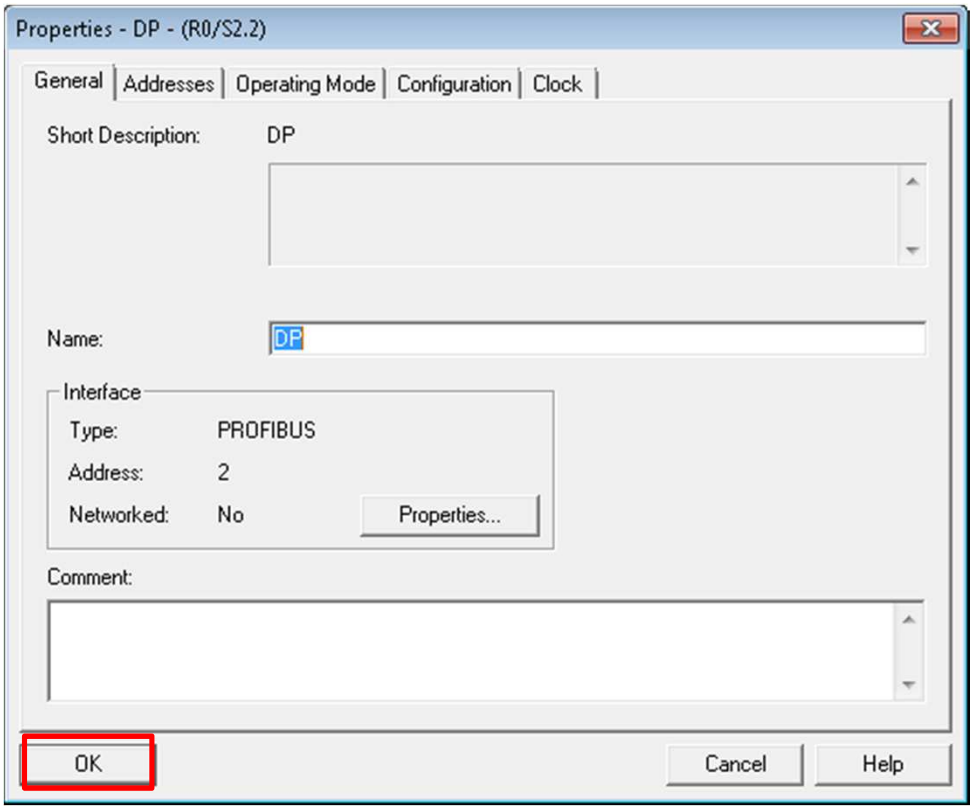
With the 'New' button in the following dialog box for the PROFINET interface 'Ethernet PN-IO', create a new network.



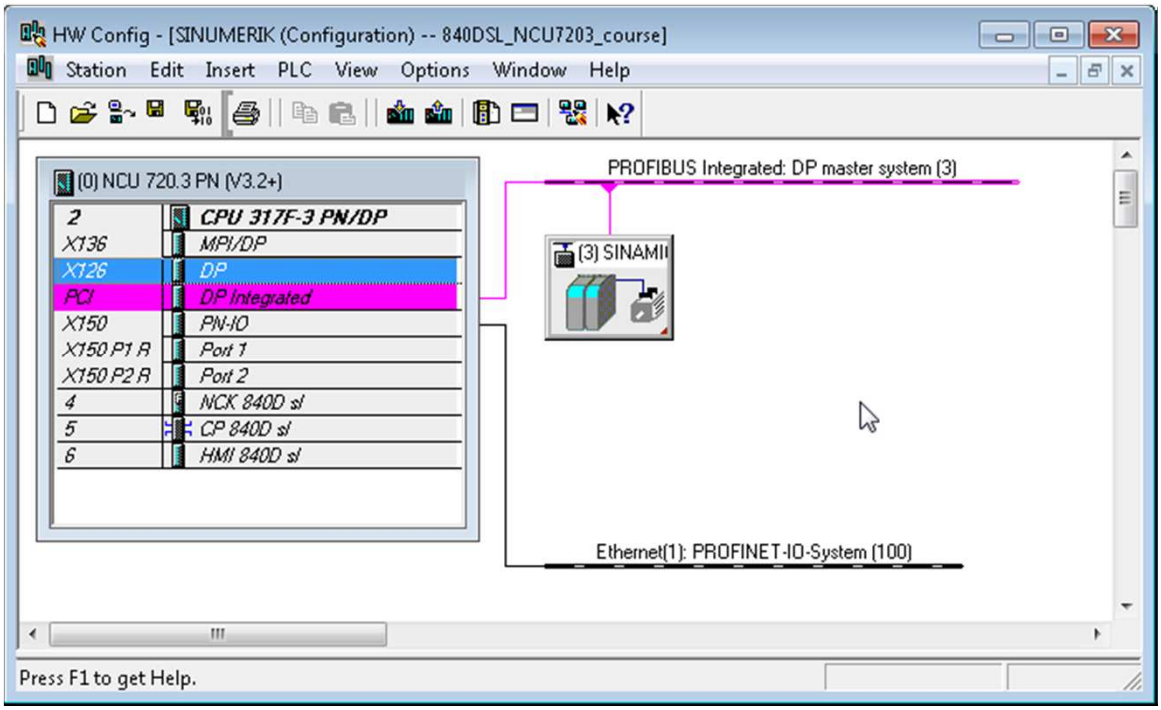


4. PROFIBUS not networked

In the following dialog box for the PROFIBUS interface DP (X126), you can leave this in the 'inactive' status using the **'OK'** button



- Arrange After **'Arrange view' (F4)**, the result should look like this:



5. Set the IP address of the CP

Objective

The programming device is connected to port X120 on the SINUMERIK NCU. Communication is established via an integrated Ethernet CP. This Ethernet CP is assigned an IP address matching the X120 port to enable communication with the PG.

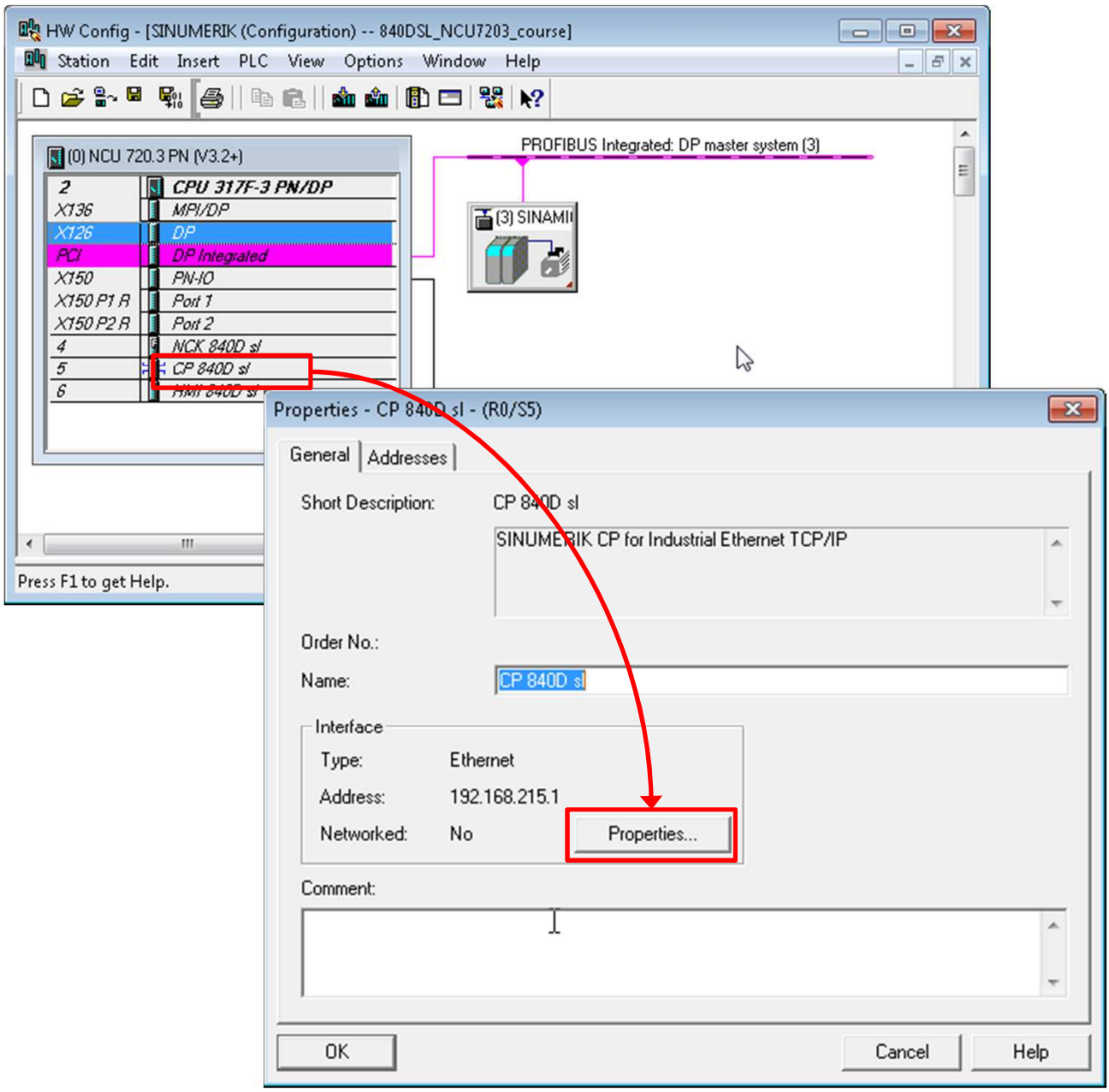
Properties

Double-click the CP 840Dsl to open dialog box

> **Properties – CP 840D**

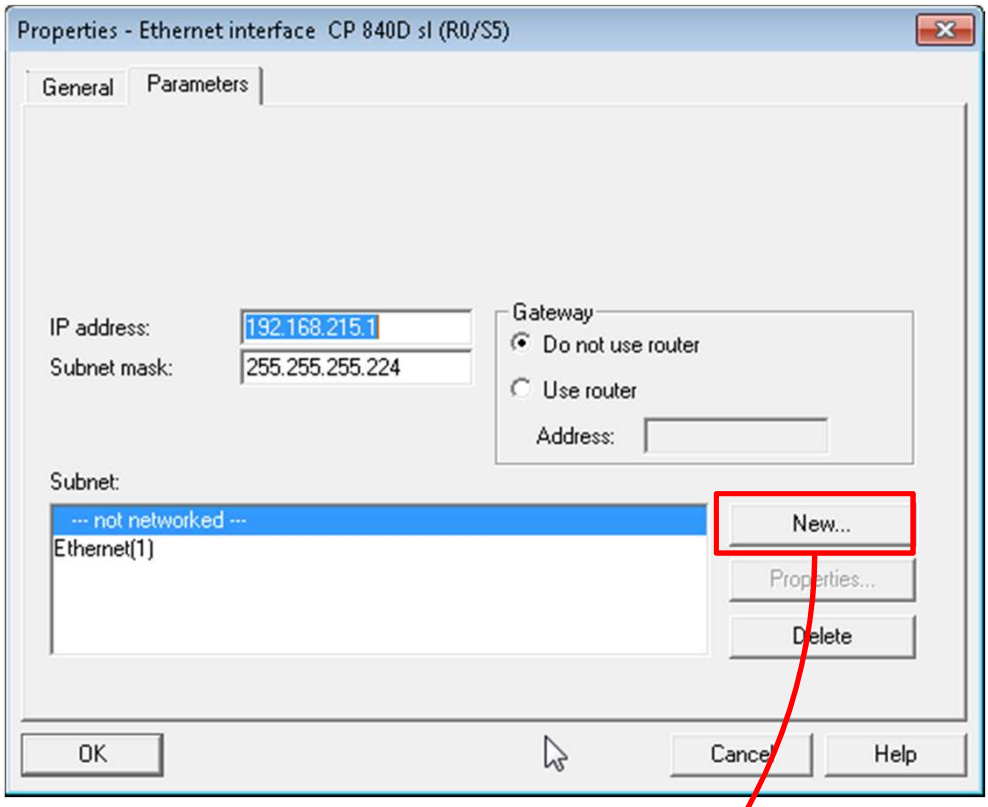
Then click:

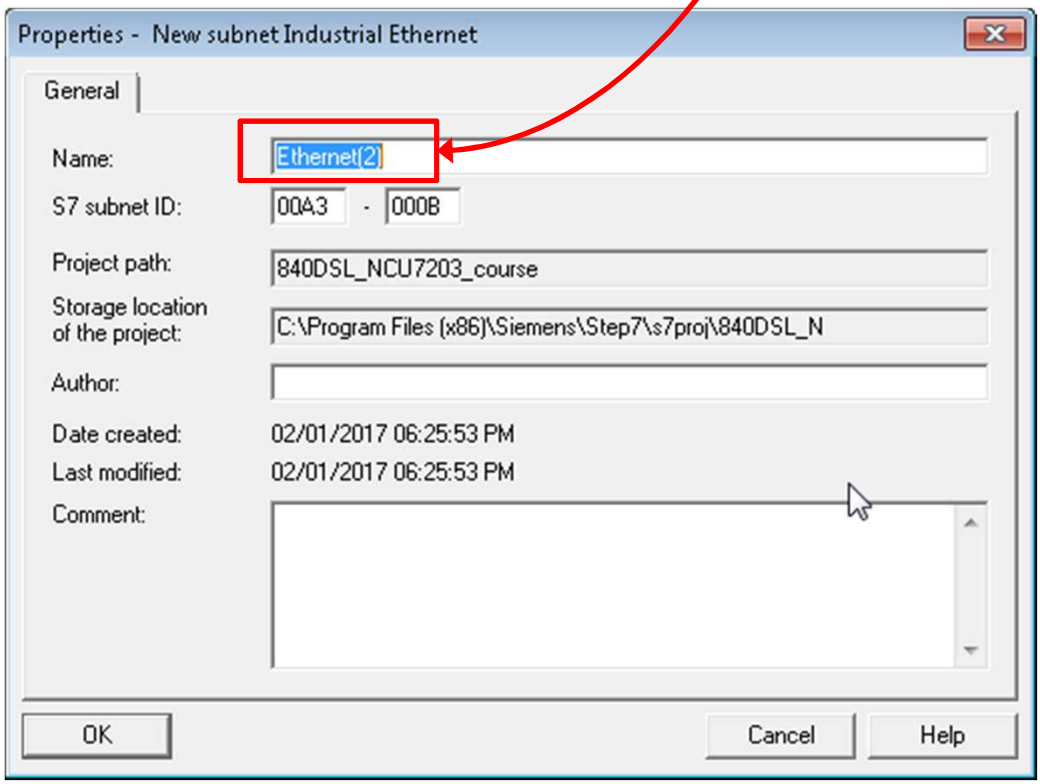
> Properties



- Network Ethernet In the following dialog box, network the Ethernet(2) by clicking the button

-> New -> OK





Parameterize the interfaces

Now select the newly networked interface Ethernet(2).

- Set the IP address

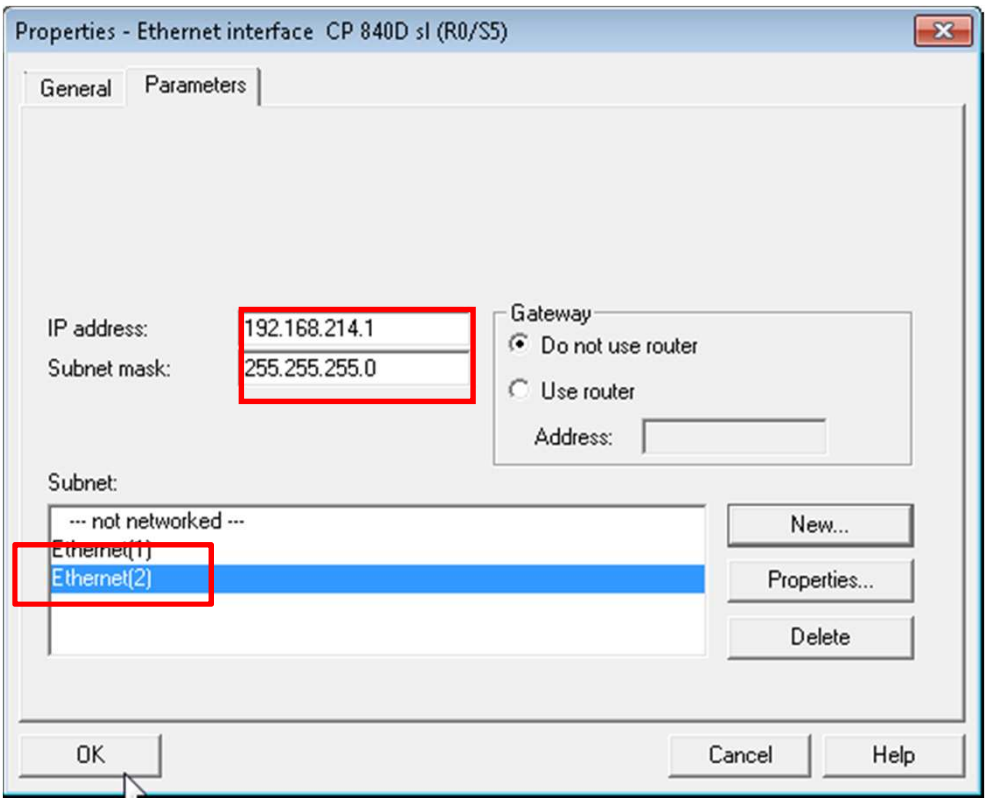
In this screen, change the IP address to the value that matches port X120:

- **192 . 168 . 214 . 1**

- Set the subnet mask

Change the subnet mask to the value

- **255 . 255 . 255 . 0**



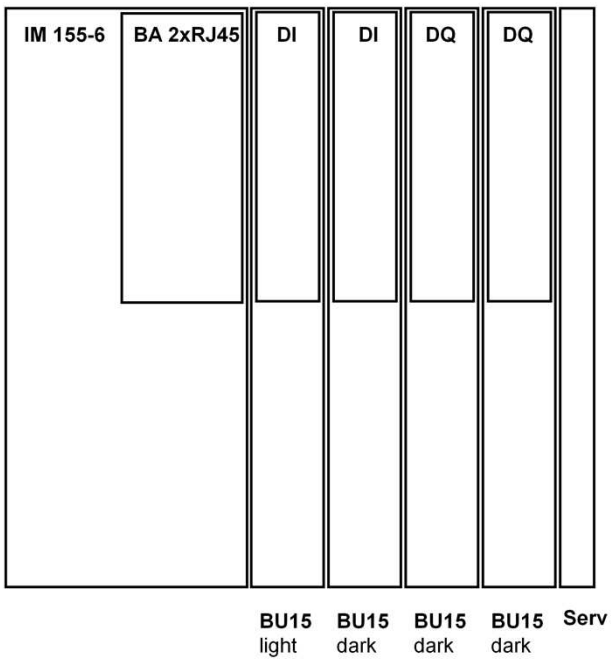
Reason

The default preassignment of the HW Config is suitable for the connection of the PG to port X127 of the NCU.

But we are connecting the PG to port X120 of the NCU via the Scalance switch.

Exercises: ET200 SP (PROFINET)
Selection of the I/O modules

SIEMENS





Mechanical configuration

Interface module	IM
Bus adapter	BA
Digital input module	DI
Digital output module	DQ
Server module	Serv
Base Unit	BU

- light: new potential group
- dark: extend the potential group

SITRAIN Course NC-84SL-SIP Page 8.2 - 37 Siemens AG © 2016

Reason

In the training unit I/O modules ET200 SP are connected to the PROFINET. Naturally, they also have to be included in the hardware configuration if the system is to function in its entirety. The ET200 I/O to be configured here are used to connect the simulator.

Objective

Create the hardware configuration of the PROFINET I/Os ET200 SP

Steps

1. Select an interface module IM
2. Install DI and DQ modules
3. Set the I/O addresses
4. Set monitoring functions
5. Check and set the potential groups
6. Check the device name of the IM and change it if necessary
7. Save and compile
8. Load the device name to the IM (initialize)
9. Download the HW Config to the CPU

1. Select an interface module IM

Select from the catalog, in directory

> PROFINET IO > I/O > ET200 SP

the folder

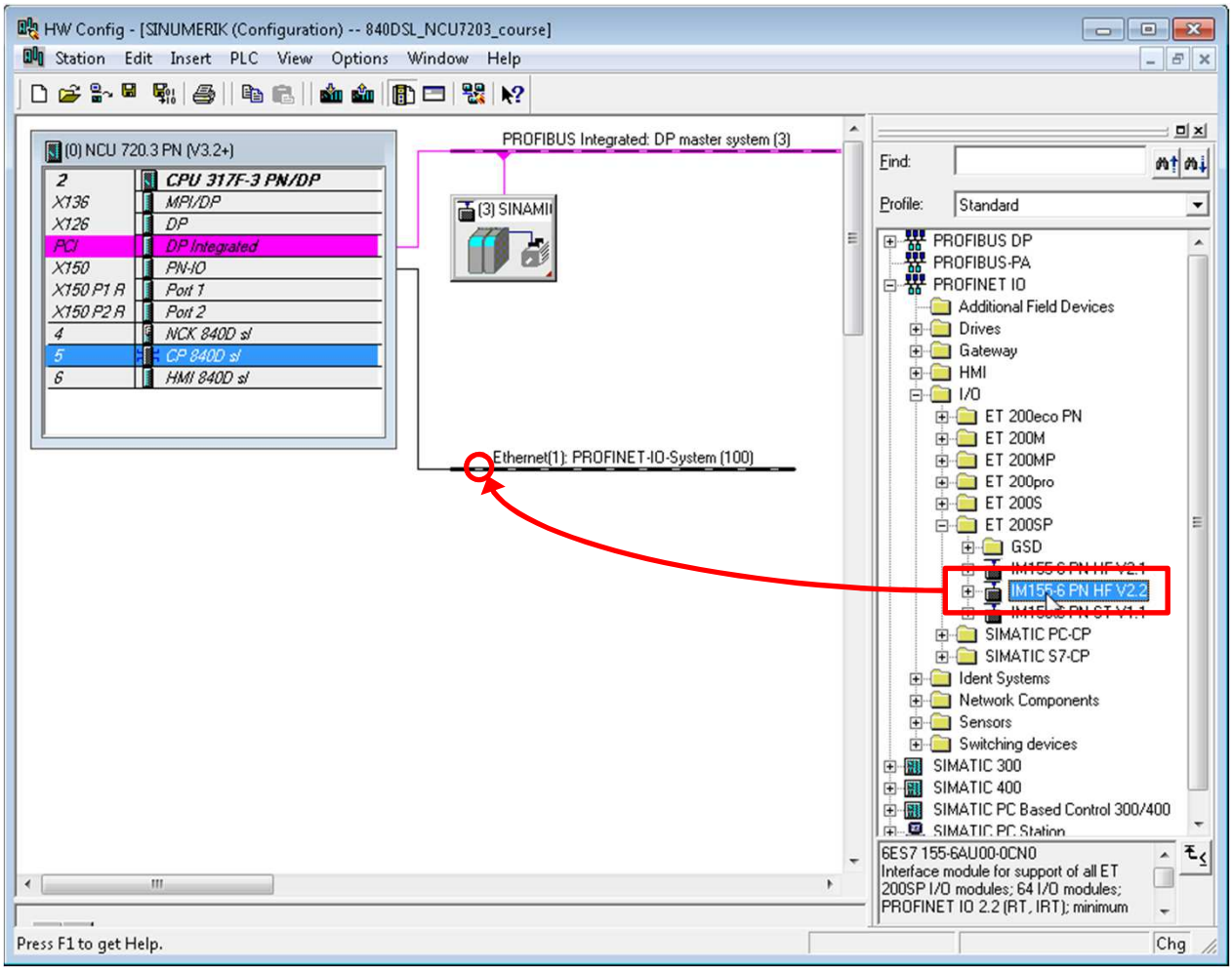
> IM 155-6 PN HF V2.2 High Feature version of the interface module

with the **correct article number** (and firmware version), i.e.:

6ES7 155-6AU00-0CN0

and drag this icon with the left mouse button to the segment

> Ethernet(1): PROFINET-IO-System (100)

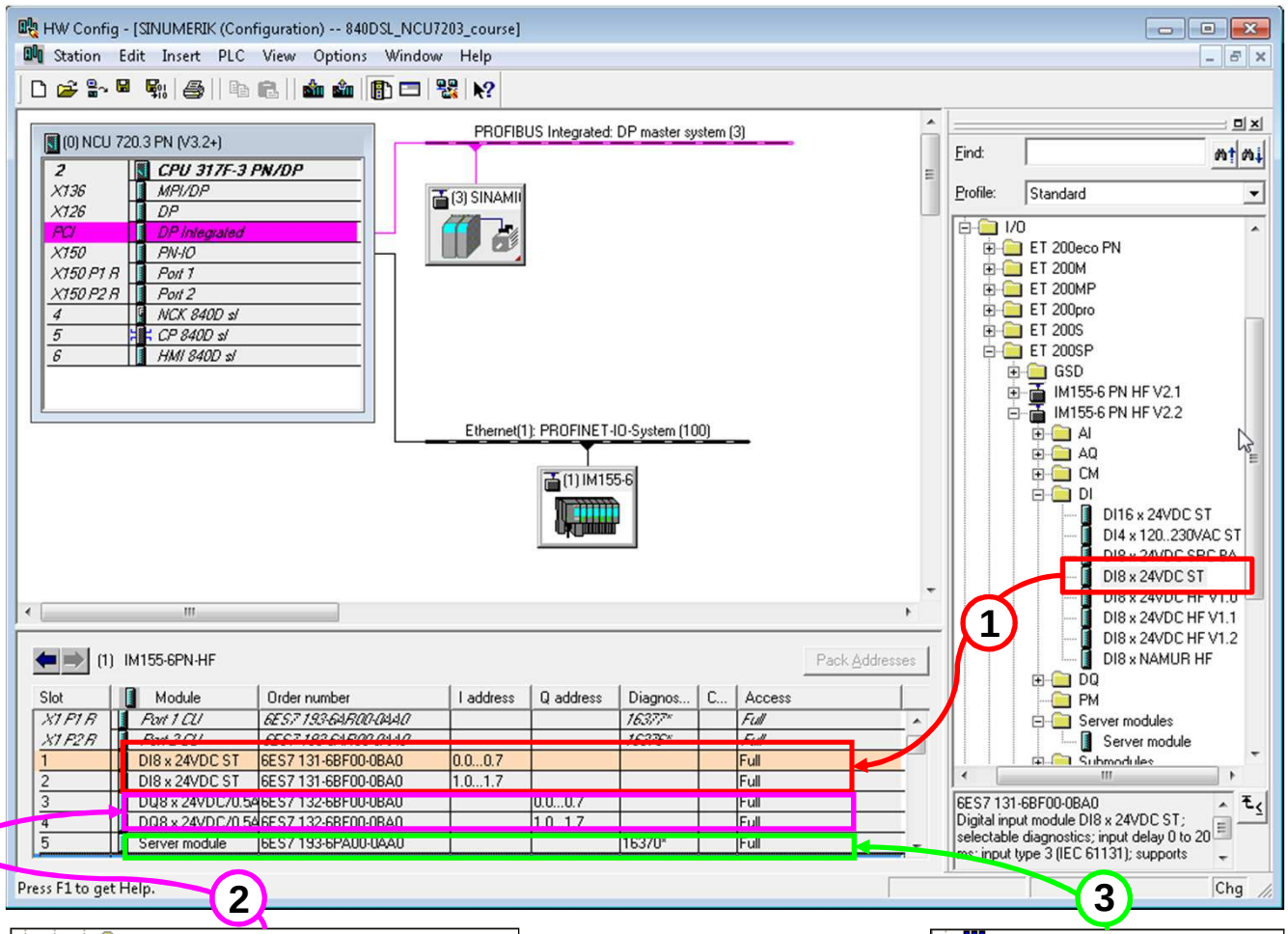


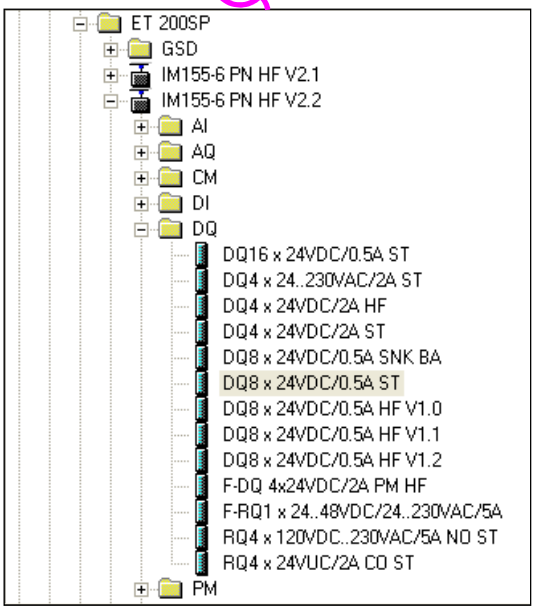
2. Install DI and DQ modules

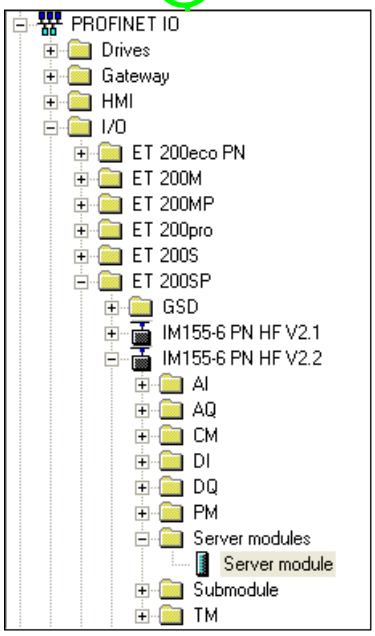
Now fit the slots of the IM with the existing digital input modules DI and digital output modules DQ and the server module (bus terminator).
This is in the subdirectories **DI**, **DQ,** and **server modules** of the IM

These **can** be the following modules. (Please check the module and version):

2 x	**DI 8x24V DC ST**	**6ES7 131-6BF00-0BA0**	
2 x	**DQ 8x24V DC ST**	**6ES7 132-6BF00-0BA0**	
1 x	**Server module**	**6ES7 193-6PA00-0AA0**	(bus terminator)



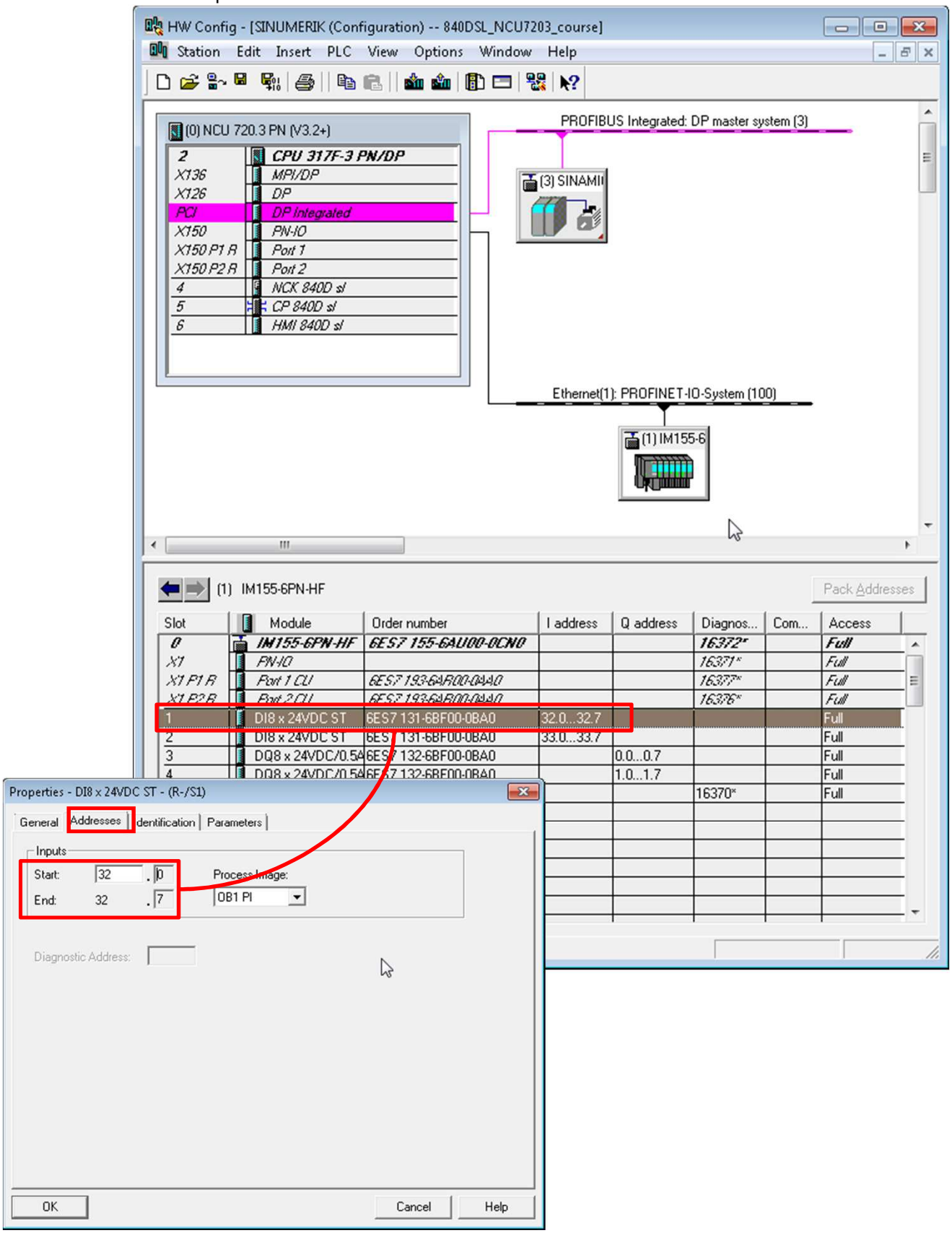




3. Set the I/O addresses

For the DI modules, set the addresses to IB 32 and IB 33, and for the DQ modules to QB 32 and QB 33.
In the screenshot, you can see the setting for the first DI module; the other modules can be parameterized analogously.

The address space as of IB 0 / QB 0 is already assigned to the machine control panel!

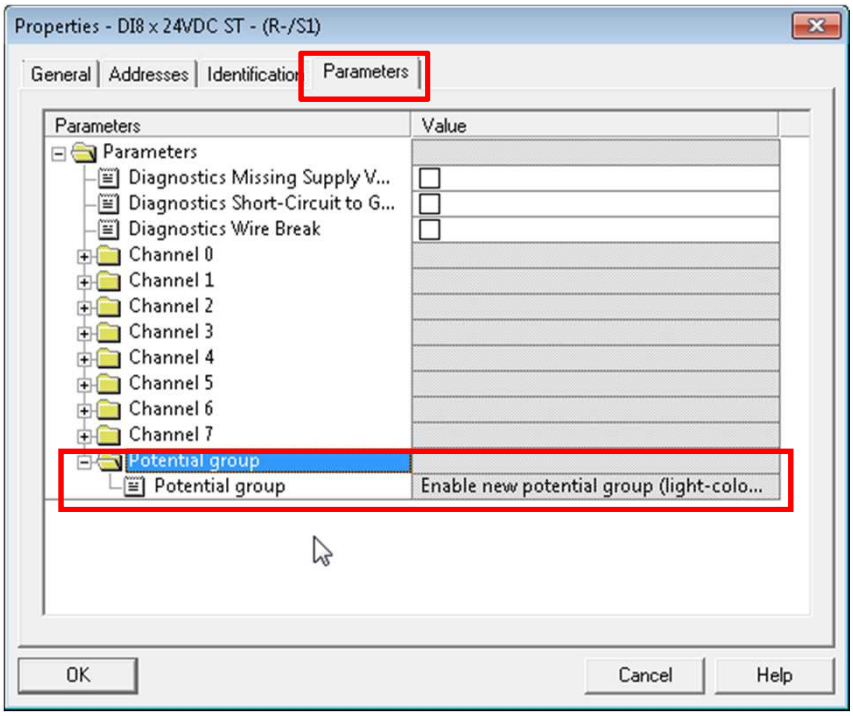


4. Set the potential groups

On the **'Parameters'** tab of the property dialog box of the DI or DQ modules, check (or change) the following settings:

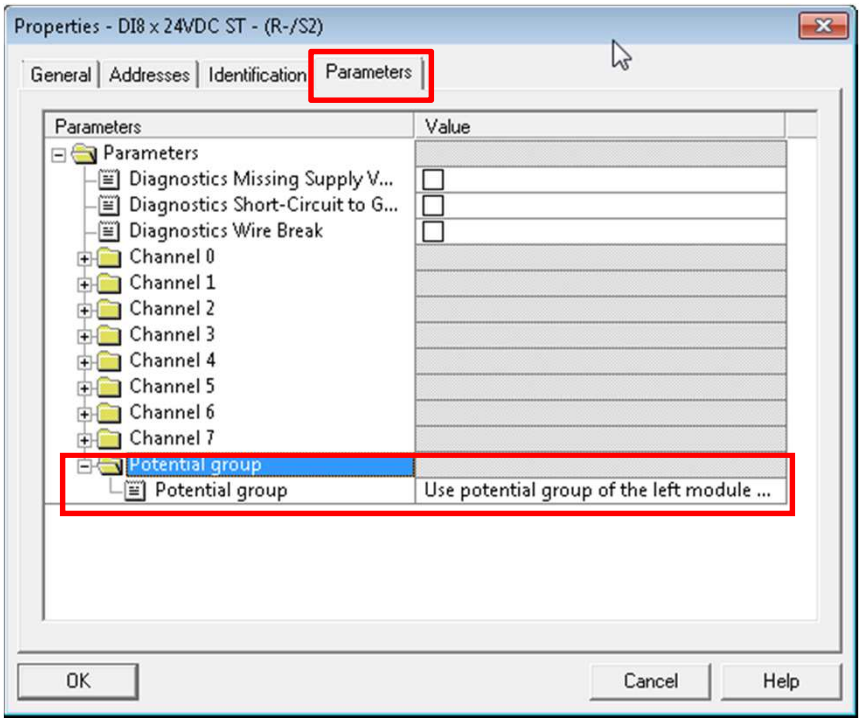
First DI module:

- Permit a new potential group (light base unit)



Further DI and DQ modules:

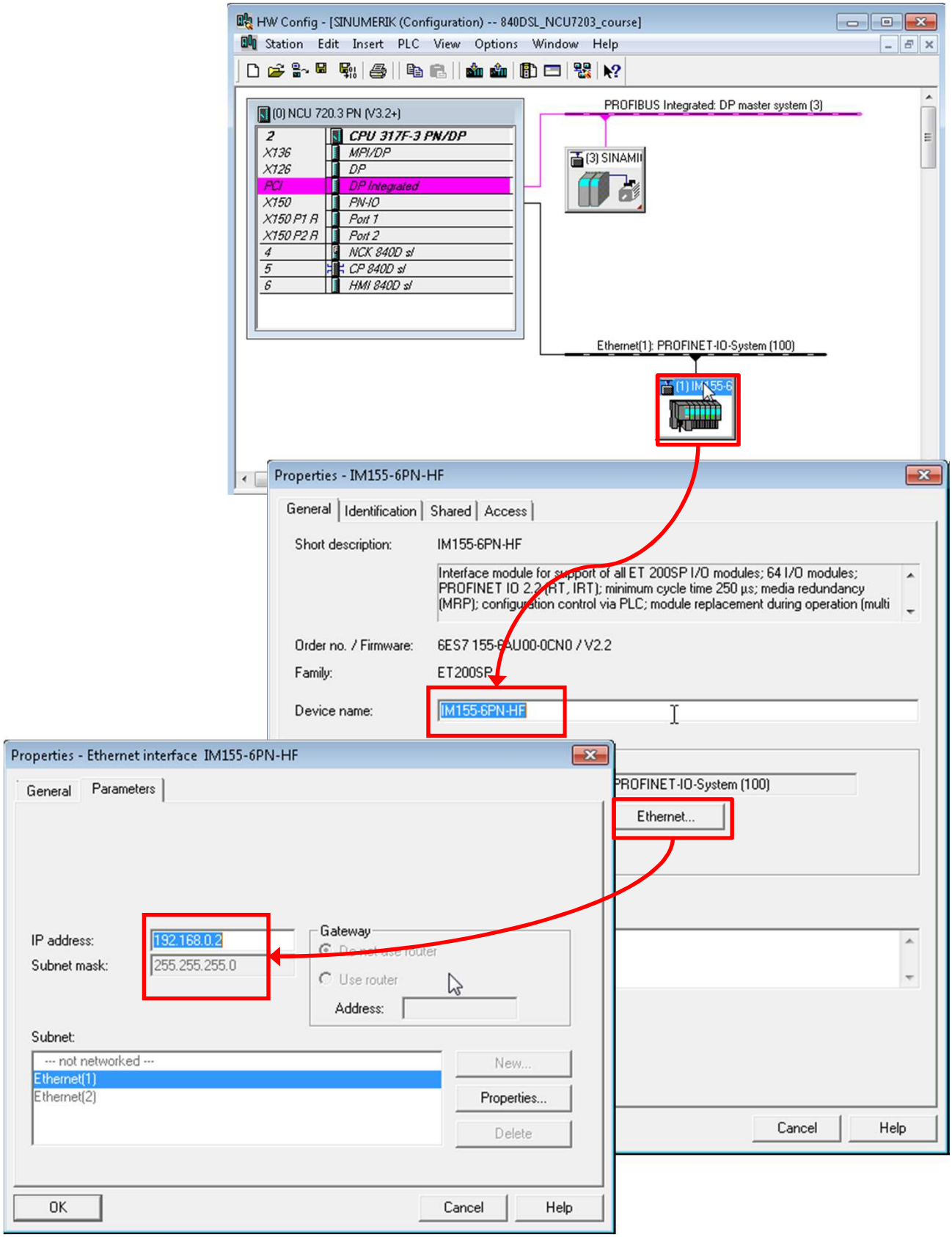
- Use the potential group of the left module (dark Base Unit)



5. Check the device name of the IM

Check the device names of the IM 155-6 and its IP address and subnet mask (**> double-click on IM > button: 'Ethernet'**).

At this point, it makes sense to note this information (in a text file) to have everything available in future initializations!

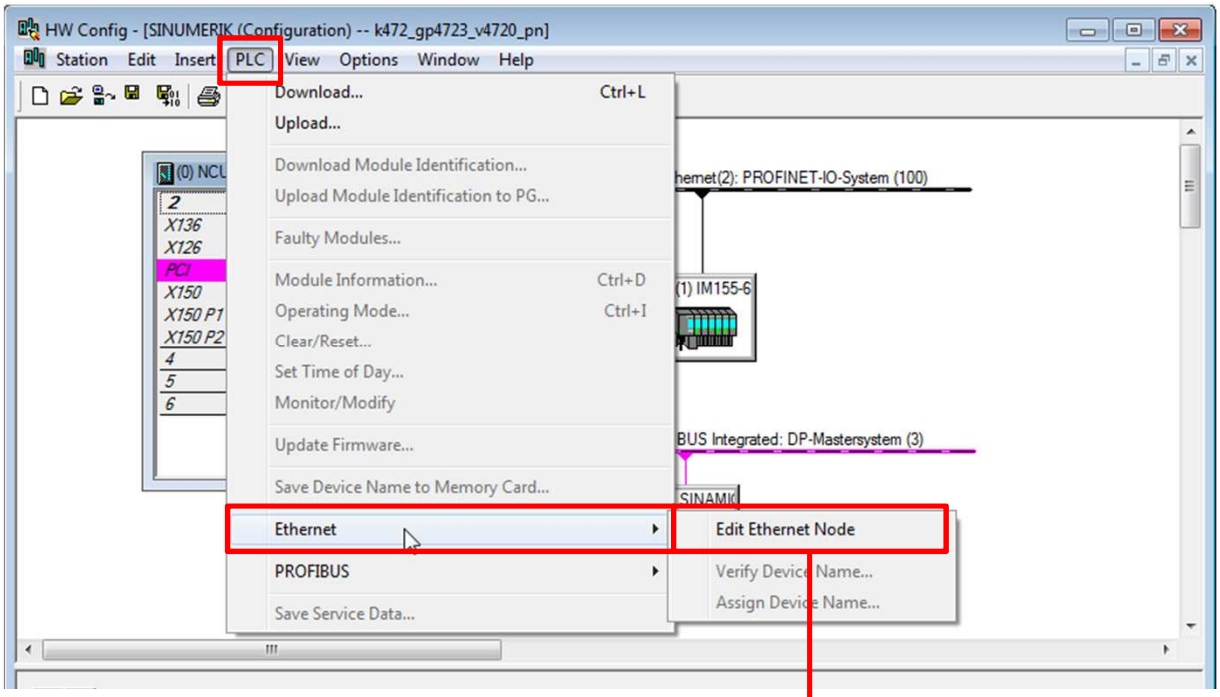


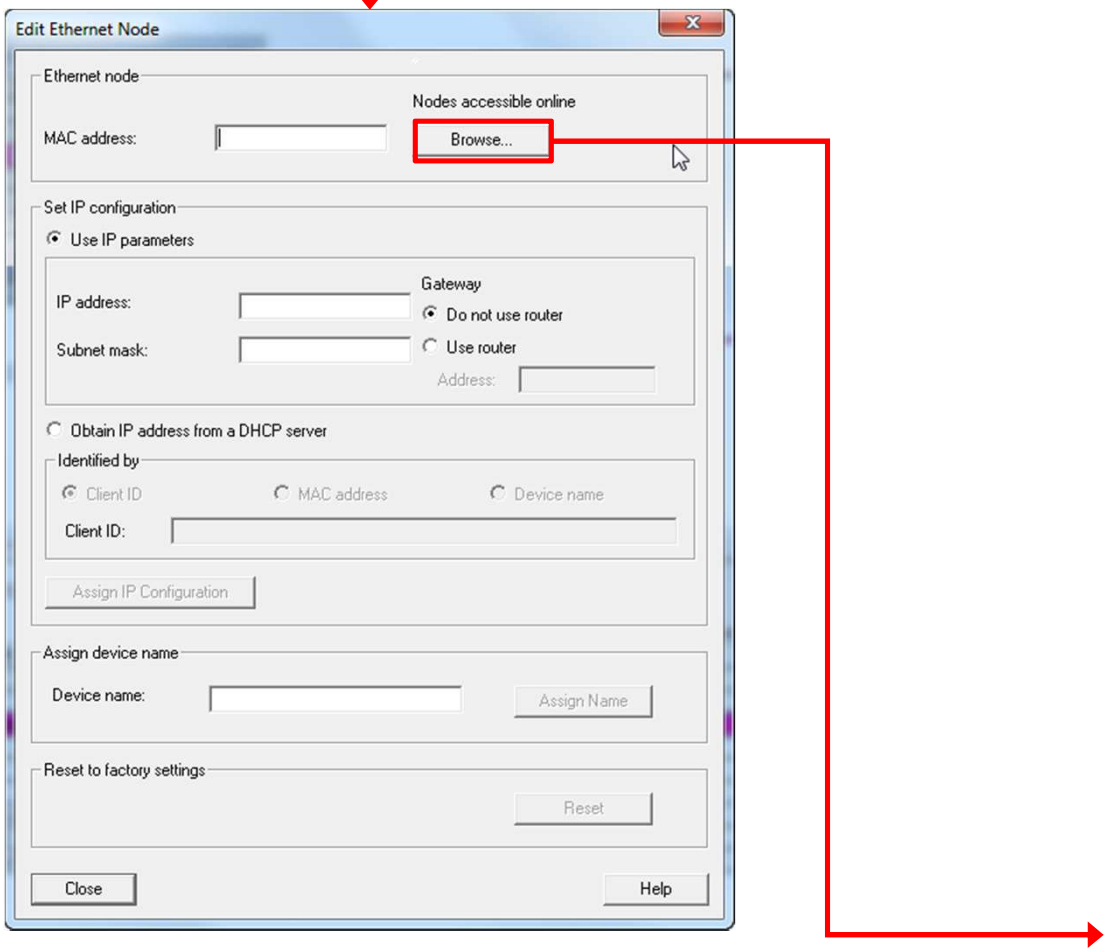
6. Load the device name to the IM (initialize)

You can now load the device name into the interface module (initialize).

To do this, select HW Config:

> Menu: 'Target system' > 'Ethernet' > 'Edit Ethernet node'





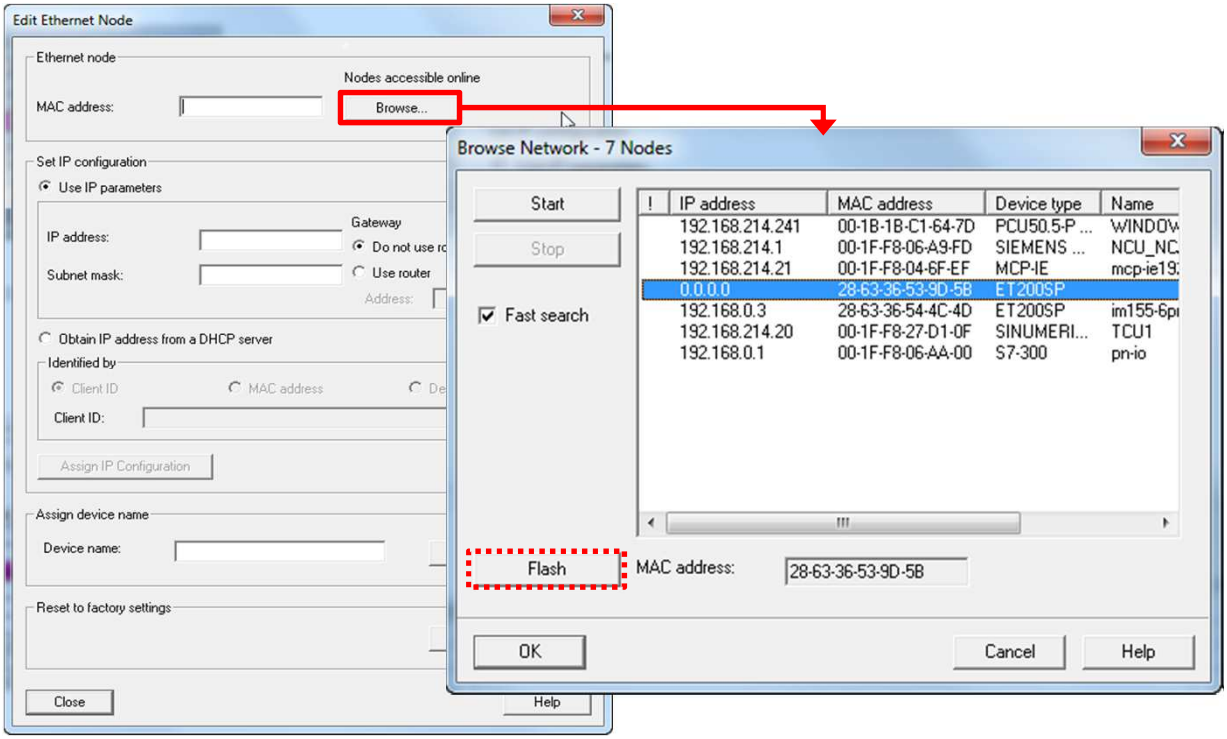
…6. Load the device name to the IM (cont.)

First scan for the accessible nodes.
To do this, select in this dialog box:

> 'Browse' button

There, select the device of type **ET200 SP**

(for multiple IMs, you must make the selection via the MAC address or use the **'Flashing'** button for identification)

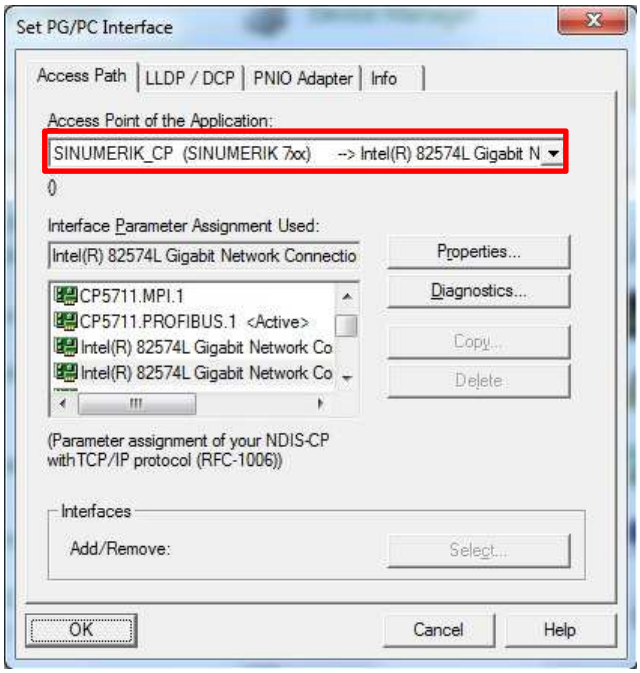


Note

If you do not find anything in this list, check the following setting:

> Control Panel > Set PG/PC interface

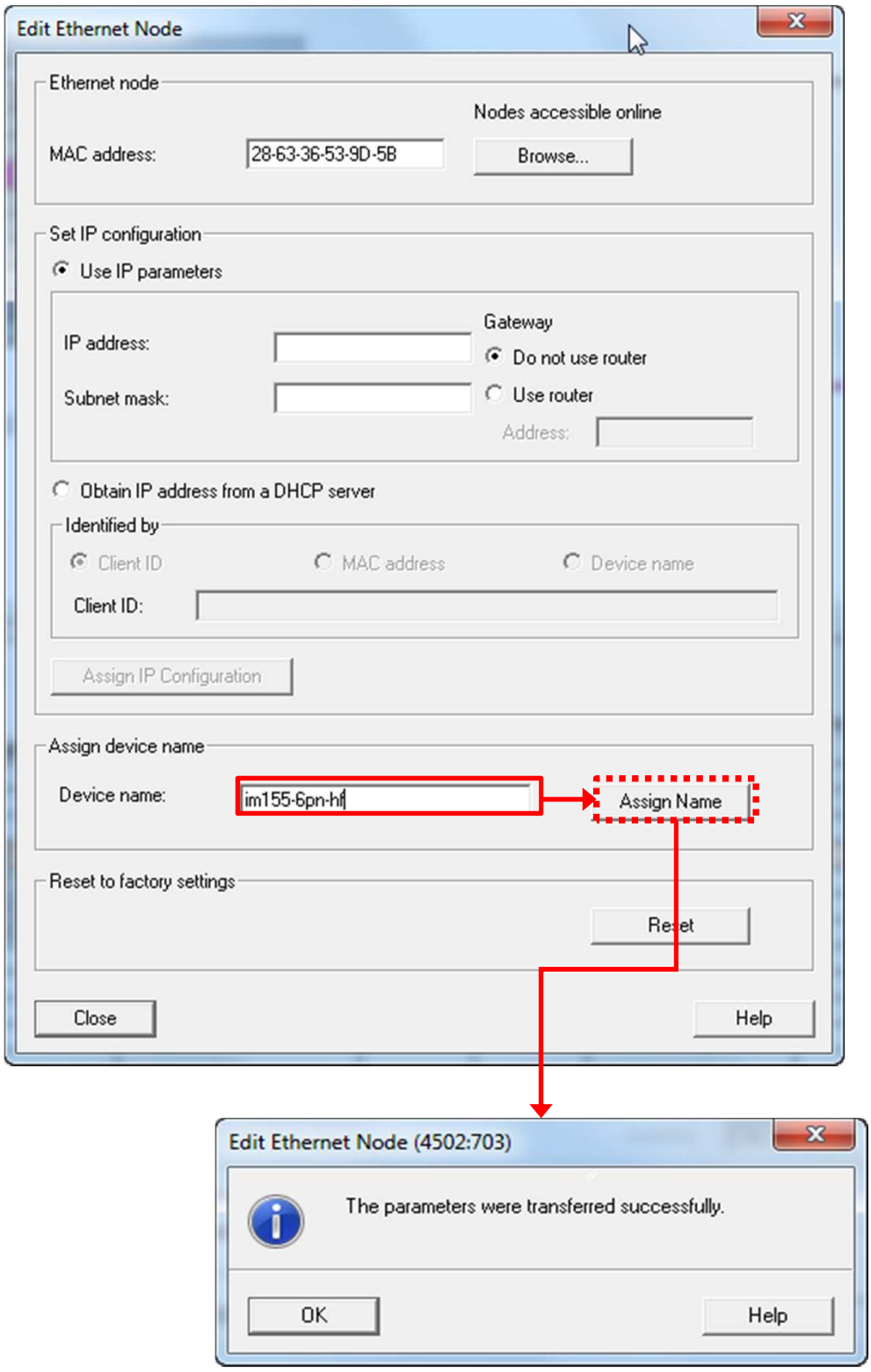
Check the access point of the application. There, you must select the interface used (and the TCP/IP protocol without 'Auto')



… 6. Load the device name to the IM (cont.)

Assign the name to the selected device in the next dialog box with **> button: 'Assign name'**

(Copy the name from the text file into the input field):

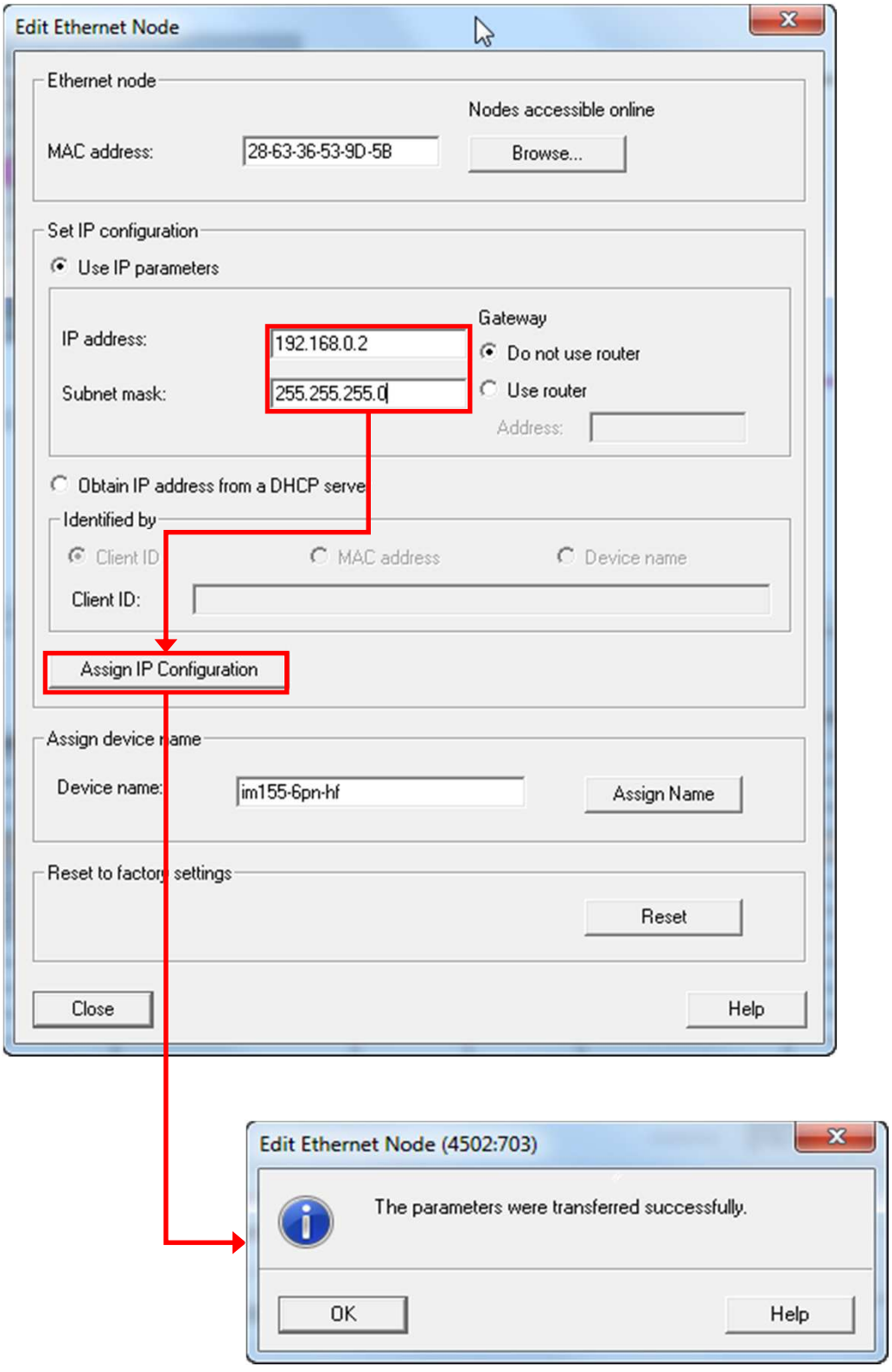


… 6. Load the device name to the IM (cont.)

Then write the IP address and subnet mask into the corresponding input fields (copy from text file) and assign them with the button:

> Assign IP configuration

(The help on the completion message only refers to CPs and IE/PB links, so is irrelevant in this case)

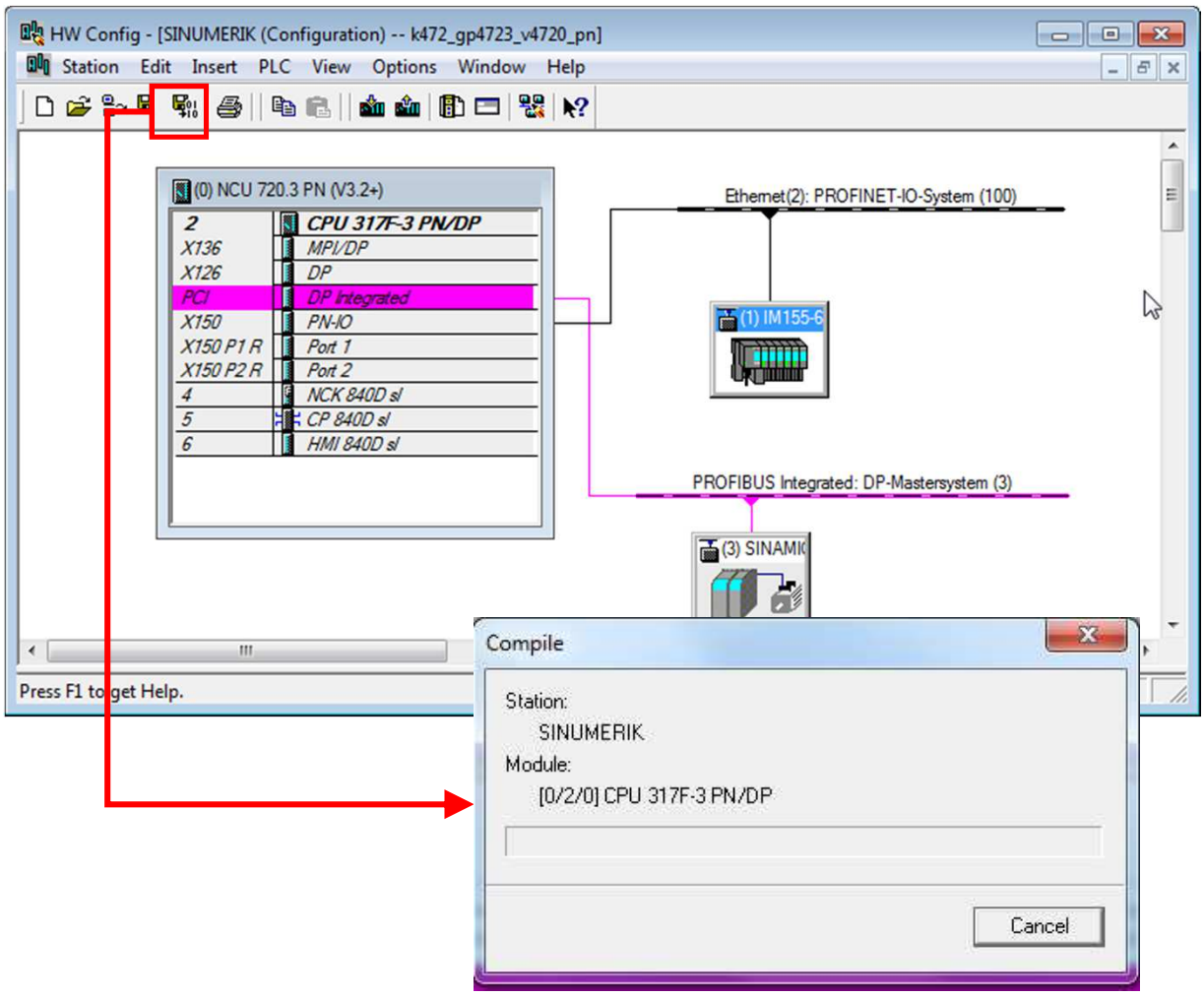


To use the configuration you have created, you must now save it in the project and then download it to the control.

7. Save and compile

Save the configuration by clicking

> Save and compile



PLC stop

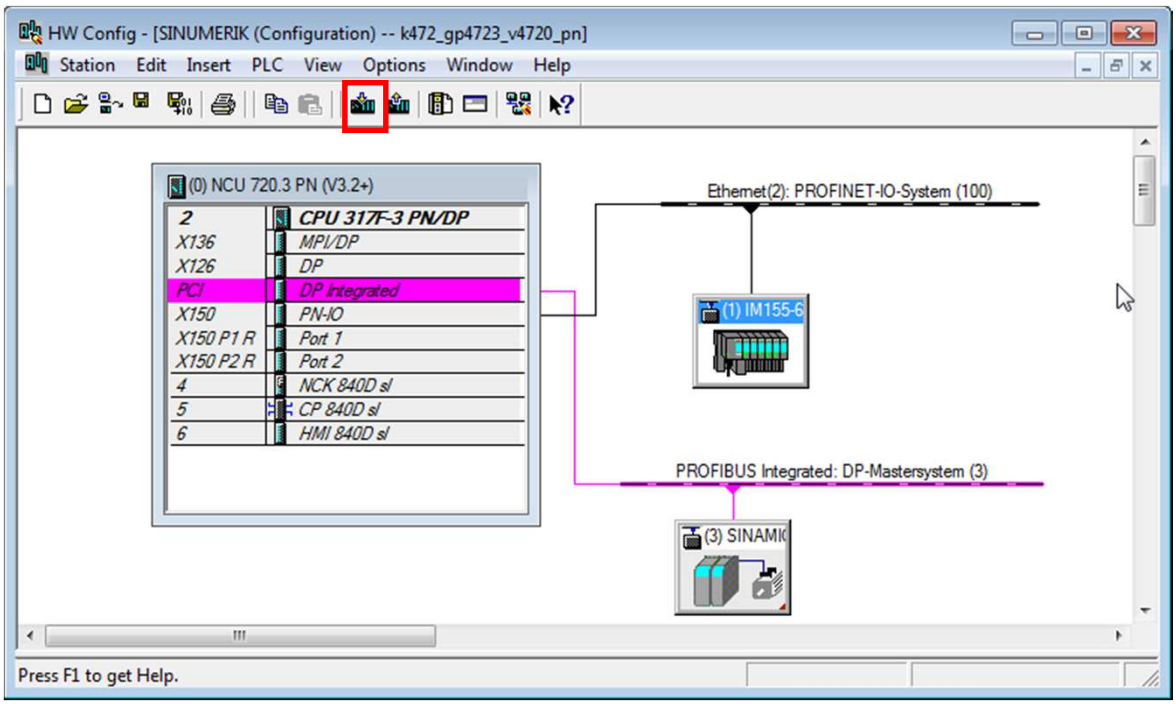
Switch the PLC to Stop by turning the rotary switch to the setting

- **PLC -> 2**

8. Download to the CPU

Download Load the configuration to he PLC by clicking

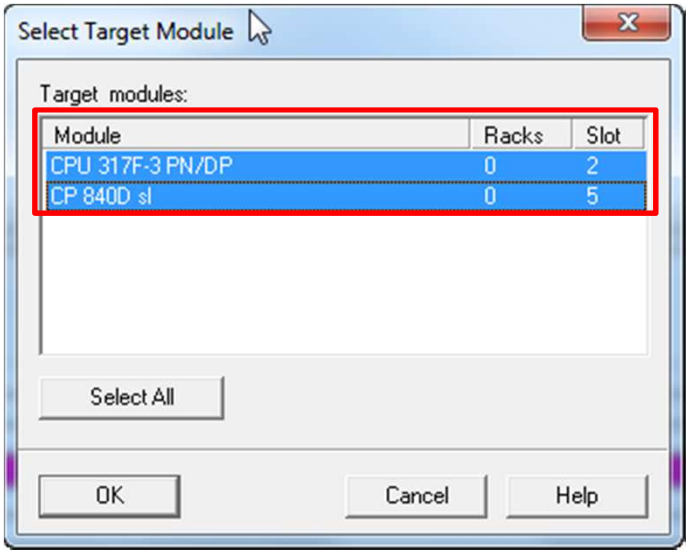
-> Download to module



Select PLC and CP

In the next dialog box, select both modules:

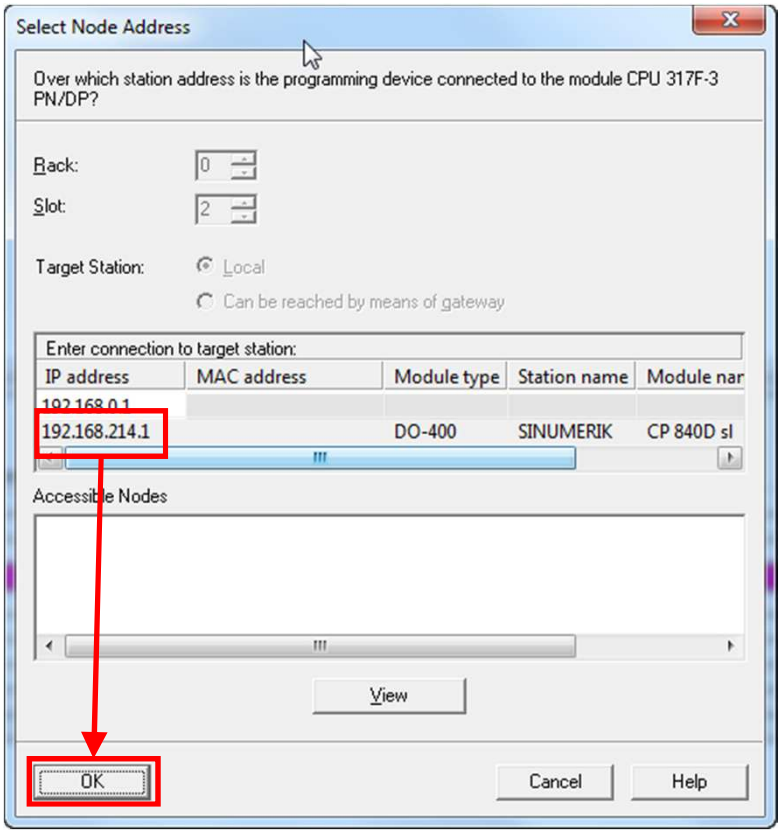
- **PLC 317F-3 PN/DP**
- **CP840D sl**



Destination address

Select the IP address

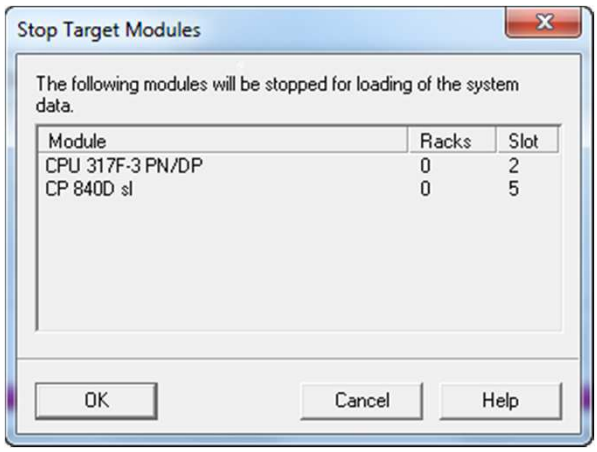
- **192.168.214.1** or
- **192.168.215.1** (on initial commissioning, CP address not yet changed)



Stop CP

Stop the CP in the dialog box

-> Stop target module



Leave PLC in Stop

Leave the 'PLC' rotary switch set in position 2 = STOP!

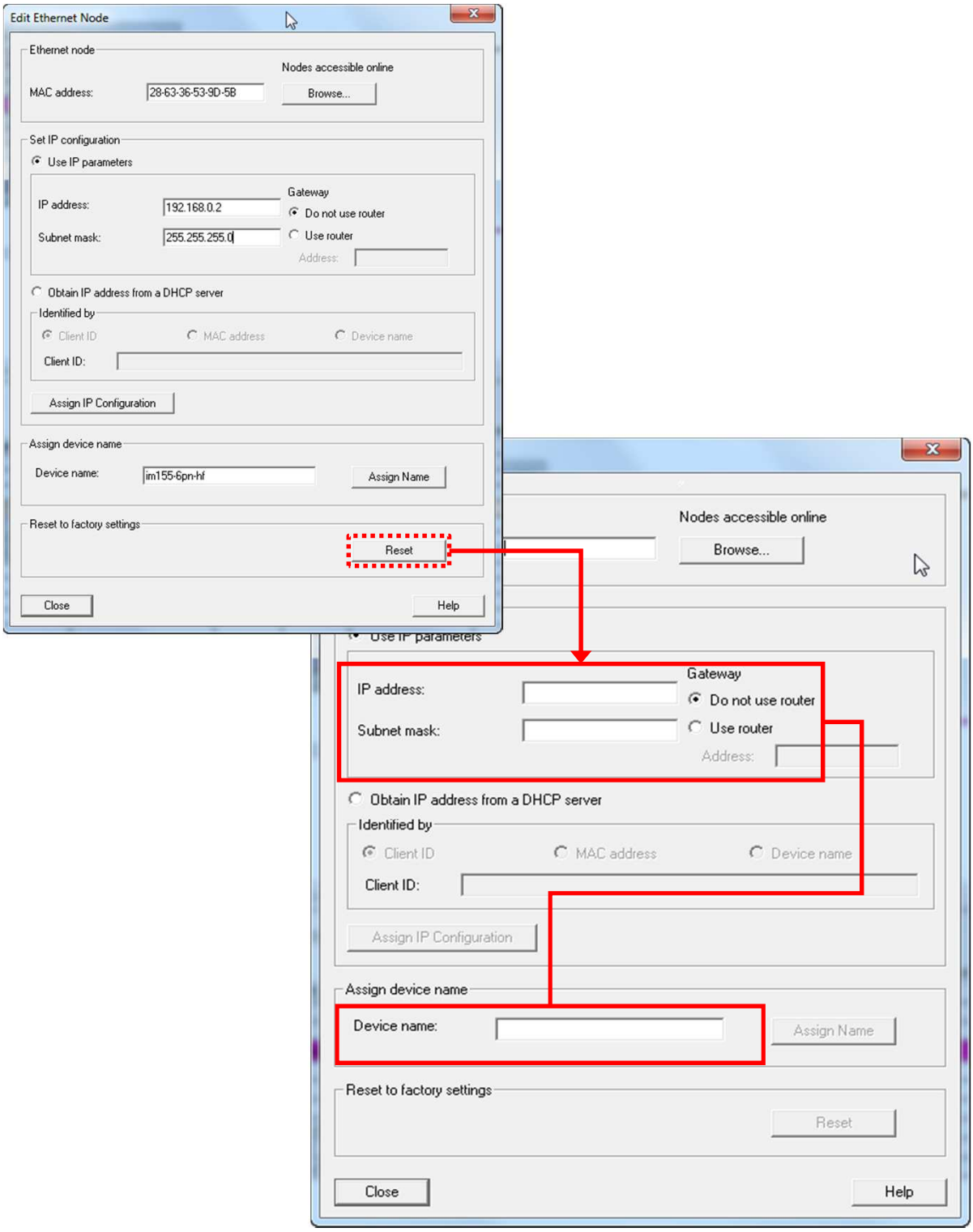
Notes on PROFINET

… Note on resetting the settings (device name, IP address) on a PN-IO device

You can reset an already configured device with

> Button: 'Reset'

That resets the name and the IP address to the as-delivered condition (no device name, no IP address).



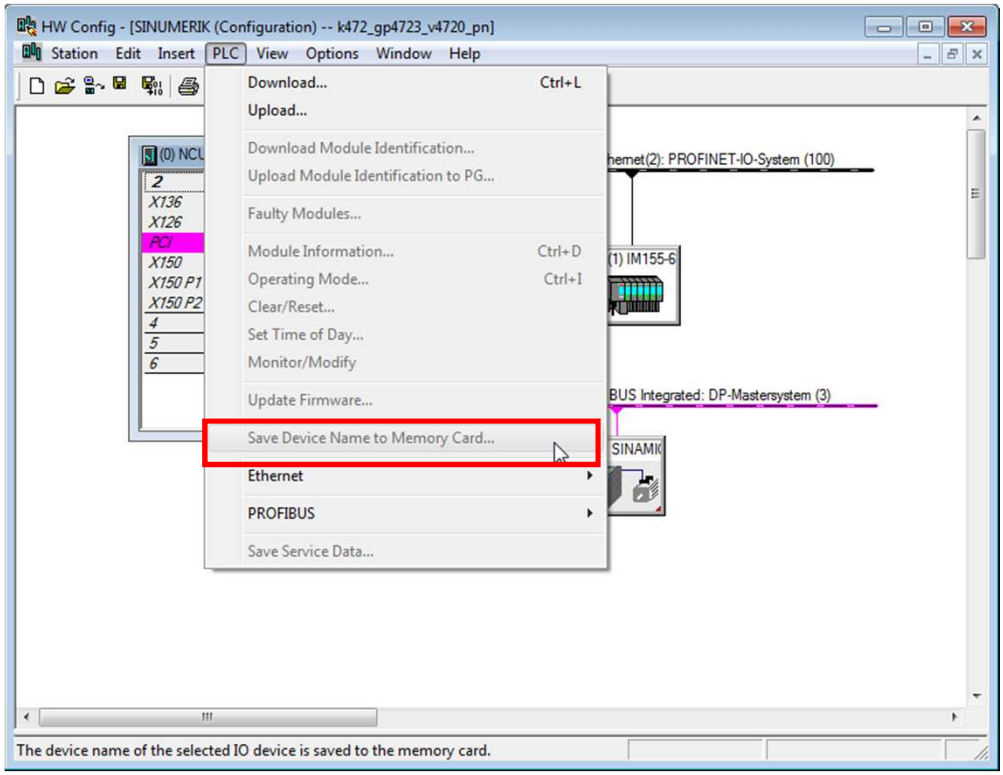
… Note on saving the name on SD card

On devices with an SD card, you can save the name on this card, too

> Button: 'Save device name to memory card'

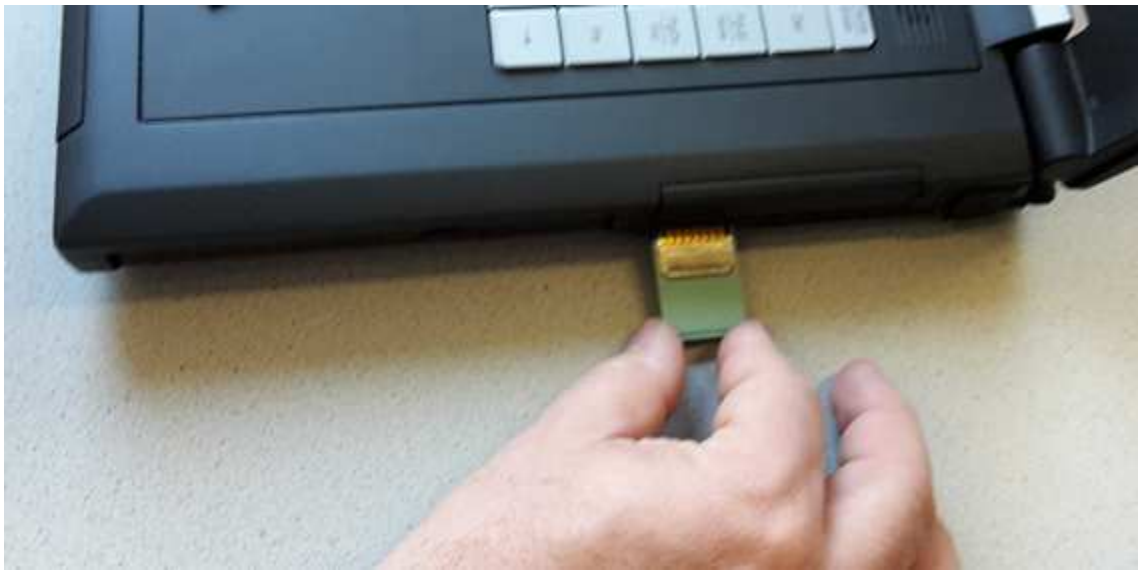
If the SD card is in the IM, the device name is read from the card on every power-up.

In this case, it is only necessary to have the SD card inserted when the modules of the IM are replaced.



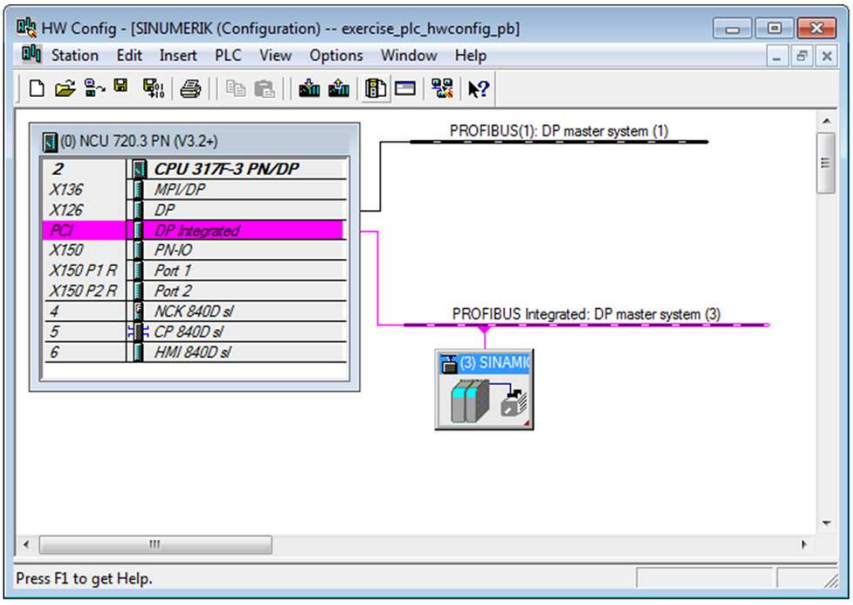
Note

However, to save the name on the card, the SD card must be inserted in the PG with the orientation shown here:



Exercises: ET200 S (PROFIBUS)
Create HW Config with NCU7x0

SIEMENS





Objective

To create a hardware configuration with a suitable NCU.

This activates the PROFIBUS interface to then be able to configure the ET200 S (PROFIBUS) I/Os.

Steps

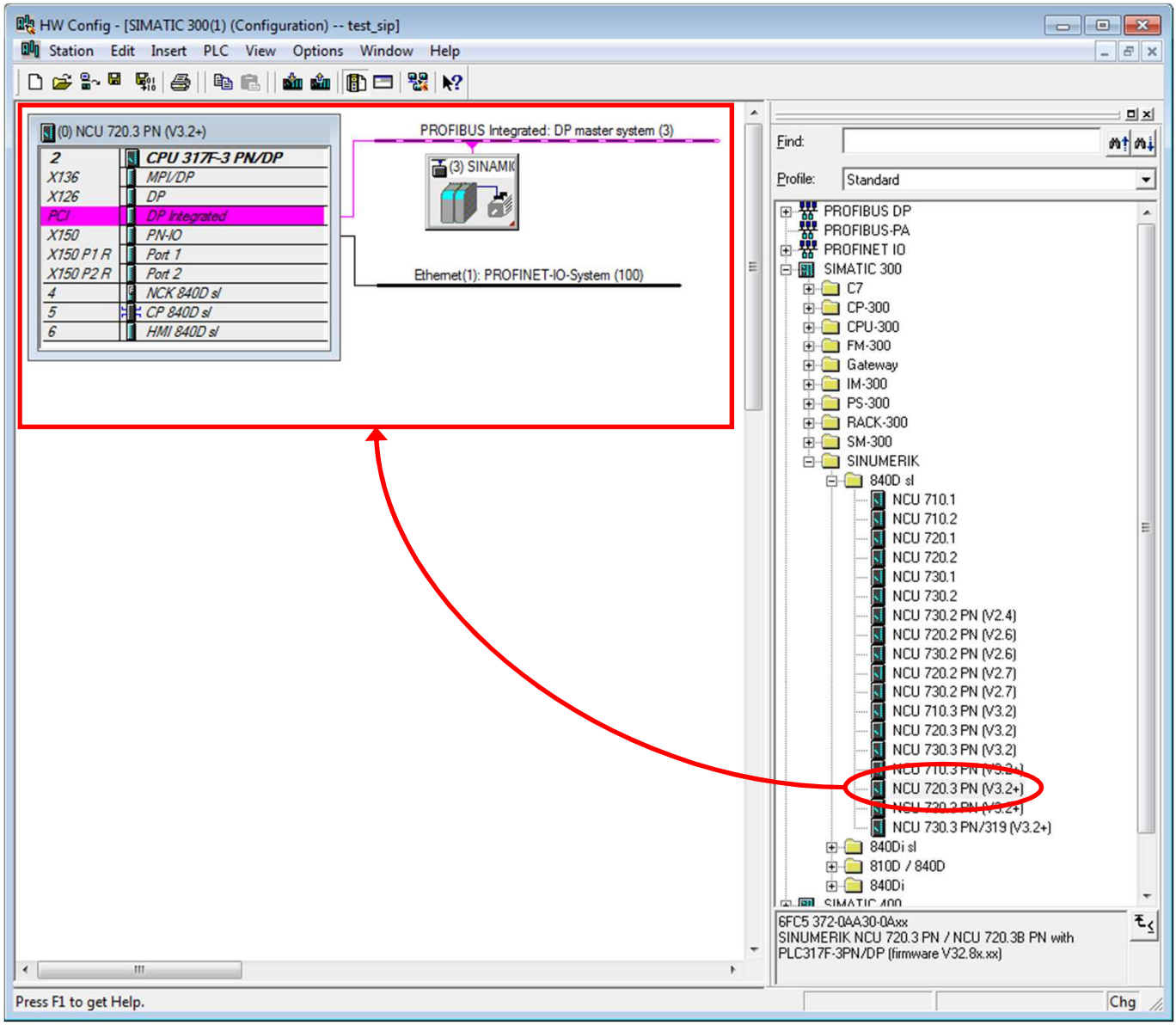
1. Start HW Config
2. Select an NCU
3. Leave the PROFINET inactive
4. Activate PROFIBUS
5. Set the IP address of the CP

1. Start HW Config From the SIMATIC Manager, launch the HW Config tool for the station you have just created.

2. Select NCU In the module catalog, select a suitable NCU 7x0 from the path

> SIMATIC 300 > SINUMERIK > 840D sl

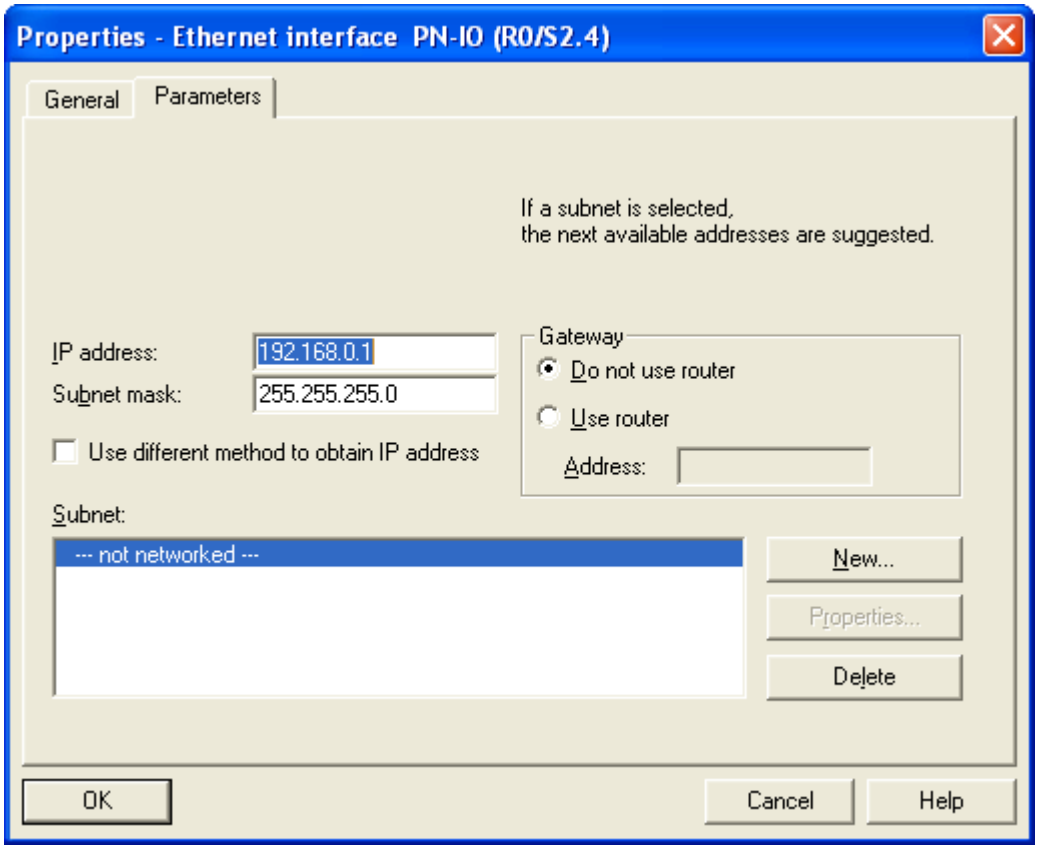
> NCU 720.3 (V3.2+)



3. Do not network PROFINET

In the dialog box for the PROFINET interface 'Ethernet PN-IO', choose 'not networked' with

> Button: OK



4. Network PROFIBUS

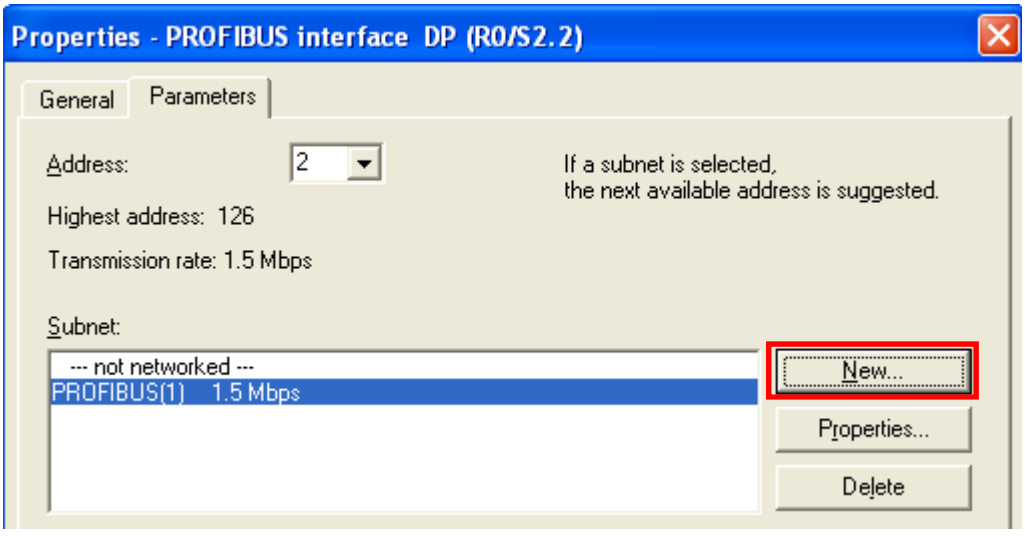
In the next dialog box that is displayed

- **Properties PROFIBUS interface**

you should activate the DP interface via

-> New -> PROFIBUS(1)

Note For drives on ADI4: Choose baudrate 12 Mbps!



5. Set the IP address of the CP

The programming device is connected to port X120 on the SINUMERIK NCU. Communication is established via an integrated Ethernet CP. This Ethernet CP is assigned an IP address matching the X120 port to enable communication with the PG.

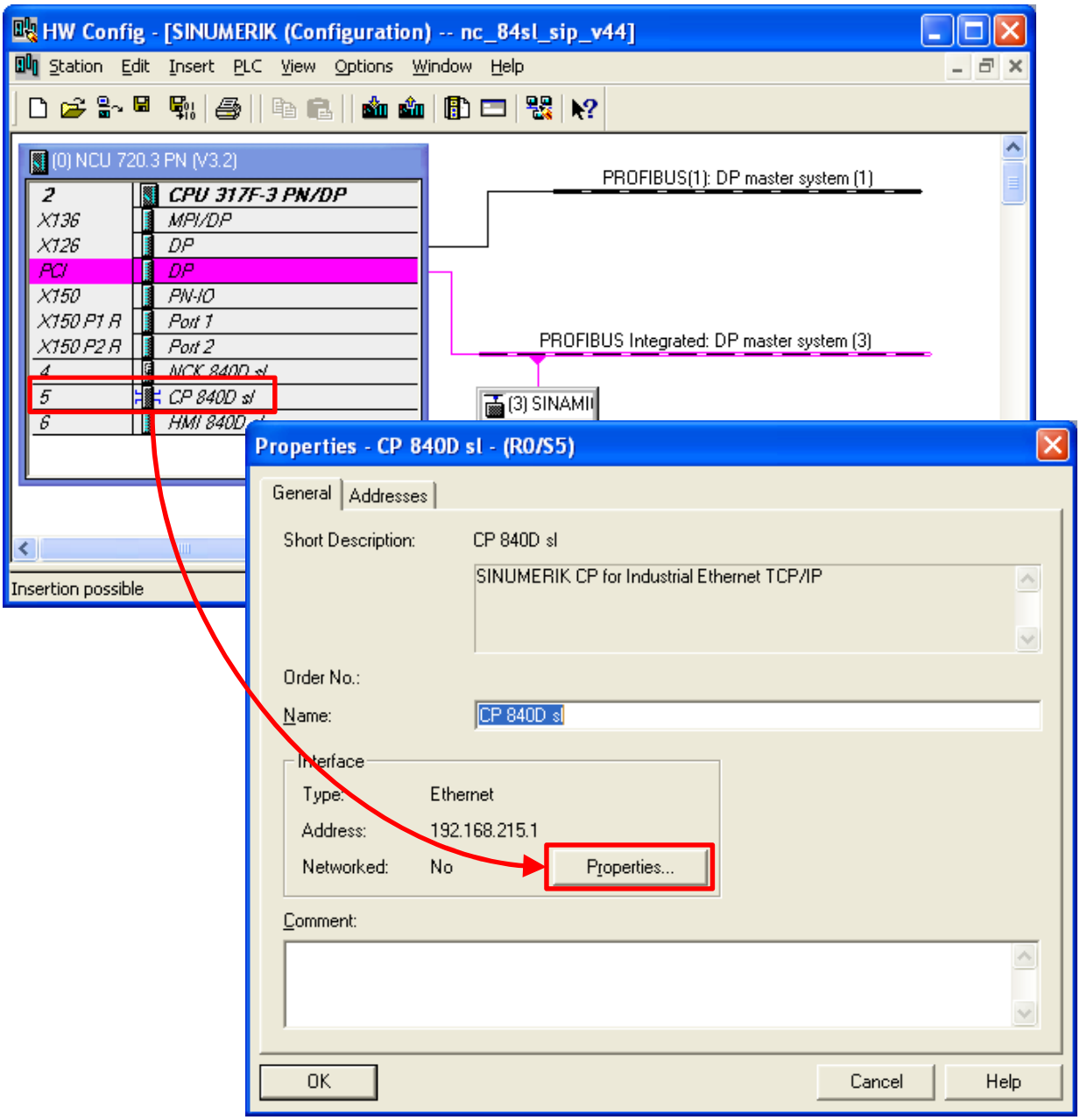
Properties CP

Double-click the CP 840Dsl to open dialog box

> **Properties – CP 840D**

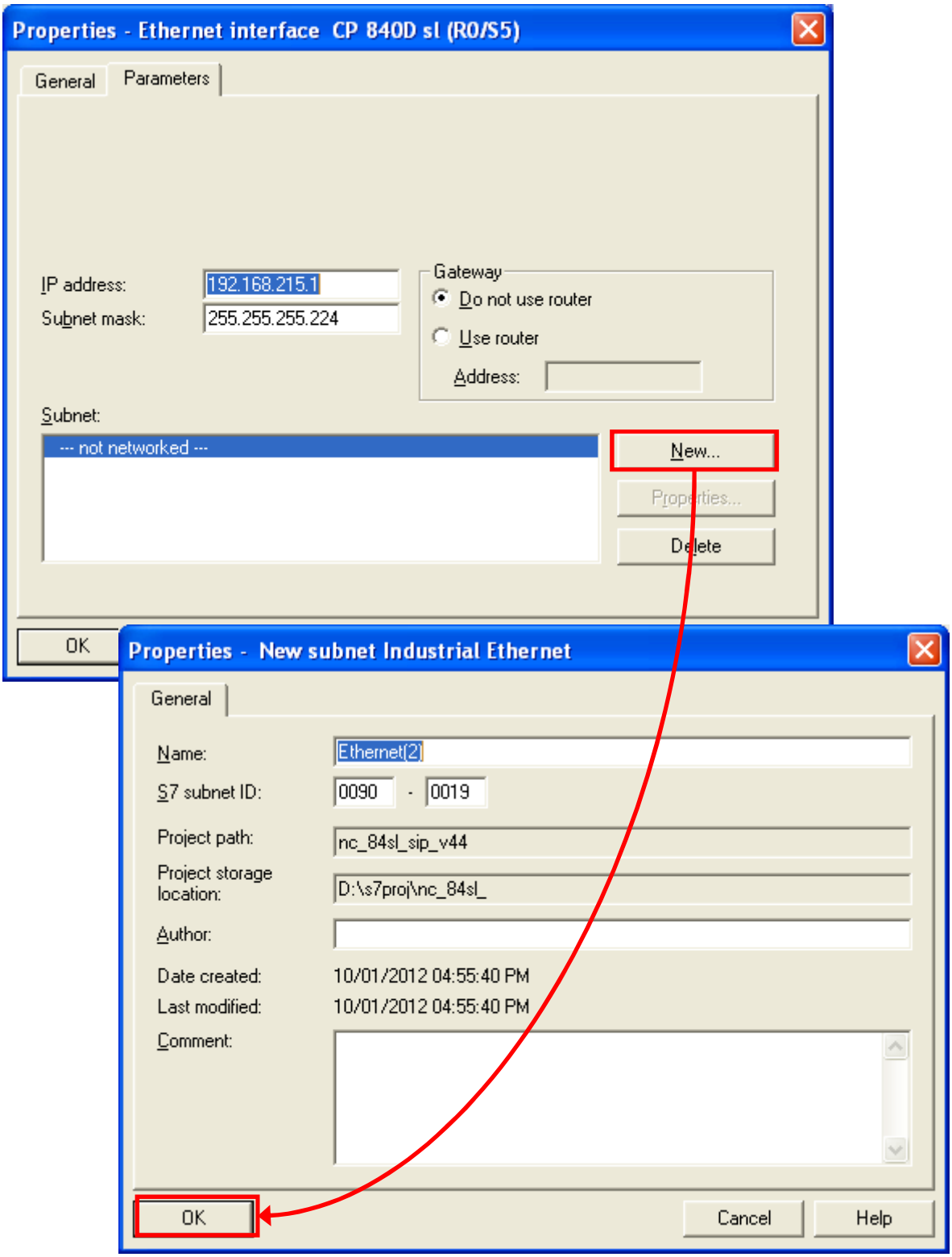
Then click:

> Properties



Network Ethernet In the dialog box that then opens, activate the Ethernet by clicking the button:

-> New -> OK



Parameterize the interfaces

Now select the newly networked interface Ethernet(1).

Set the IP address

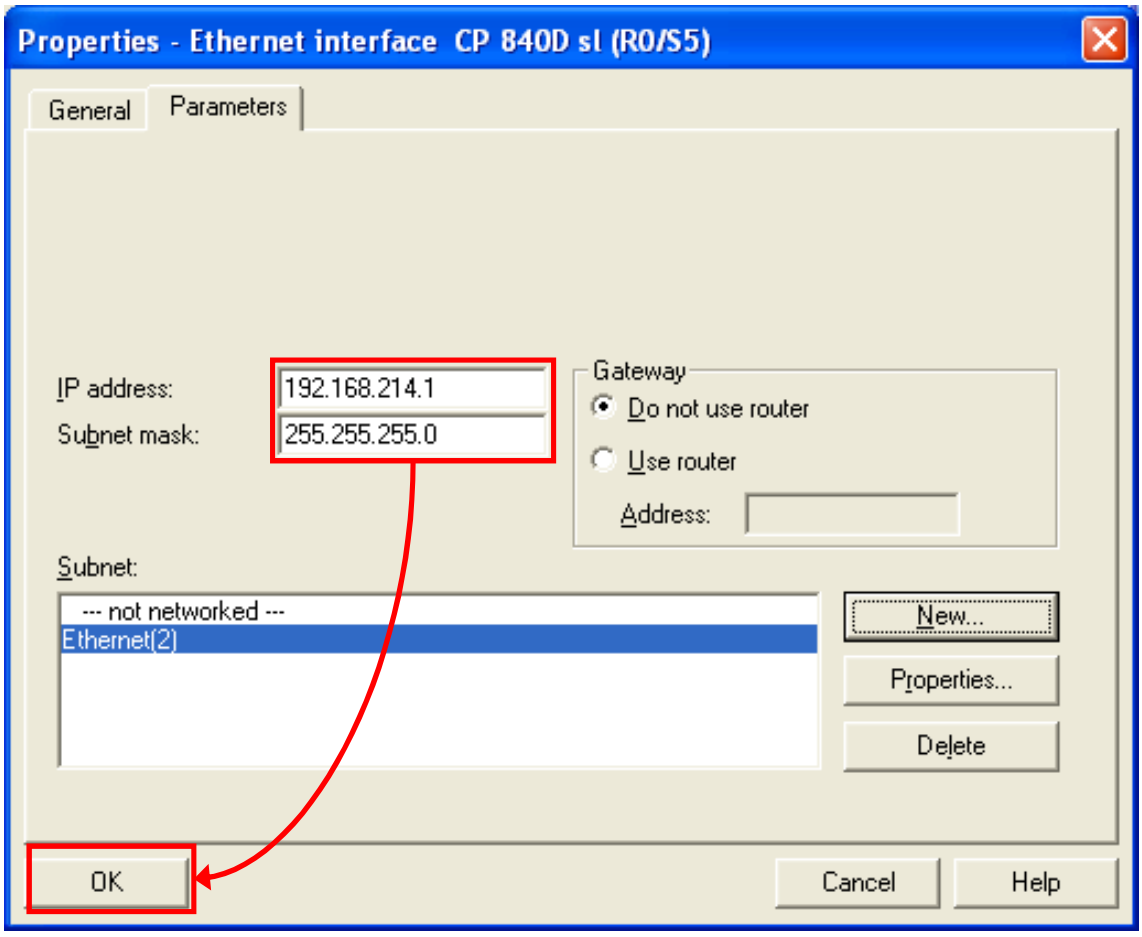
In this screen, change the IP address to the value that matches port X120:

- **192 . 168 . 214 . 1**

Set the subnet mask

Change the subnet mask to the value

- **255 . 255 . 255 . 0**

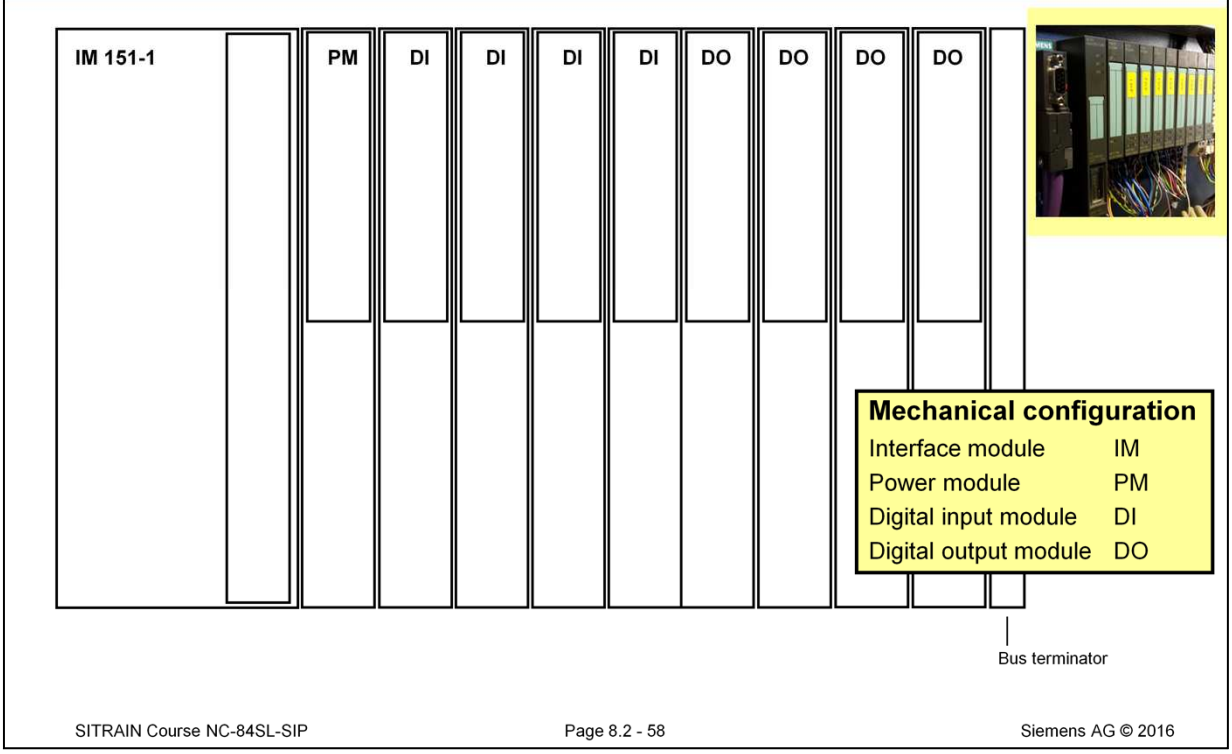


Reason

The default preassignment of the HW Config is suitable for the connection of the PG to port X127 of the NCU.

But we are connecting the PG to port X120 of the NCU via the Scalance switch.

Exercises: ET200 S (PROFIBUS)
Selection of the I/O modules



Reason

In the training unit I/O modules ET200 S are connected to the PROFIBUS. Naturally, they also have to be included in the hardware configuration if the system is to function in its entirety. The ET200 I/O to be configured here is used to connect the simulator.

Objective

To create the hardware configuration of the PROFIBUS I/Os ET200 S

Steps

1. Select an interface module IM
2. Fit the PM, DI, and DO modules
3. Set the I/O addresses
4. Save and compile
5. Download the HW Config into the CPU

1. Select an interface module IM

Select from the catalog, in directory

> PROFIBUS DP > ET200S (do not confuse with ET200iS !)

the folder

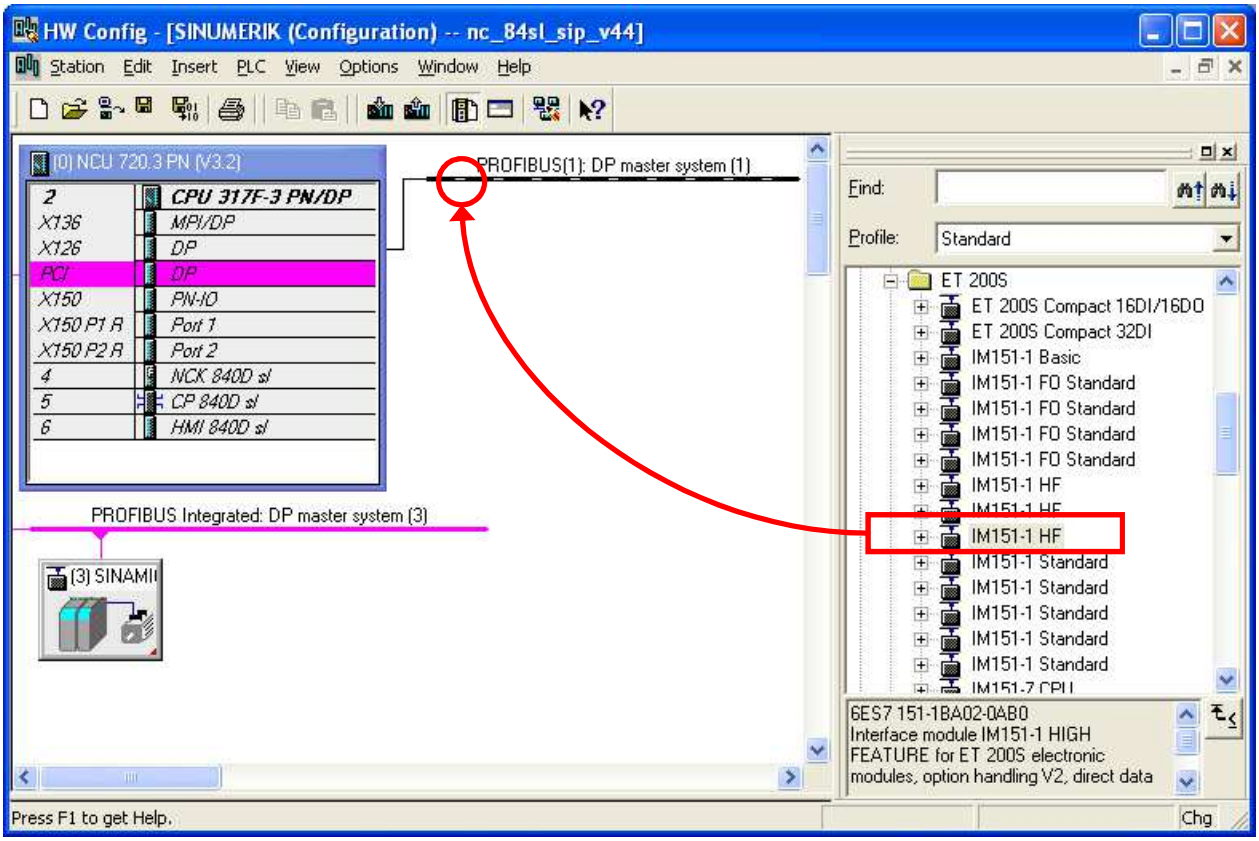
> IM 151-1 HF High Feature version of the interface module

with the **correct module version (!)**, e.g.:

6ES7 151-1BA**02**-0AB0: **3.** Interface module series IM151-1 HF

and drag this icon with the left mouse button to segment

- **PROFIBUS DP master system (1)**



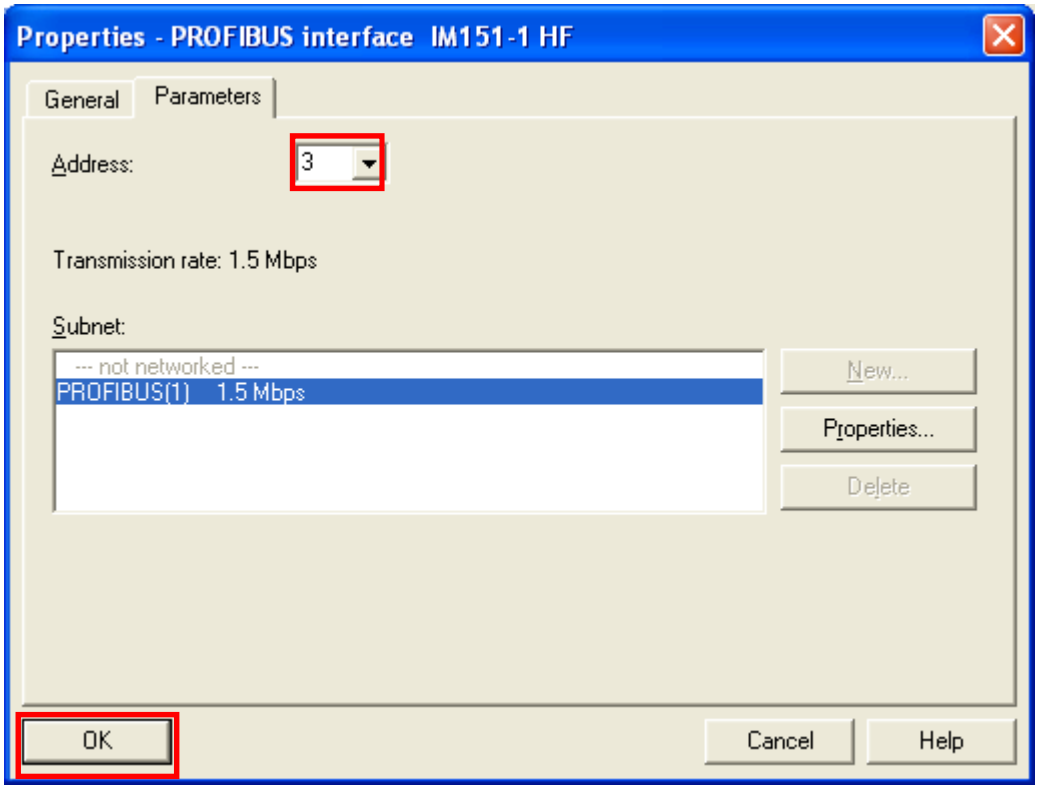
Bus address

In the dialog box, change the bus address:

- **Address = 3 -> OK**

(You can also change this later by double-clicking the IM151-HF and clicking

-> PROFIBUS)



Address switch

The PROFIBUS address of the IM151-1 is set at the switches designated

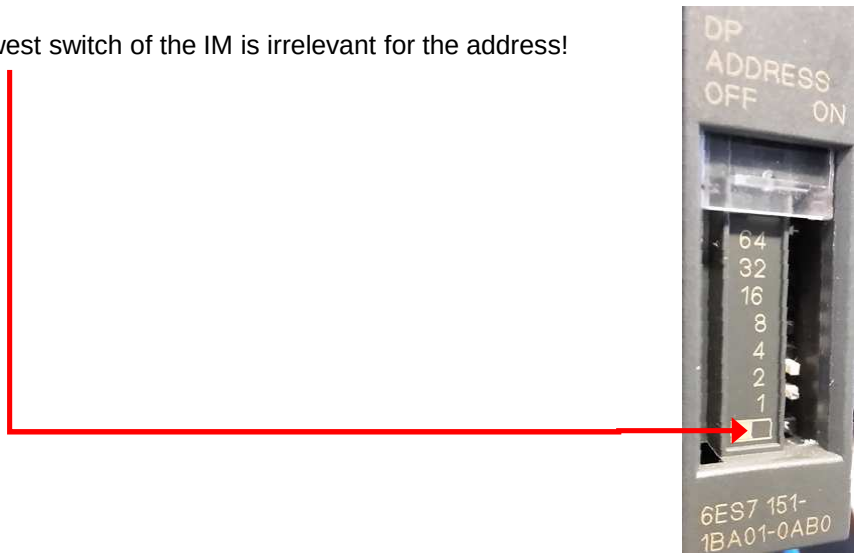
- **1, 2, 4, 8, ..., 64** .

The designation corresponds to to the significance of the switch.

The default setting is 1 = ON, 2 = ON, 4, ..., 64 = OFF, therefore PROFIBUS address of the IM = 3

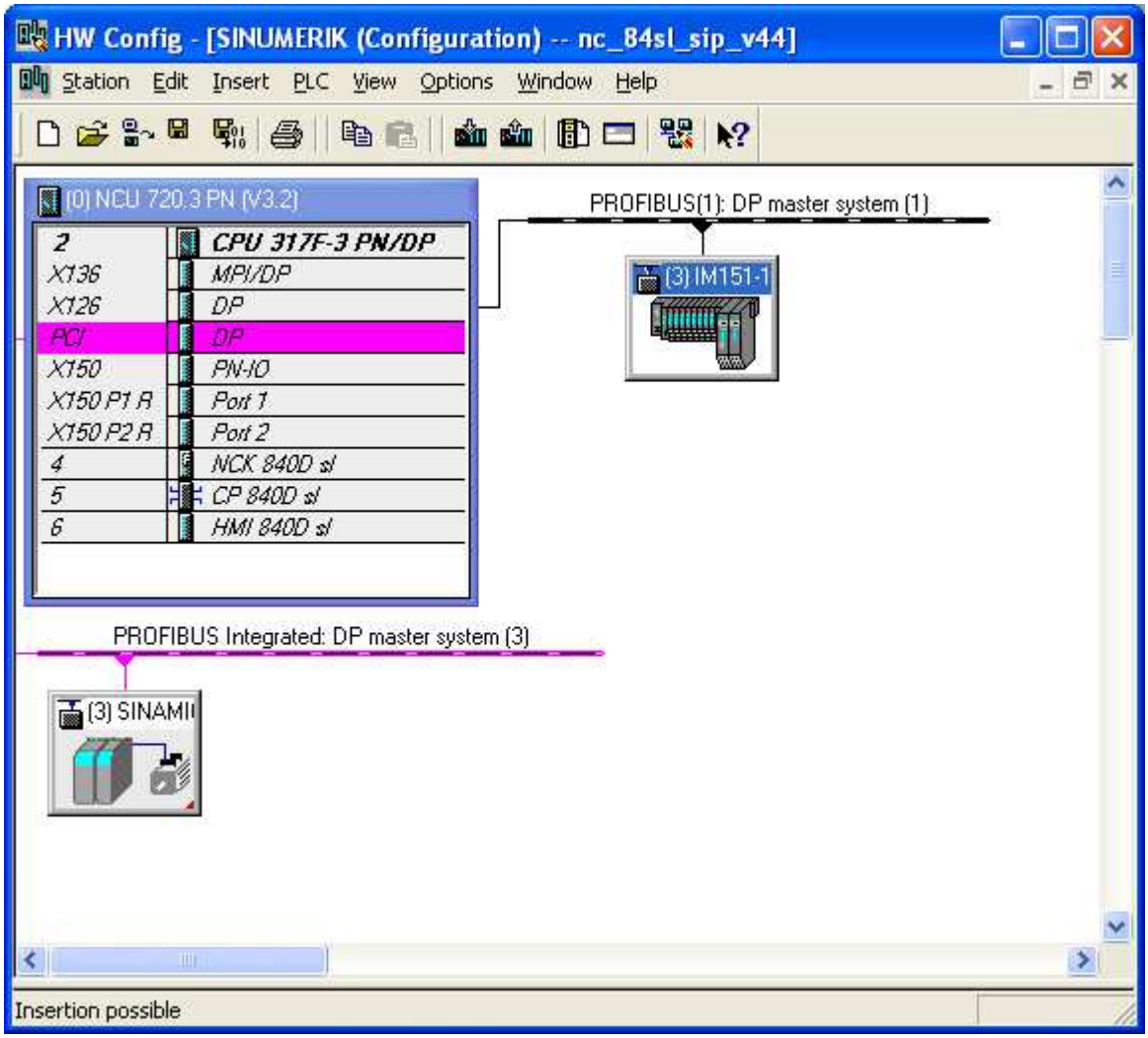
Note

The lowest switch of the IM is irrelevant for the address!



Check

The hardware project now contains an IM151-1 HF with PROFIBUS address 3.



2. Fit the PM, DI, and DO modules

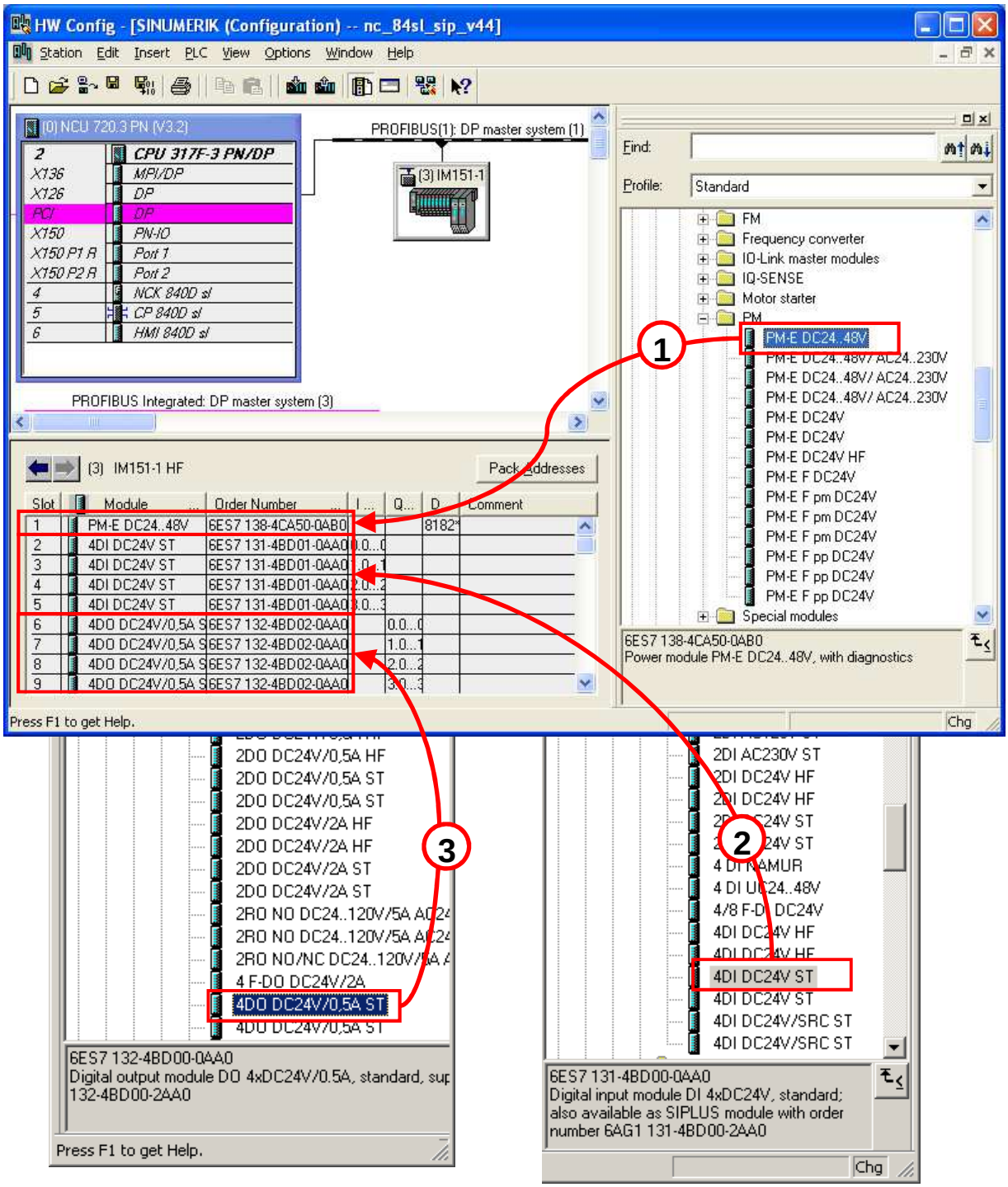
Now fit the slots of the IM with the existing digital input modules DI and digital output modules DO and the power module PM.

You will find these in the subfolders **PM, DI,** and **DO** of the IM

Specifically, these are:

- a power module: **1x** **PM-E DC24..48 V** **(newer devices)**
  or 1x PM-E DC 24 V (older devices)
- four input modules: **4x** **DI DC24V ST**
- four output modules: **4x** **DO DC24V /0.5A ST**

Make sure that you select the correct module version!

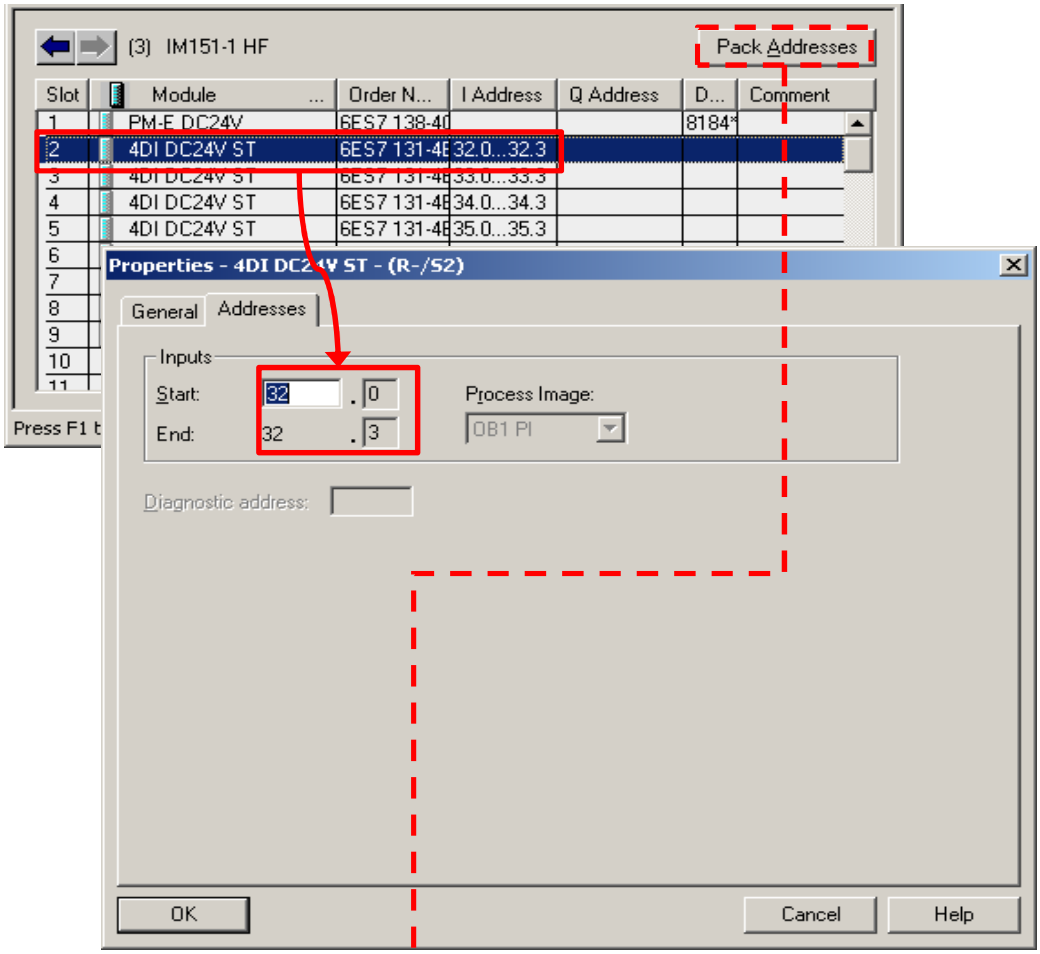


3. Set the I/O addresses

For the DI modules, set the addresses to IB 32 and IB 33,
and for the DQ modules to QB 32 and QB 33.
In the screenshot, you can see the setting for the first DI module; the other modules can be parameterized analogously.

Reason

The address space as of IB 0 / QB 0 is already assigned to the machine control panel!



Pack addresses

You can even just enter the address of the first input or output module and use the "Pack addresses" button for the remaining modules.
Select all DI or DO modules.

At the end, the addresses should look like this:

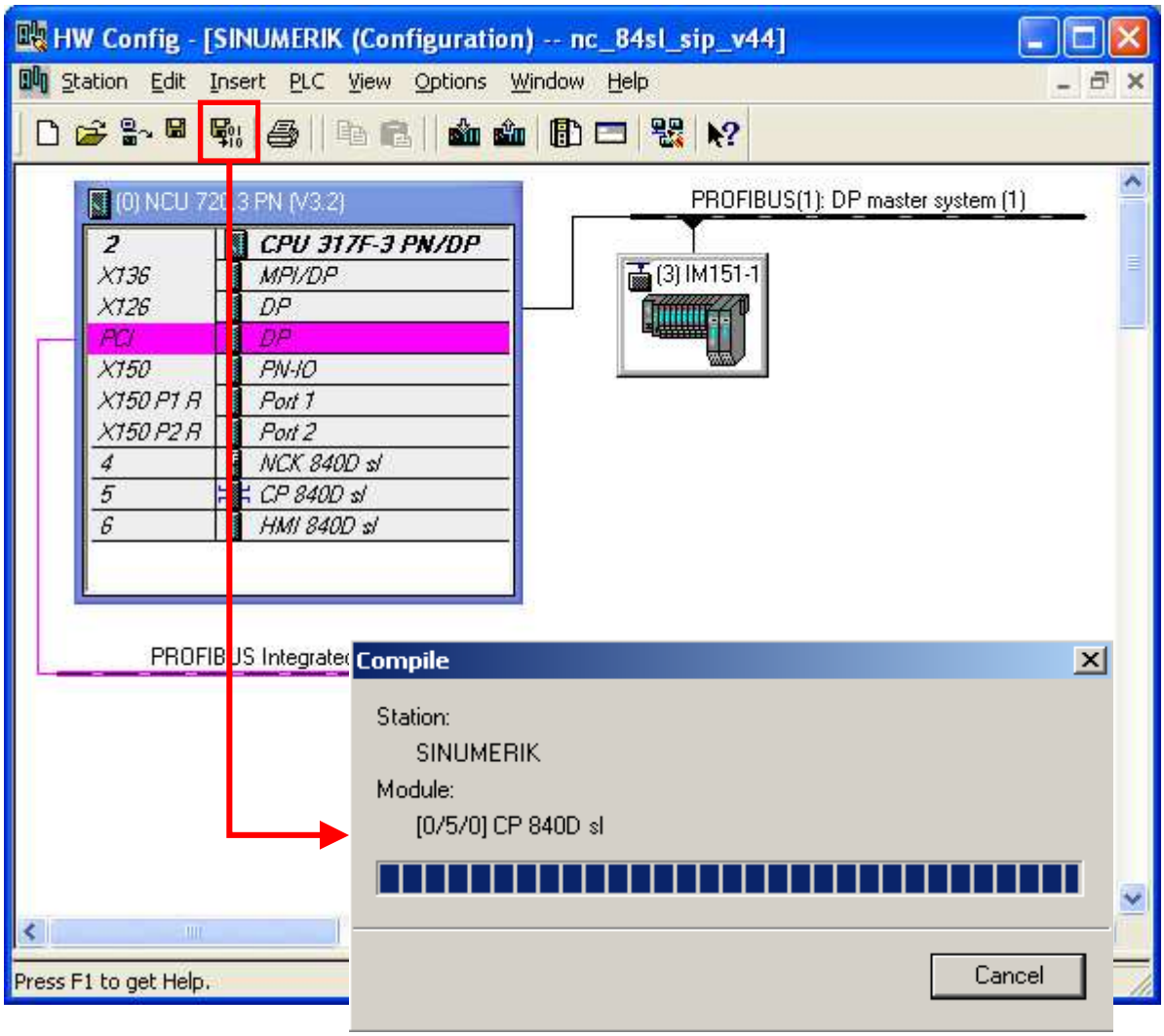
1	PM-E DC24..48V	6ES7 138-4CA50-0AB0		
2	4DI DC24V ST	6ES7 131-4BD01-0AA0	32.0...32.3	
3	4DI DC24V ST	6ES7 131-4BD01-0AA0	32.4...32.7	
4	4DI DC24V ST	6ES7 131-4BD01-0AA0	33.0...33.3	
5	4DI DC24V ST	6ES7 131-4BD01-0AA0	33.4...33.7	
6	4DO DC24V/0,5A ST	6ES7 132-4BD01-0AA0		32.0...32.3
7	4DO DC24V/0,5A ST	6ES7 132-4BD01-0AA0		32.4...32.7
8	4DO DC24V/0,5A ST	6ES7 132-4BD01-0AA0		33.0...33.3
9	4DO DC24V/0,5A ST	6ES7 132-4BD01-0AA0		33.4...33.7

To use the configuration you have created, you must now save it in the project and then download it to the control.

4. Save and compile

Save the configuration by clicking

> Save and compile



PLC stop

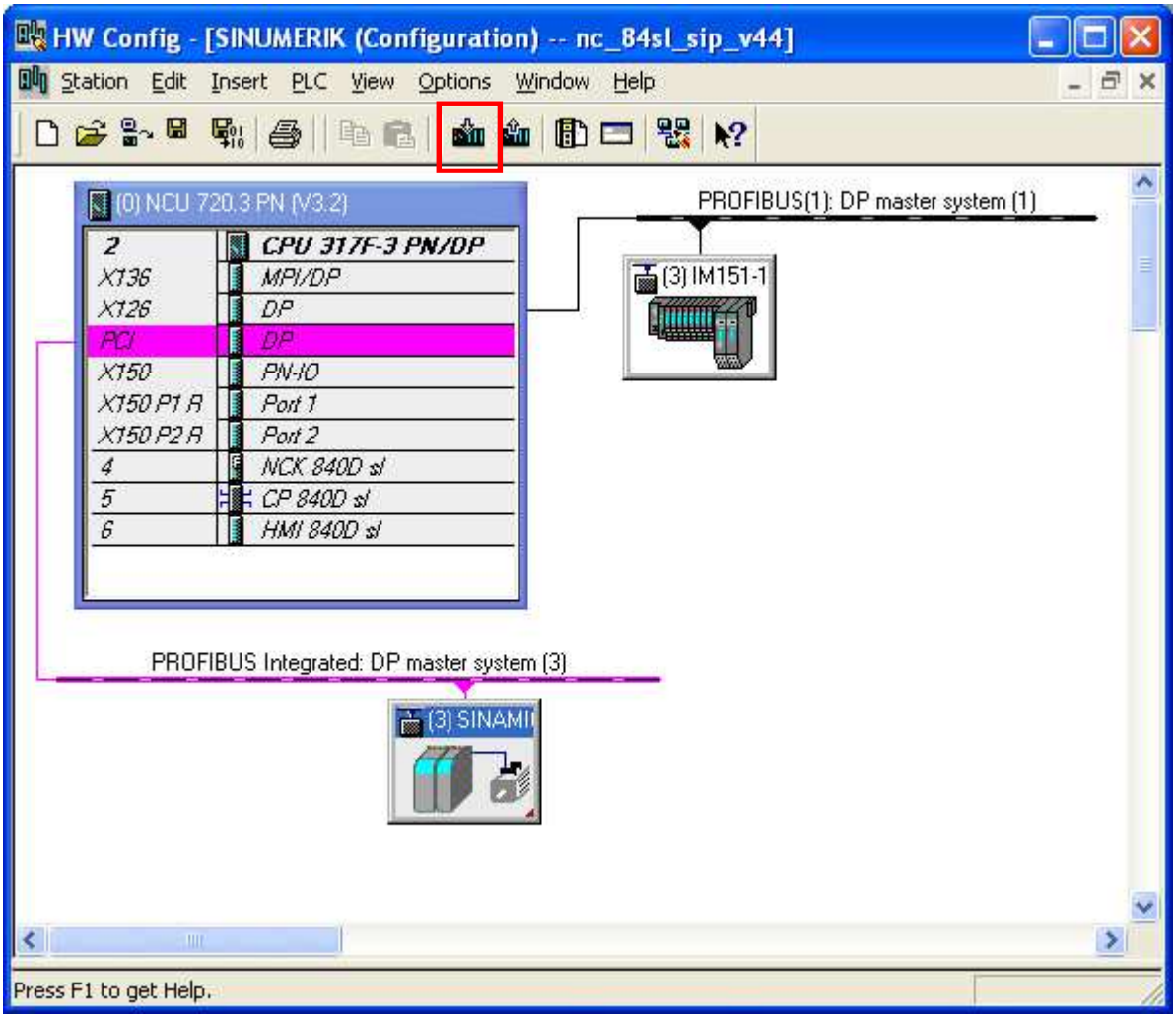
Switch the PLC to Stop by turning the rotary switch to the setting

- **PLC -> 2**

5. Download to the CPU

Download Load the configuration to he PLC by clicking

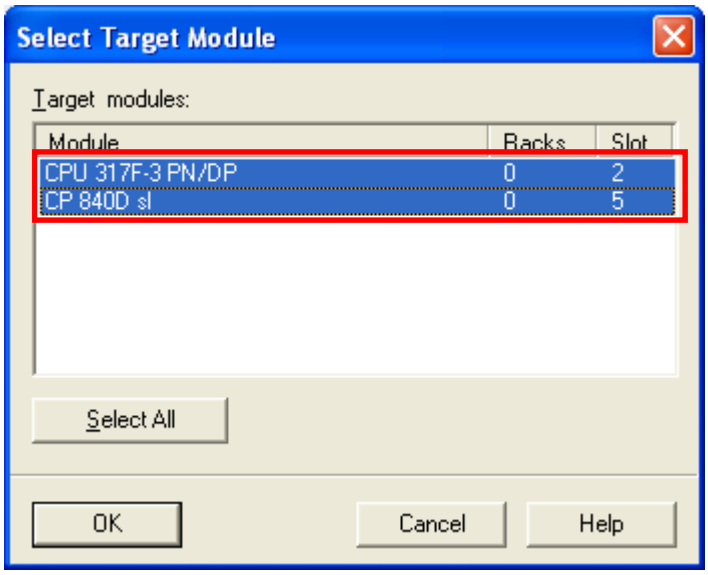
-> Download to module



Select PLC and CP

In the next dialog box, select both modules:

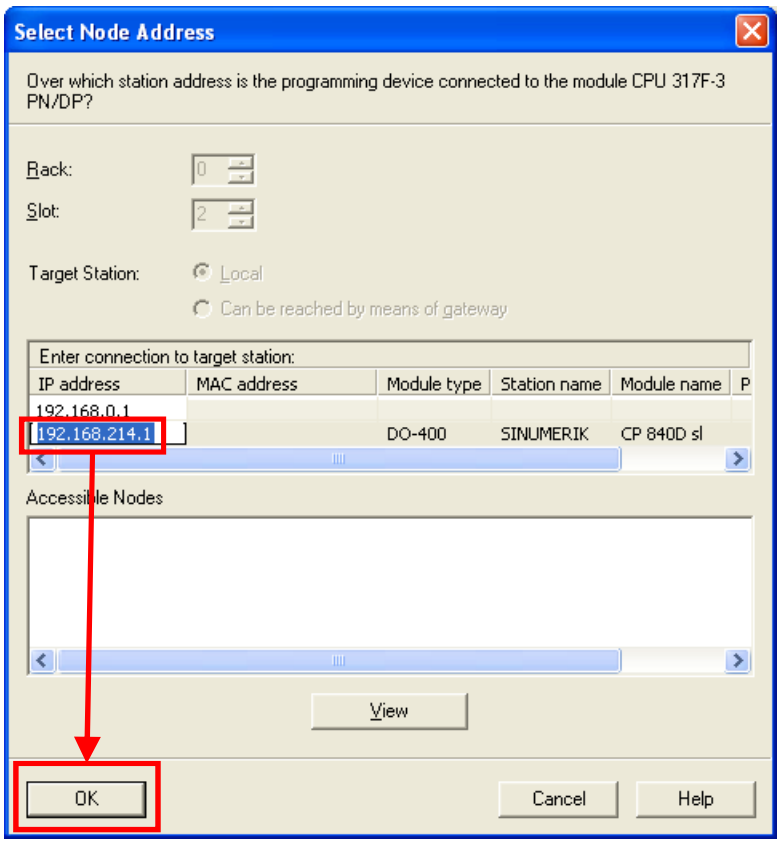
- **PLC 317F-3 PN/DP**
- **CP840D sl**



Destination address

Select the IP address

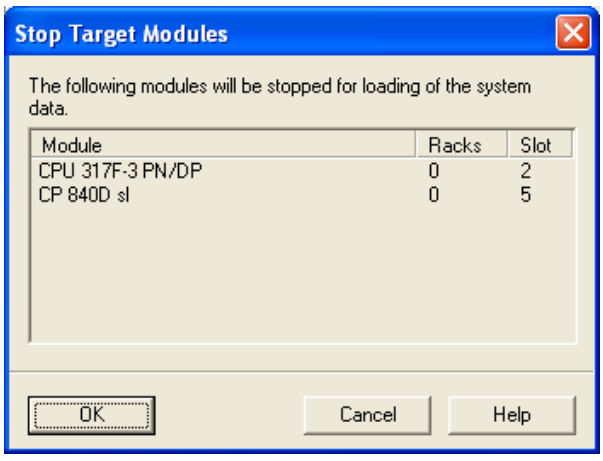
- **192.168.214.1** or
- **192.168.215.1** (on initial commissioning, CP address not yet changed)



Stop CP

Stop the CP in the dialog box

-> Stop target module



Leave PLC in Stop

Leave the 'PLC' rotary switch set in position 2 = STOP!

Exercise: Basic program: Copy to project

Essentially, the PLC basic program on an SINUMERIK control consists of 3 parts. The commissioning part (FB1), the cyclic part FC2, and the interrupt processing part FC3. Extensions for controlling a machine control panel are also required, for example, along with various control and enable signals. This combination of blocks and interface signals is needed to operate the machine. The basic program is supplemented by the machine-specific control logic for tool changers, protective doors, etc.

Objective

Copy the basic program library to the project as a starting point for your own machine programs.

Basic program

Open the basic program via the menu option

-> File -> Open

Select tab

- **Libraries**

and the basic program library there

- **bp7x0_ab** (as of = version; example V4.5: bp7x0_45)

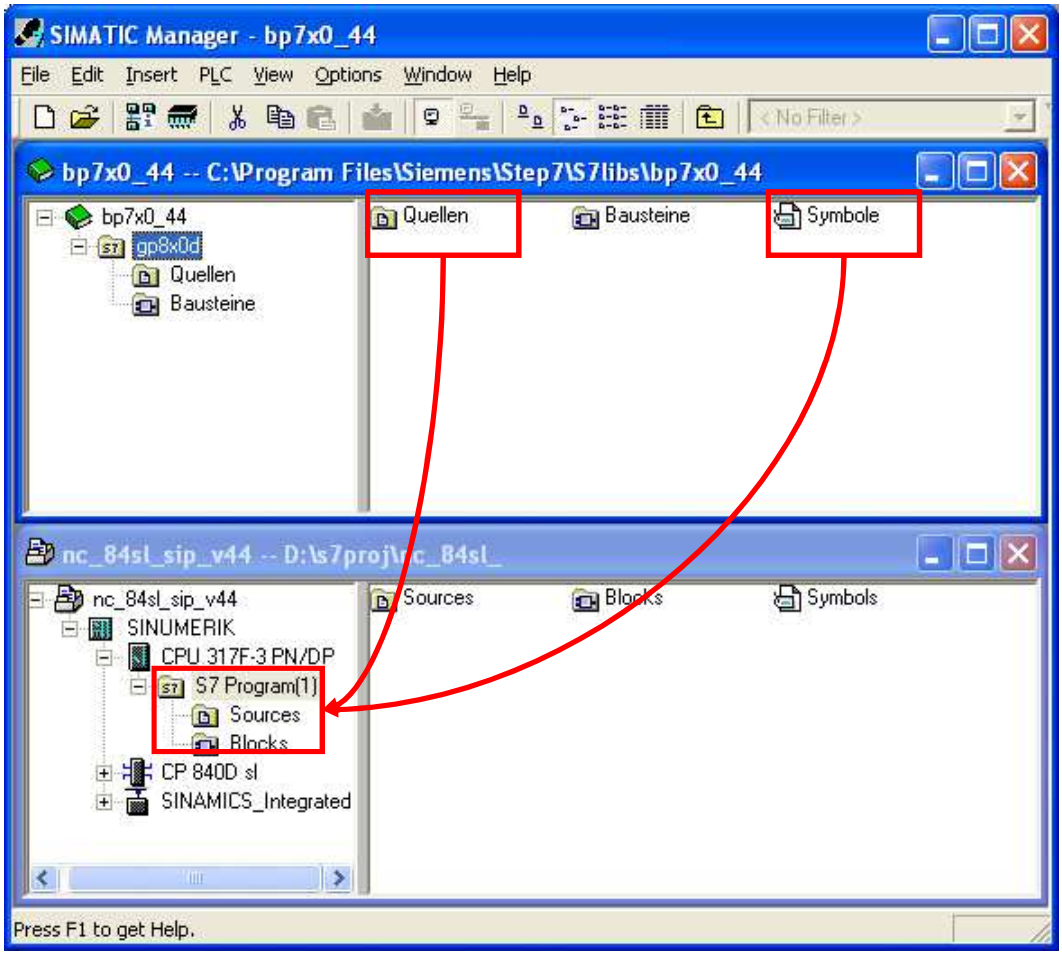
Arrange

Make both windows visible via

-> Windows-> Arrange -> Vertical

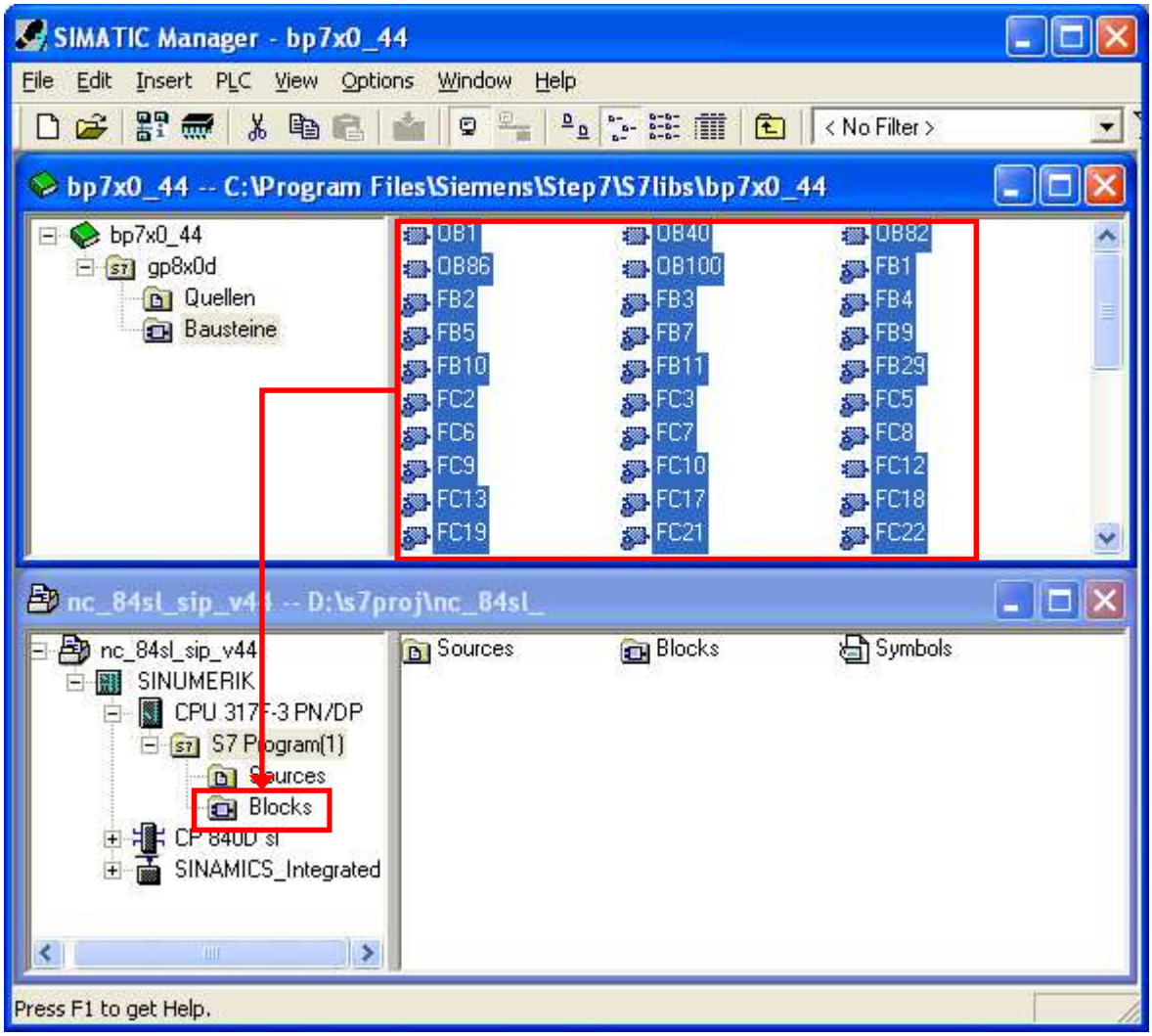
Copy sources, symbols

Copy the **Sources** and **Symbols** objects to your S7 program.



Copy blocks Open the Blocks folder, select all and copy the blocks to your blocks folder.

Notice Do not copy the entire block folder, otherwise your system data will disappear from the HW Config!



Close You can now exit the basic program library.

Exercise: OB1 and OB100 for MCP

OB 100 contains the commissioning-specific part of the basic program.

OB100 with MCP

The following parameters must be preassigned in OB100 for the MCP:

- **MCPNum** **:=** 1
- **MCP1In** **:=** P#I 0.0
- **MCP1Out** **:=** P#Q 0.0
- **MCP1StatSend** **:=** P#Q 8.0
- **MCP1StatREc** **:=** P#Q 12.0
- **MCP1BusAdr** **:= 192 (IE) /** 6 (PROFIBUS)
- **MCPBusType** **:= B#16#55 (IE)** / B#16#33 (PROFIBUS)

This assignment is suitable for the training rack with 3 motors.

OB100 without MCP

In this case just set

- **MCPNum** **:= 0**

Leave the other parameters set in the default settings.

OB 1 contains the cyclic part of the basic program.

OB1 with MCP

In OB1, function FC19 is to be called after function call FC2:

- OB1, NW1:

  Call FC 2

- NW2:

  Call FC 19

```
BAGNo      := B#16#1          // Mode group no.
ChanNo     := B#16#1          // Channel no.
SpindleIFNo := B#16#4         // Axis no. Spindle
FeedHold   := DB21.DBX6.0     // Feedrate stop
SpindleHold := DB34.DBX4.3    // Spindle stop
```

Enables

The enables must be supplied for all axes:

- OB1, NW3:

```
A I 32.0          // Switch PLC DI/DO DI0
= DB31.DBX1.5     // Meas. system 1 axis 1
= DB31.DBX2.1     // Controller enable axis 1
= DB31.DBX21.7    // Pulse enable axis 1
=  …              // Ditto other axes (DB32, …, DB34)
```

Exercise: Download blocks

Once the blocks have been edited successfully, they must be copied to the hardware for processing.

Objective

To download blocks, thereby activating PLC communication with the drives.

PLC stop

Switch the PLC to Stop (if it is not yet in Stop).

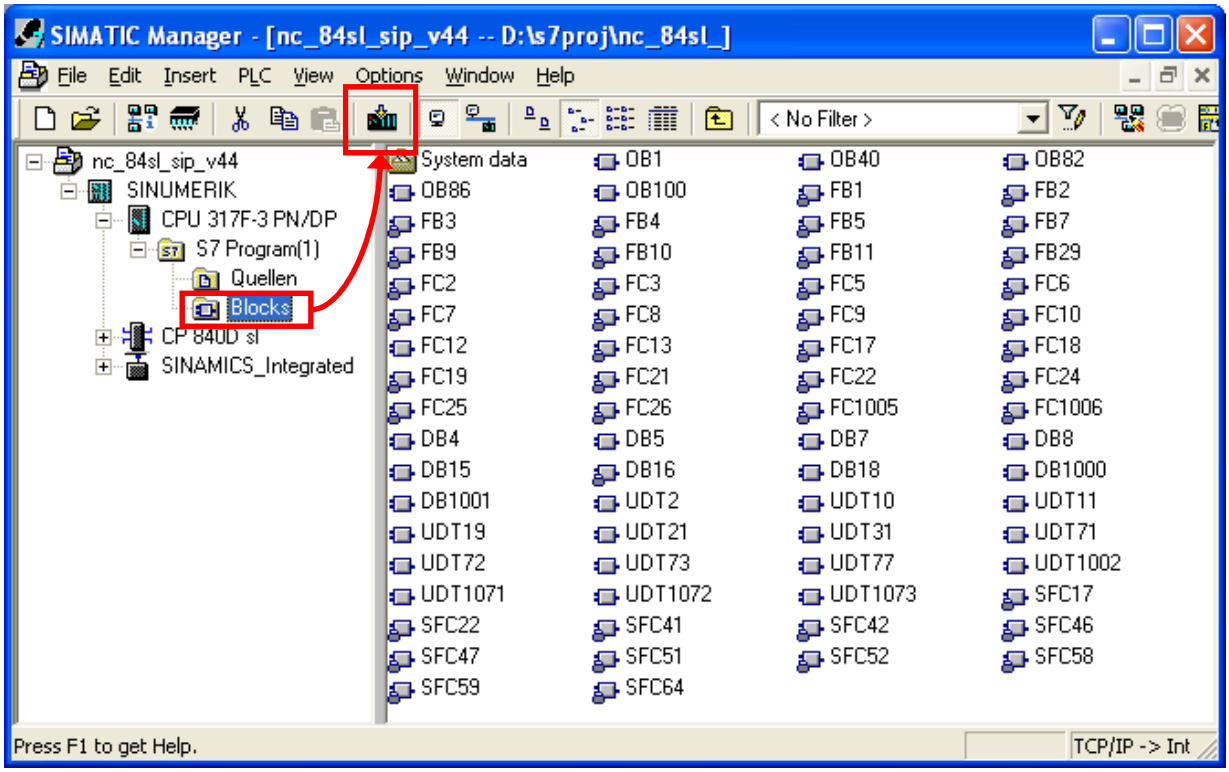
Load PLC

Select the folder

> CPU317F-3 PN/DP -> ... > blocks

and click the button

-> Download



Dialogs

Respond to any prompts as follows:

- **Check ... the sequence** with • **OK**
- **Do you want to load the system data?** with • **Yes**
- **Do you want to replace the system data ... with offline system data?** with • **Yes**
- **.. Do you want to start the module?** with • **No**

Download CP

It is not necessary to download the CP blocks as the only data is system data. The CP blocks were downloaded when the hardware configuration was downloaded.

Start the PLC

Switch the PLC to Run by turning the rotary switch to the setting

- **PLC = 0**

Power On

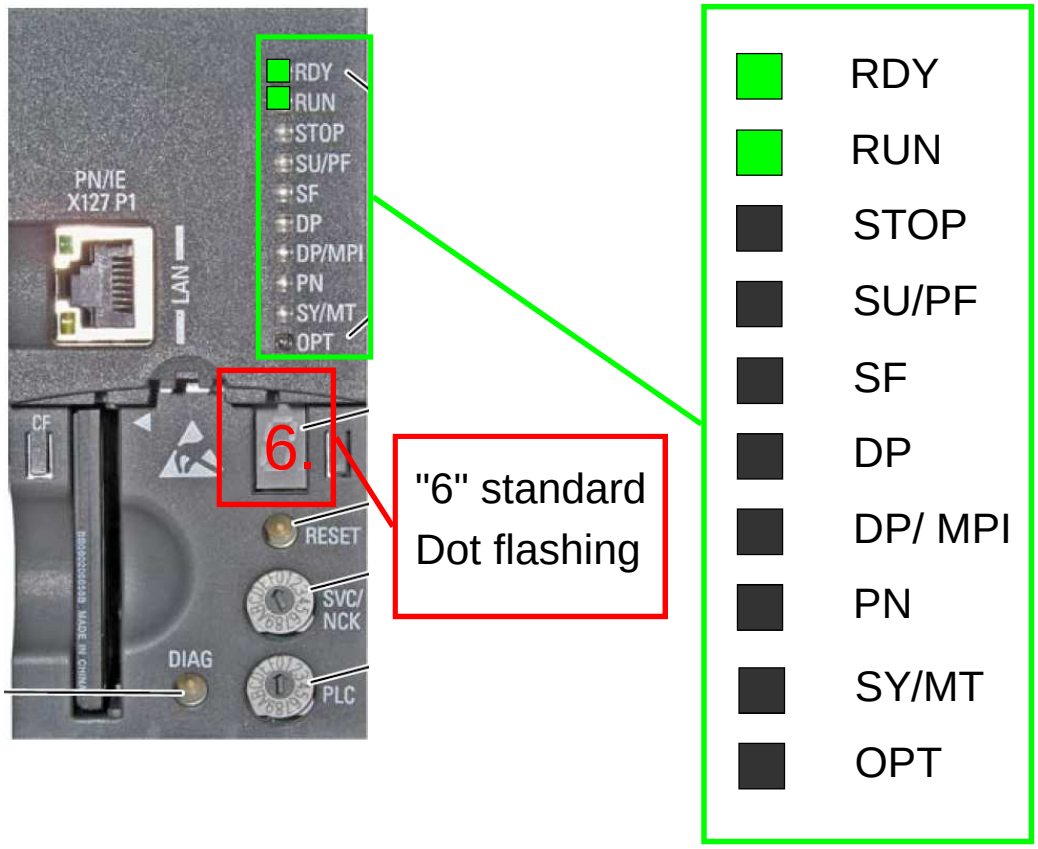
In order for all configuration changes to become effective, switch the 24 V supply for NCU and drives off completely and back on again.

Exercise: Test PLC start-up

The PLC configuration process is completed by checking the function created so far.

Objective To check that PLC start-up has been successful.

NCU 7x0.3 LED standard state:



Check PLC start-up: Check the following points after start-up:

- **No PLC STOP or errors:**
  - LED 'RUN' and 'RDY' green
  - LED 'STOP' off (if yellow: check the diagnostic buffer)
  - LED 'SF' off (if red: check HW Config / PROFIBUS)
- **No alarms:**
  - 2000 'sign-of-life monitoring PLC'
  - 2001 'PLC start-up failed'
- **MCP running** (no LEDs flashing)

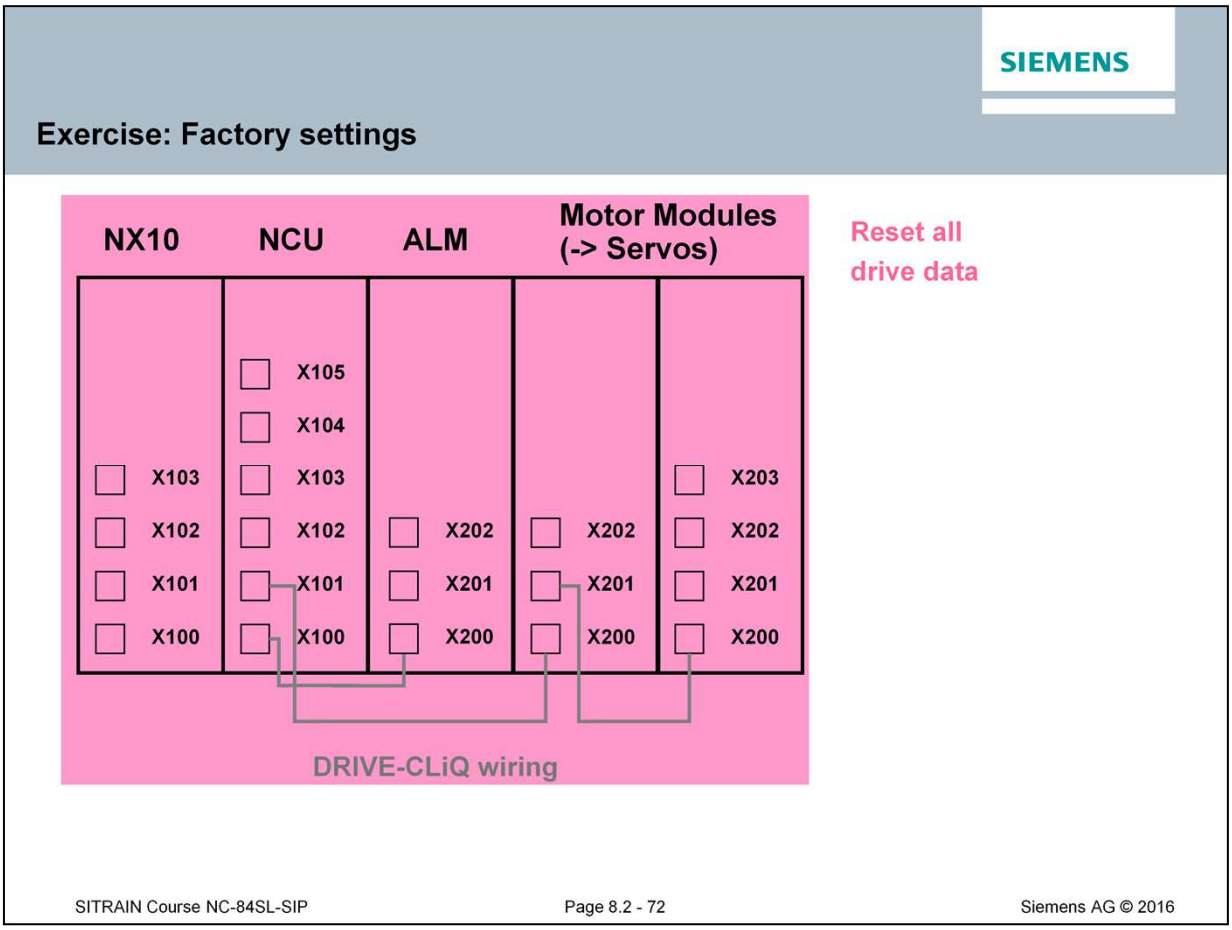
Diagnostics of SINAMICS

To view corresponding SINAMICS alarms in the HMI, set

- **MD 13150 = 3F3Fh** (general MD)

Note Depending on the system software used (6/31 axes), 3 or 4 axes are created in the standard machine data set. Therefore, there might not be a fourth axis and thus no DB34.

Accordingly, the PLC might currently still be in the stop state (check diagnostic buffer).
If it is, we will have to activate the C axis as the 4th axis and spindle.



Objective

To set the control to a defined initial state with the factory settings.

The factory settings must be restored before the device can be configured.

1. Select the factory settings

Select:

> OA: 'Setup' > Softkey: 'Drive system'

> Softkey: 'Factory settings' > Softkey: 'Drive system/drive unit'

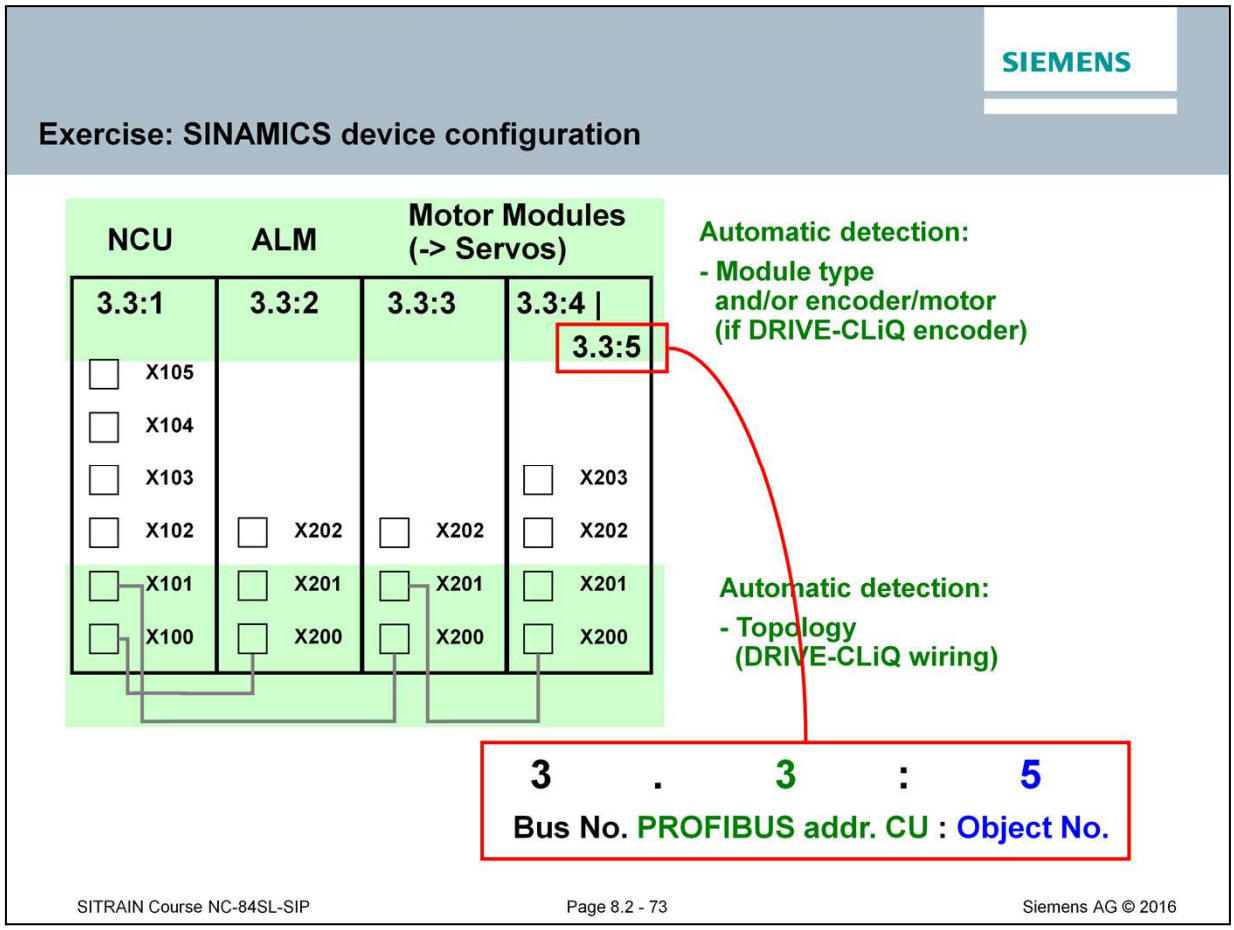
2. Load the factory settings

Confirm the prompt with

> Softkey: 'OK'

Wait for the prompt for automatic device configuration.

This is described in the following exercise:



Objective

As the control does not have any drive data in the factory settings, the first time the "Drive units" softkey is selected, the option to start device configuration is offered.

Requirements

You have completed the "SINAMICS factory settings" exercise.

The DRIVE-CLiQ wiring is as shown above.

Steps

1. Select the softkey: Drive units
2. Perform device configuration
3. Check the topology

Sequence:

1. Softkey: Drive units Select the device configuration with

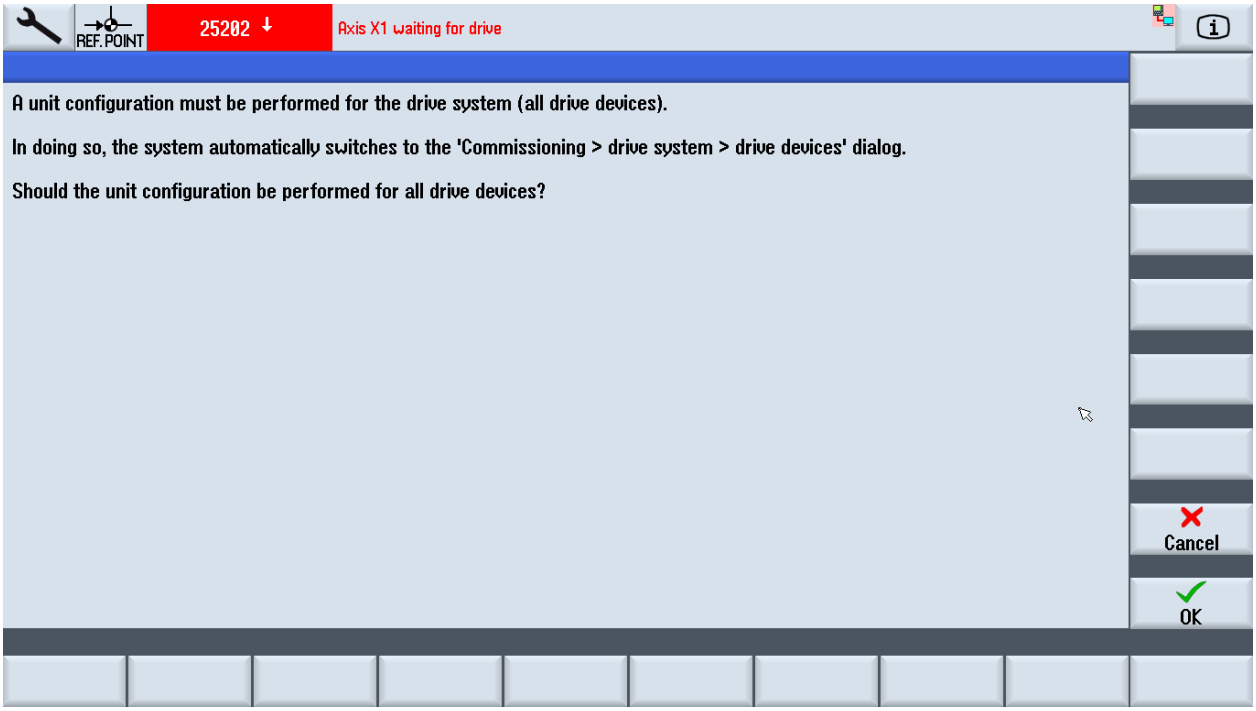
> OA: 'Setup' > Softkey: 'Drive system' -> Softkey: 'Drive unit'

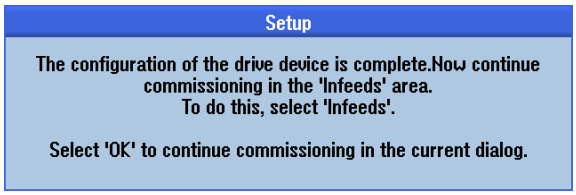
You are prompted to perform automatic device configuration.

2. Perform device configuration

Confirm the individual steps up to saving and reset with

> Softkey: 'OK' or **> Softkey: 'Yes'**





Commissioning of the infeed will only be performed after the topology has been checked. Therefore select **> Softkey: 'OK'**

Result: All components connected via DRIVE-CLiQ have been declared and saved in the target topology.

3. Check the topology

In this step, we are going to test whether the detected topology matches the one that is available and has been configured.

Check that the components were successfully detected under

> OA: 'Setup' > Softkey: 'Drive system' -> Softkey: 'Drive unit'

In the first screenshot opened **> Softkey: 'Configuration',** all detected components are listed.

In the screenshot **> Softkey: 'Topology'**, the connections of the components are listed.

Display options The display in both screens also depends on the

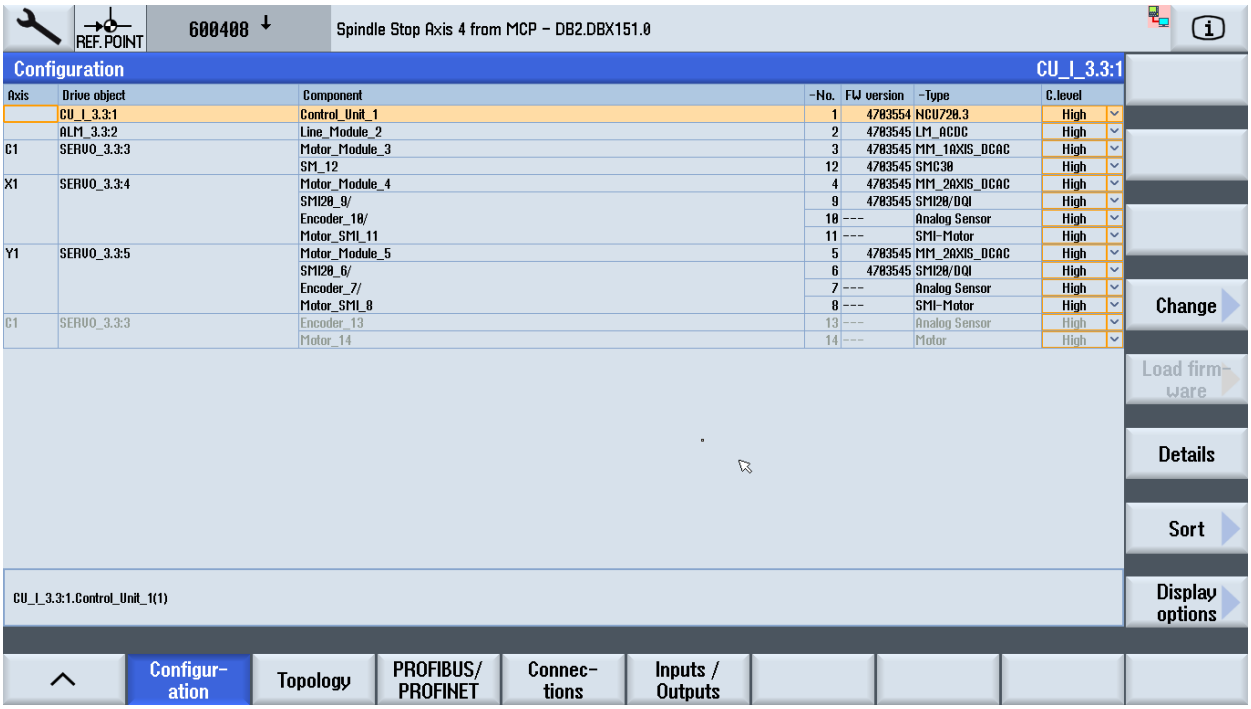
-> Display options

There select

- **Topology = Target topology**
- **Filter = off**

Compare Now check whether all components of your configuration are visible here.

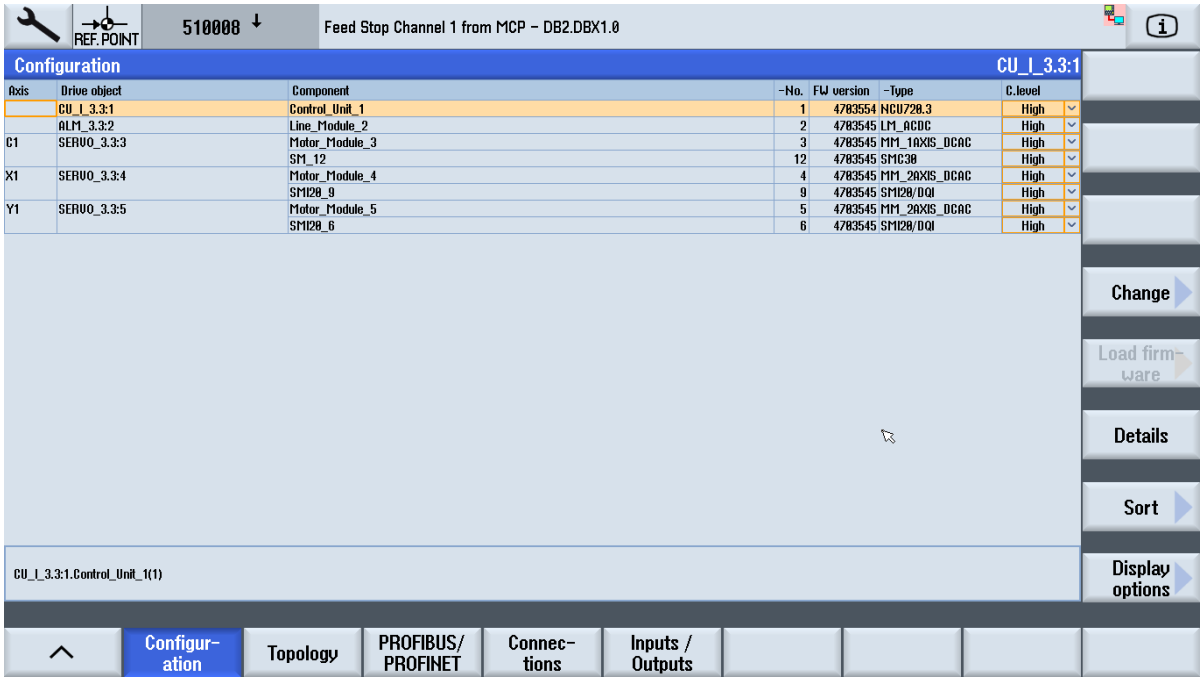
Example configuration:



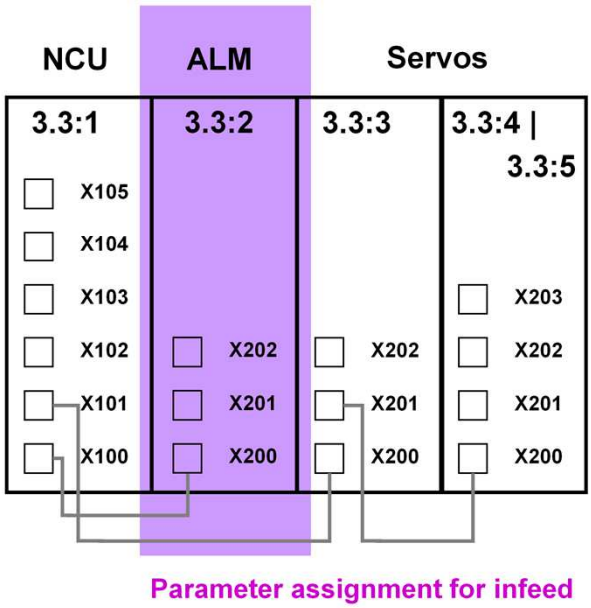
Example topology:

The filter must be set again in the display options.

The encoders are not visible in the topology view.



Exercise: ALM commissioning



Parameter assignment for infeed

SITRAIN Course NC-84SL-SIP Page 8.2 - 77 Siemens AG © 2016

Objective To practice commissioning the line infeed (Active Line Module = ALM).

Requirement You have completed the "SINAMICS device configuration" exercise.

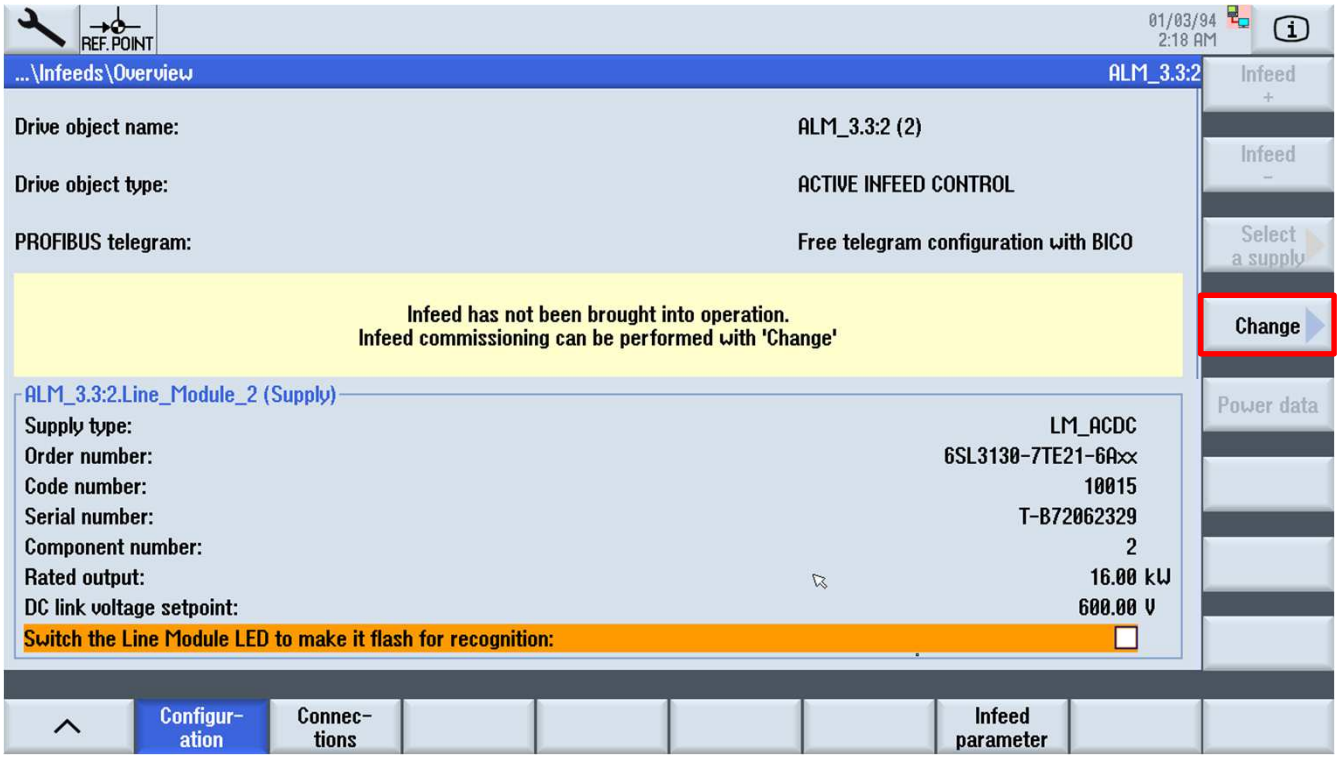
Steps

1. Softkey: Infeeds
2. Select line filter
3. Preselect line identification
4. Drive unit line supply voltage
5. Line contactor control
6. Summary
7. Save
8. NC DI0 switch
9. Power ON
10. Line identification

Sequence:

1. Softkey: Infeeds Select the menu:

OA: 'Setup' > Softkey: 'Drive system' > Softkey: 'Infeeds'



The text in the event dialog box indicates that the infeed has not yet been commissioned.

To continue, select

> Softkey: 'Change'

2. ALM and line filter

The ALM has been detected automatically (ALM 16 kW).

The appropriate Active Interface Module (AIM 16 kW) is preassigned to the selected line filter. It is suitable for the training rack.

Further modules If there is a brake control module/Basic Line Filter on the machine, enter the relevant information here. In our training rack, none of these exist.

3. Preselect line identification

In the screenshot, the information about the line/DC-link identification has already been entered.
Line/DC-link identification is performed at the end of commissioning, when the voltage is connected to terminal EP X21.1 (ALM) and ON/OFF1 X122.1 (NCU) for the first time.

4. Drive unit line supply voltage

The drive unit line supply voltage is correctly preset to 400 V.

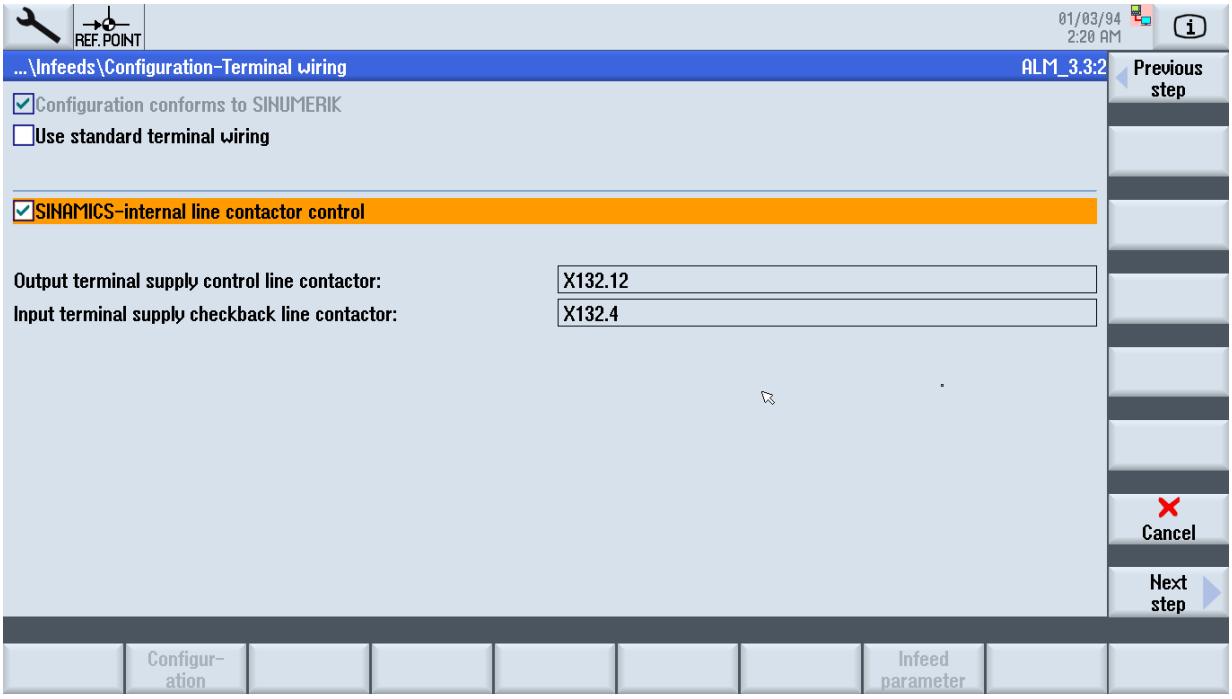
Final installation location

After delivery of the machine, the line identification, possibly with a drive unit supply voltage adapted to the final installation location, must be performed again.

Check that the EP terminal is not energized
(disconnect X21 (ALM) if necessary).

5. Line contactor control

In the screenshot, the SINAMICS line contactor control is being activated and configured. The proposed BICO inputs and outputs correspond to that wired on the training rack.



6. Summary Following a summary:
you continue with **> Softkey: 'Completed'**

7. Save All that remains is to save the data: **> Softkey: 'Yes'**

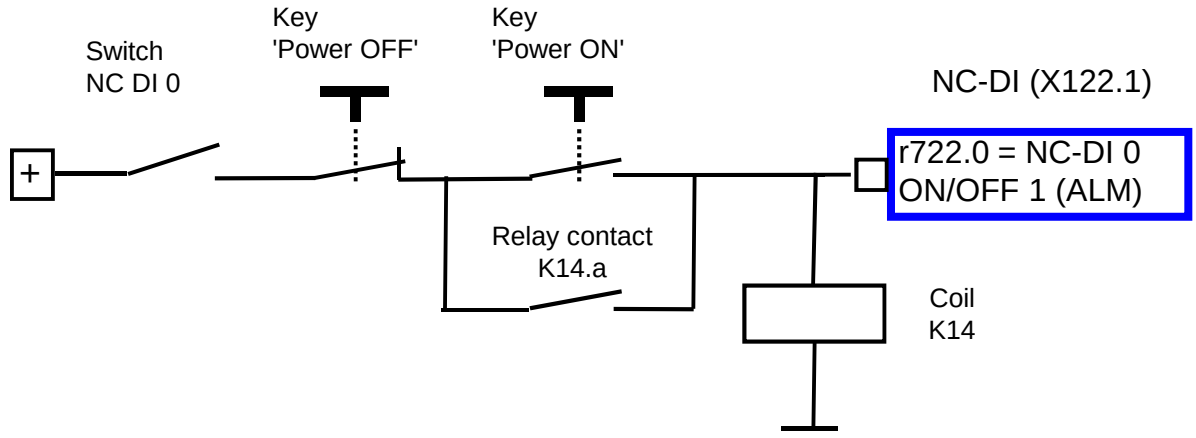
8. Switch NC DI0 Switch on the **NC DI 0** switch:



9. Power ON Now switch on the power with the '**Power ON**' button.



This switches the contactor relay K14 to modal.
As a result, signal NC DI 0 = ON/OFF 1 (ALM) is activated.
The line contactor is then switched via NC DO14.



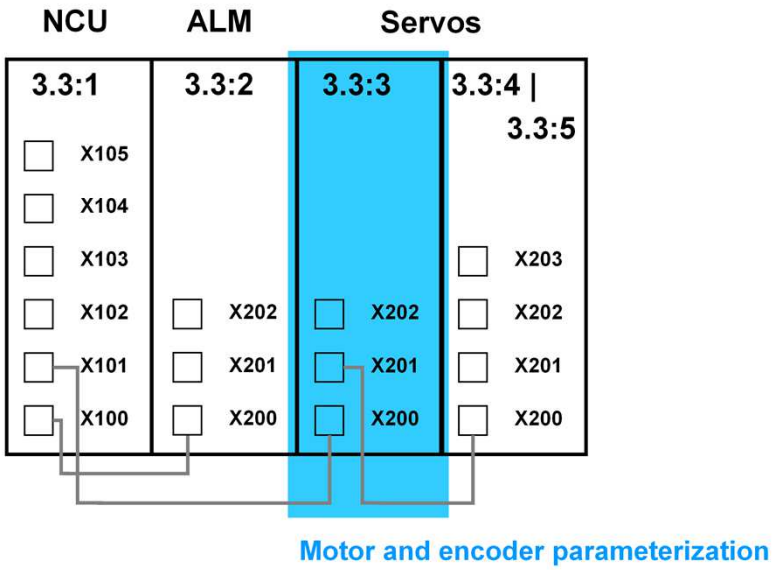
10. Line identification

Line/DC-link identification is performed at this point.

This is displayed with message 206400 and the ALM makes a "buzzing" noise.

Identification is complete when message 206400 disappears.

Exercise: Commissioning of Servo_3.3:3 with the Drive Wizard



SITRAIN Course NC-84SL-SIP Page 8.2 - 81 Siemens AG © 2016

Objective

To practice commissioning the servo on the Single Motor Module (Servo_3.3:3 on device with ALM).

Requirement

You have completed the "SINAMICS device configuration" exercise and, if applicable, the "ALM commissioning" exercise.

Steps

1. Start the Drive Wizard
2. Select servo
3. Power unit
4. Motor type: induction, rotary
5. Holding brake
6. Motor data
7. Optional motor data
8. Sensor Module
9. Select encoder from list
10. Control type and telegram
11. BICO wiring
12. Data overview
13. Save
14. Reset

Sequence:

1. Start the Drive Wizard

Start the Drive Wizard:

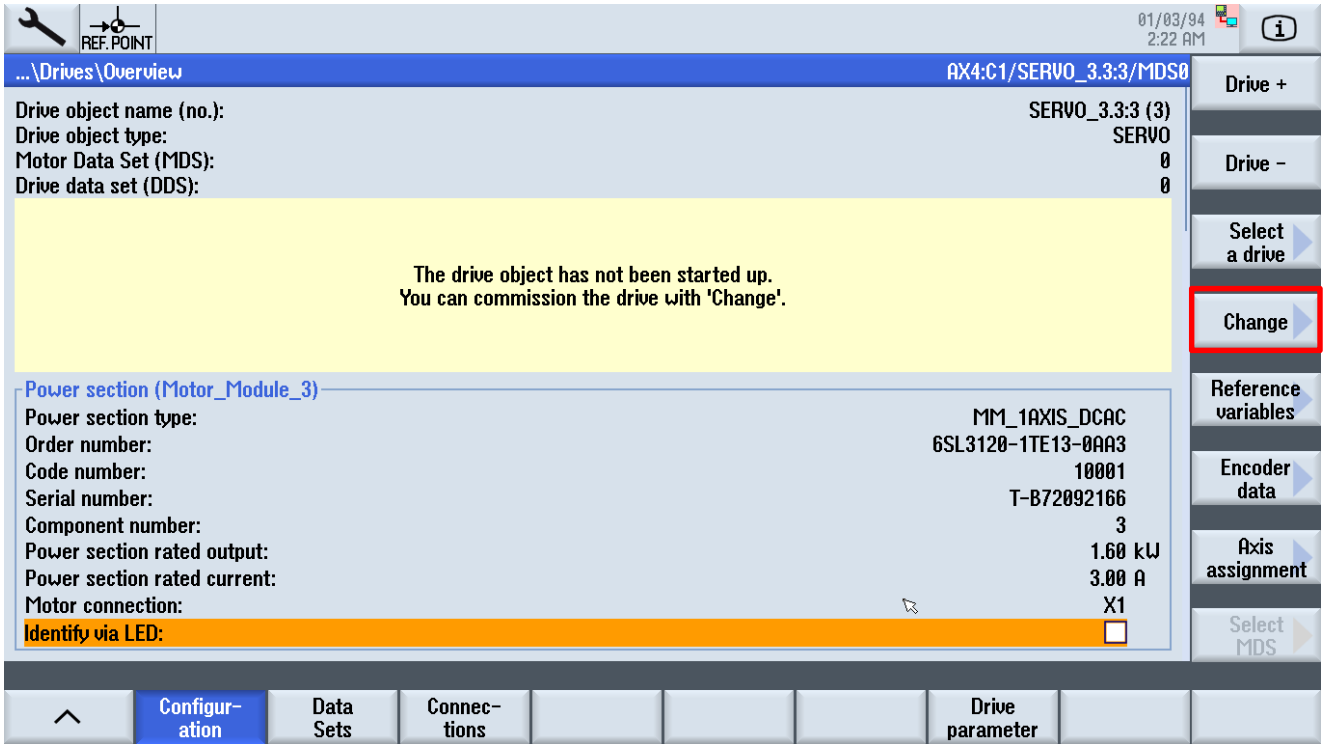
> OA: 'Setup' > Softkey: 'Drive system' -> Softkey: 'Drives'

2. Select Servo_3.3:3

Select the first drive: **Servo_3.3:3**.

Switch off the power or NC DI 0.

Press **> Softkey: 'Change'**



3. Confirm power unit

The power unit is detected automatically. Therefore, you simply need to confirm it. Among the function modules offered, select none.

> Softkey: 'Next step'

4. Motor type

You can now choose between "Select standard motor from list" and "Enter motor data".
Because our motor is not in the database

- **Enter the motor data** correctly

For the motor, select:

- **Induction motor (rotating)**

> Softkey: 'Next step'

5. Holding brake Next, confirm that there is no holding brake:

> Softkey: 'Next step'

6. Enter the motor data

Now enter the motor data as shown in the screenshot:

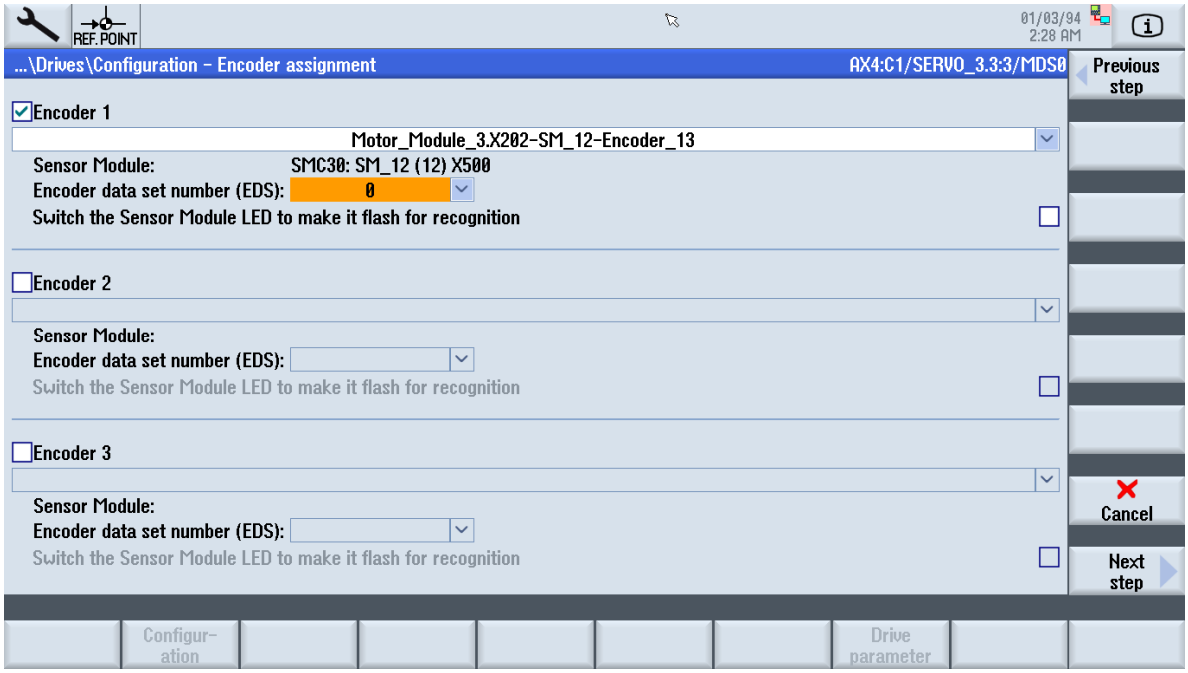


> Softkey: 'Next step'

7. The optional motor data do not have to be entered

> Softkey: 'Next step'

8. Sensor Module Check that the first encoder is selected. Inputs are not necessary here.



> Softkey: 'Next step'

9. Select encoder from list

Select the encoder with the code number from the list

3001: 1024 HTL A/B R

Select a motor encoder

Encoder type	Code number
20000 nm, 1 Vpp, A/B R distance-coded	2111
40000 nm, 1 Vpp, A/B R distance-coded	2112
16000 nm, 1 Vpp, A/B, EnDat, resolution 100 nm	2151
1024 HTL A/B R	3001
1024 TTL A/B R	3002
2048 HTL A/B R	3003
1024 HTL A/B	3005
1024 TTL A/B	3006
2048 HTL A/B	3007
2048 TTL A/B	3008
1024 HTL A/B unipolar	3009
2048 HTL A/B unipolar	3011
2048 TTL A/B R, with sense	3020

> Softkey: 'Next step'

10. Control type and telegram

Leave the default control type:

- **Speed control (with encoder)**

and the telegram unchanged

- **SIEMENS telegram 136, PZD-15/19**

-> Softkey: Next step

11. BICO wiring

Next, confirm the SINUMERIK-compliant BICO interconnection for the terminal wiring:

> Softkey: 'Next step'

12. Summary

Following a summary:

you continue with

> Softkey: 'Completed'

13. Save

Answer the following prompt to save the data with

> Softkey: 'Yes'

14. Reset

Finally, perform a Power On Reset (PO) with

> Softkey: 'Reset (po)'

Connect enables

Reconnect the 2 enables to NC DI0 and DI1:

> Simulator: NC-DI/DO: DI 0 = ON and DI 1 = ON

Switch on power

Switch on the power:

> Simulator: Power ON

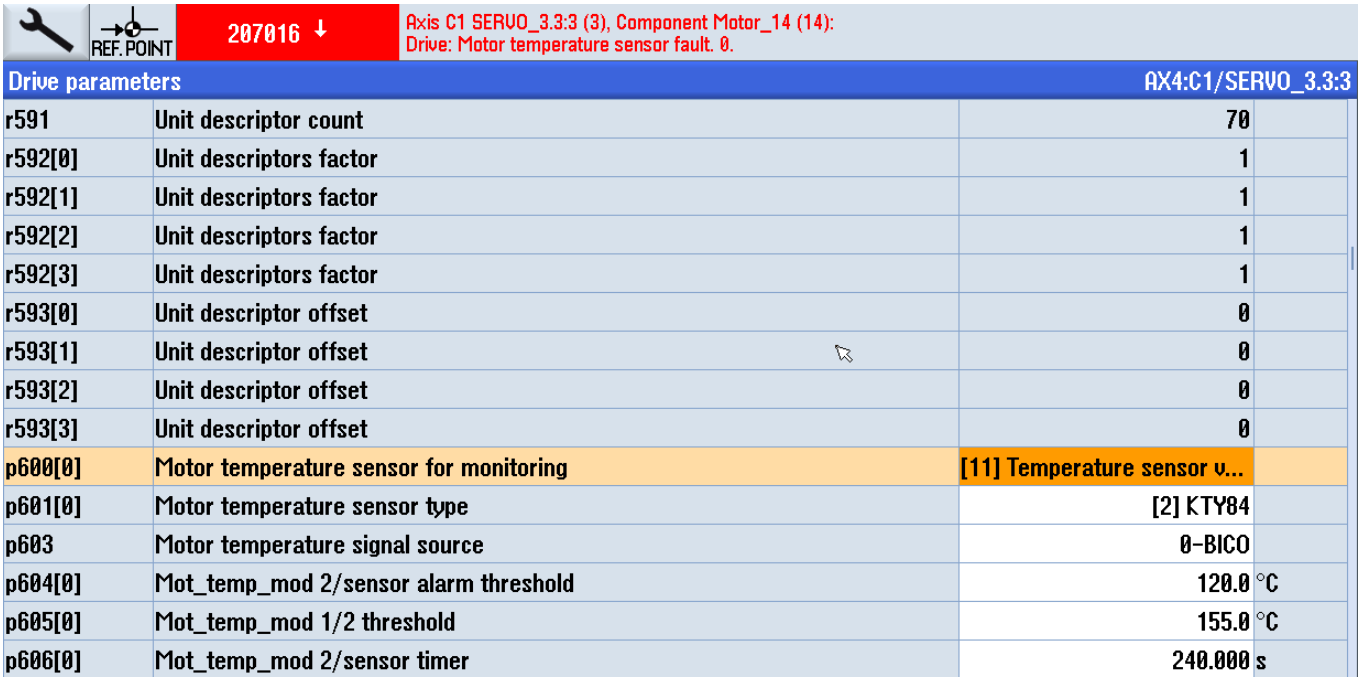
Correct the parameters

Because the temperature sensor is not wired via the encoder-1 cable but to the Motor Module, p600 must be changed:

> OA: 'Setup' > Softkey: 'Mach. data' > Softkey: 'Drive parameters'

with **> Softkey: 'Drive +/-'** select **Servo 3_3_3.**

Set **p600=11** (temp. sensor via Motor Module)



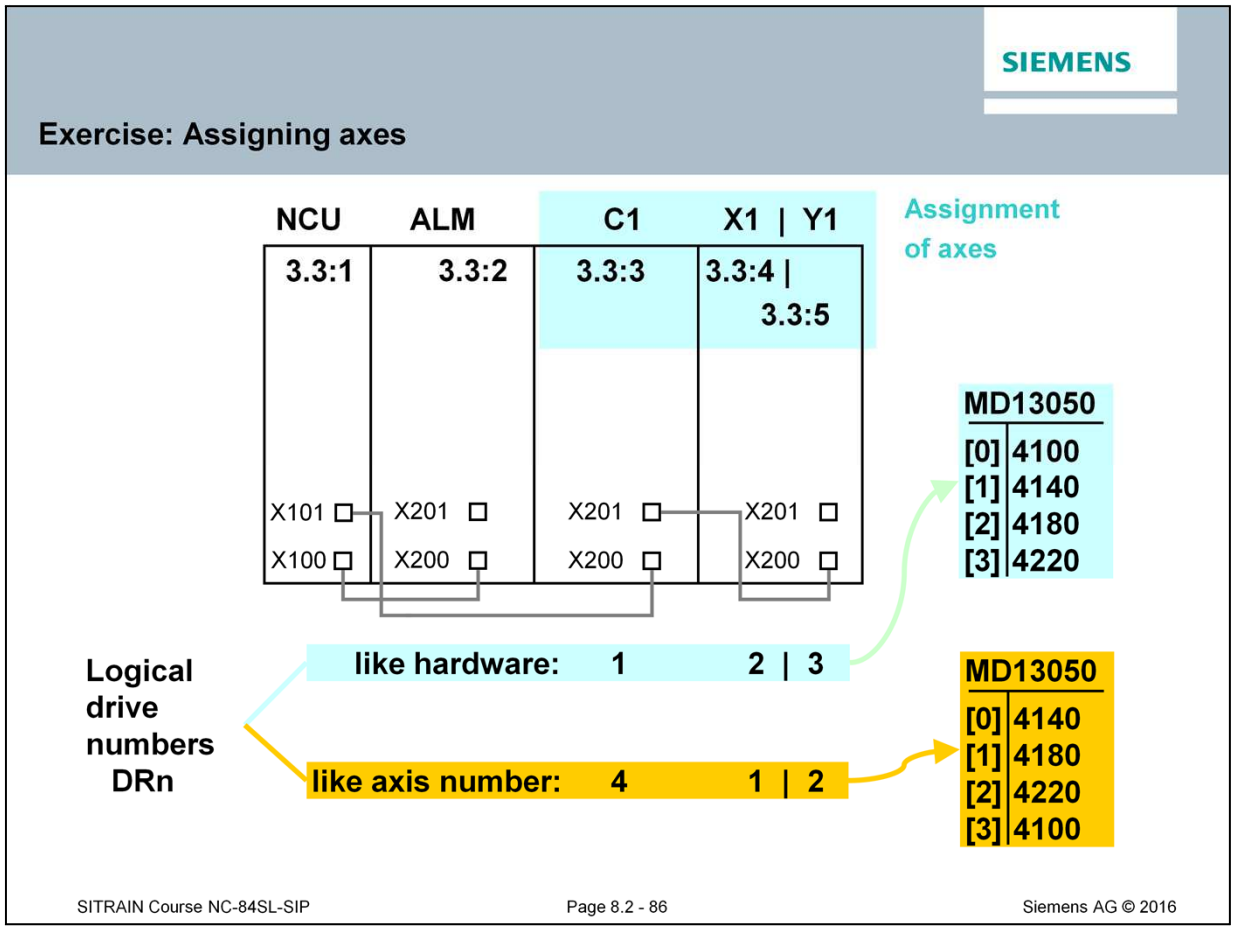
However, an incorrectly parameterized temperature sensor connection will only be noticed after assignment:

> Alarm 201716: Drive temperature sensor fault!

Save

Save the modified data:

> Softkey: 'Save/Reset' > Softkey: 'Save' > Softkey: 'Current drive obj.'



Objective

Starting with the default settings of MD 30110 / 30220 … and 13050, we will create a drive configuration for the course data record from this menu without touching the MD 30110 / 30220 … and 13050.

Requirement

- Machine data set to default values:

Axis	**X**	**Y**	**Z**	**C**
MD 30110	1	2	3	4
MD 30130	0	0	0	0
MD 30220[0]/[1]	1	2	3	4
MD 30240	0	0	0	0
Index Idx	**0**	**1**	**2**	**3**
MD 13050[Idx]	4100	4140	4180	4220

- Drive device commissioning (topology match) is already complete

Steps

1. Preset MD values (see above) or result after standard commissioning
2. Select menu: 'Axis assignment'
3. Select axes and encoders
4. Check the values of MD 30110/30220 … and 13050, **13060**
5. Optional additional exercise: Select logical drive number
6. Optional additional exercise: Check the values of MD 30110/30220 … and 13050 plus axis motion

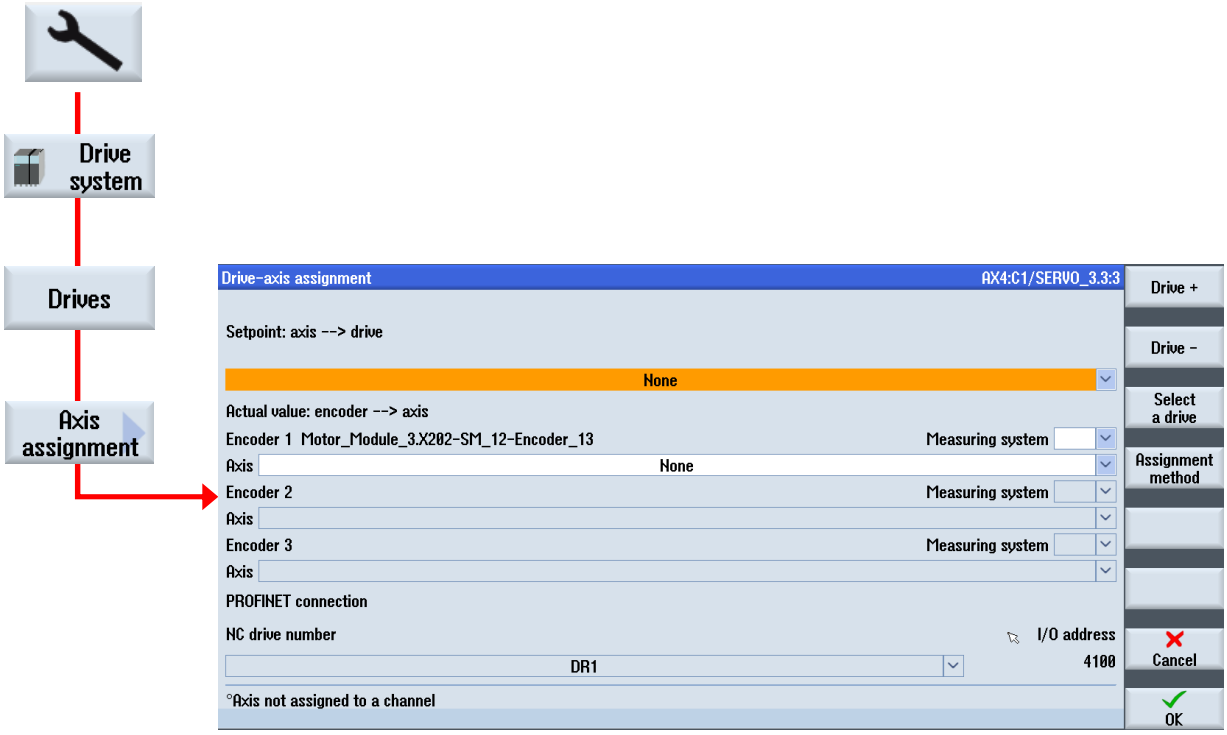
Sequence:

1. Preassign MD Assign values to or check the following machine data:

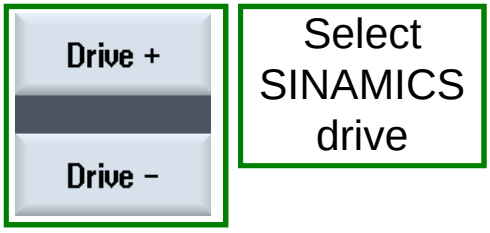
Axis	**X**	**Y**	**Z**	**C**
MD 30110	1	2	3	4
MD 30130	0	0	0	0
MD 30220[0]/[1]	1	2	3	4
MD 30240	0	0	0	0
Index Idx	**0**	**1**	**2**	**3**
MD 13050[Idx]	4100	4140	4180	4220

2. Select menu: 'Assign axis'

- Select the following menu:
  > OA: 'Setup' > Softkey: 'Drive system'
  > Softkey: 'Drive' > Softkey: 'Axis assignment'



- Select the drives one after another using softkeys:
  Softkey: 'Drive +' / 'Drive –'
  Softkey: 'Change'
  and make the changes described on the following pages.

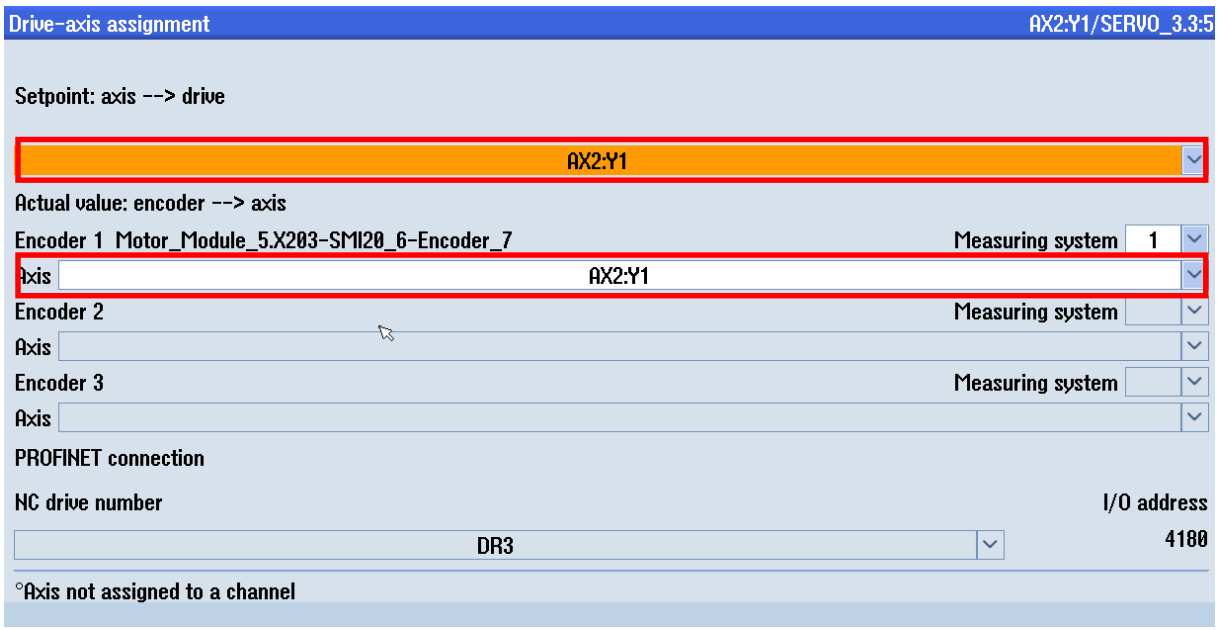


3. Select axes and encoders

First enter the setpoint and actual value assignments described below:

Drive:	**Servo_3.3:3**	**Servo_3.3:4**	**Servo_3.3:5**
Setpoint (axis)	C1	X1	Y1
Actual value (1st encoder)	C1	X1	Y1

Example screenshots for the Y axis



4. Check

- **After power ON**, check the following machine data:

MD 30110, 30220, and MD 13050

Axis	**X**	**Y**	**Z**	**C**
MD 30110	2	3	3	1
MD 30130	1	1	0	1
MD 30220[0]/[1]	2	3	3	1
MD 30240	1	4	0	1

No. 3 duplicated? Z simulated, OK!

Index Idx	**0**	**1**	**2**	**3**
MD 13050[Idx]	4100	4140	4180	4220
MD 13060[Idx]	**136**	**136**	(116)	**136**

- If necessary, correct the **bold** values in MD 13060.
  If you do not do this, alarm 26015 will be generated.

Traverse axes

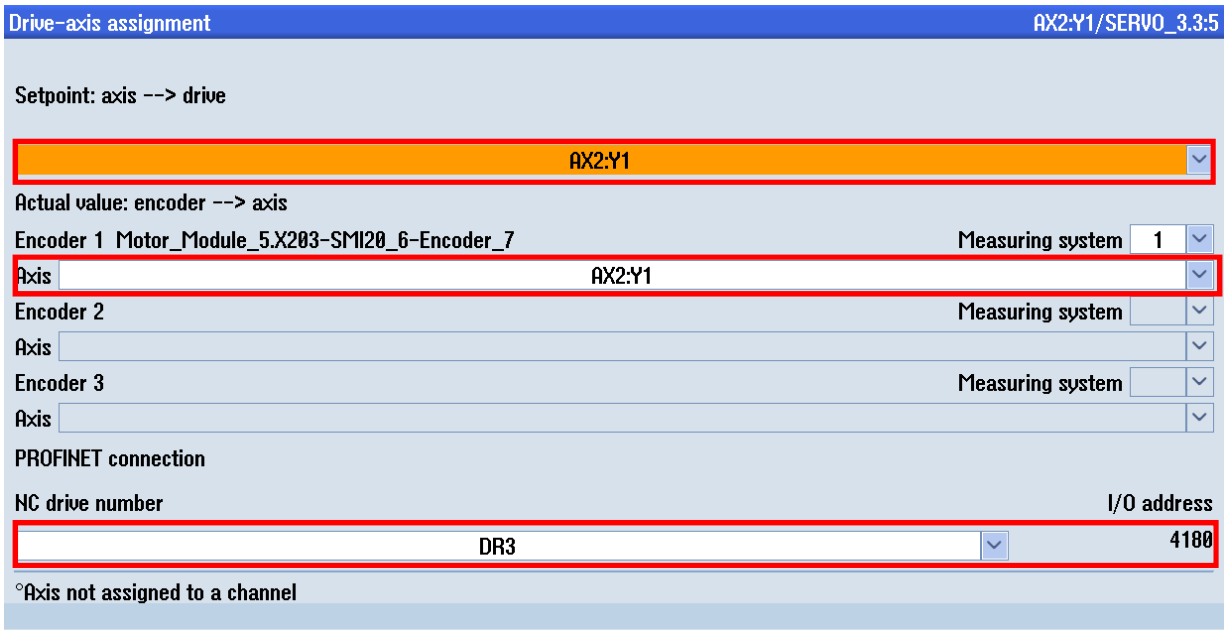
- Check that the right axes move in JOG mode!
  If enable signals are missing -> "Check for missing enables" exercise

5. Optional additional exercise: Select logical drive number

Now modify the logical drive numbers:

Drive:	**Servo_3.3:3**	**Servo_3.3:4**	**Servo_3.3:5**
Setpoint (axis)	C1	X1	Y1
Actual value (1st encoder)	C1	X1	Y1
PROFINET connection	**DR4**	**DR1**	**DR2**

Example screenshots for the Y axis **following confirmation:**



Please note: The **I/O addresses** assigned to the axes or drives **have not changed!**

6. Optional additional exercise: Check: MD 30110, 30220, and MD 13050

- **After power ON**, check the following machine data:

Axis	**X**	**Y**	**Z**	**C**
MD 30110	1	2	3	4
MD 30130	1	1	0	1
MD 30220[0]/[1]	1	2	4	4
MD 30240	1	4	0	1

Index Idx	**0**	**1**	**2**	**3**
MD 13050[Idx]	4140	4180	4220	4100

This means, for example, that: Y is using the 2nd entry [Idx=1] from list MD 13050 as an I/O address: -> **IO(Y) = 4180**

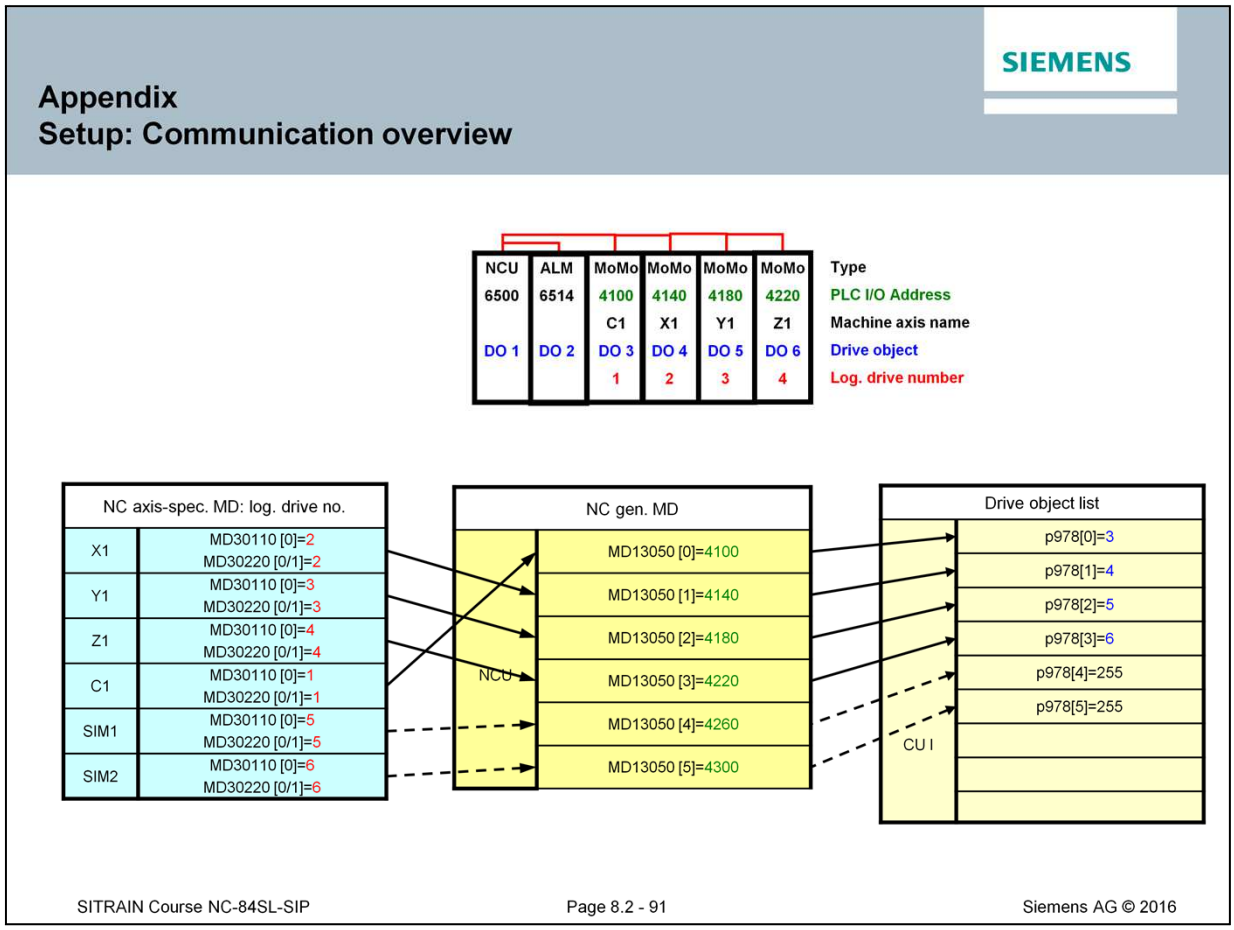
- Check that the right axes move in JOG mode!
  If enable signals are missing -> "Check for missing enables" exercise

Appendix



SITRAIN Course NC-84SL-SIP Page 8.2 - 90 Siemens AG © 2016

Note The following pages either contain additional information or are used as reference to complete a specific topic.



General MD

The following data is relevant for communication:

- **MD 13050[n=0..30] I/O addresses of drives**

Here, the index is the axis number. The MD already has a suitable preassignment.

- **MD 13060[n=0..30] Telegram type**

136 should be entered for every active axis here.

- **MD 13120[n=0..6] I/O addresses CUs**

- **CU: p978[n] List of drive objects** (DriveObjects, DO)

The index n is the PLC object number - 1, the value of the MD is the DO no.

special values: 255 Object does not exist

0 End of list

Active objects whose PB telegrams do not contain any data (e.g. 2 (ALM)) appear at the end of the list.

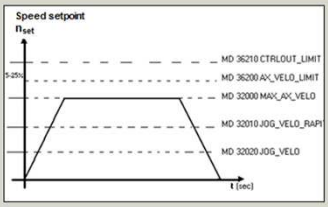
Example

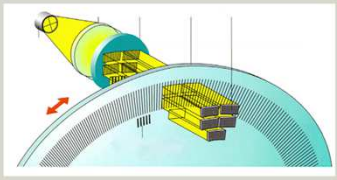
The axis configuration is as shown above, for example.
The first drive unit contains axes X1, Y1, Z1, and C1, where spindle C1 has been assembled directly adjacent to the infeed as the most powerful axis.
The PLC I/O addresses are as per the hardware configuration in the PLC.

Addressing

The logical drive numbers are used as pointers to the index of MD 13050 to define which drive-specific data set is output at which I/O address and thus at which module.

SIEMENS





Chapter 9.1

Startup axes: Monitoring functions, encoder

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Commissioning sequence 3
Measuring system settings 4
Velocities and monitoring 5
End position monitoring 6
Tolerance window monitoring 7
Encoder types 8
Referencing/adjustment 12
Measuring system monitoring 17
Exercise: 18
Exercise: 21
Additional exercise 27

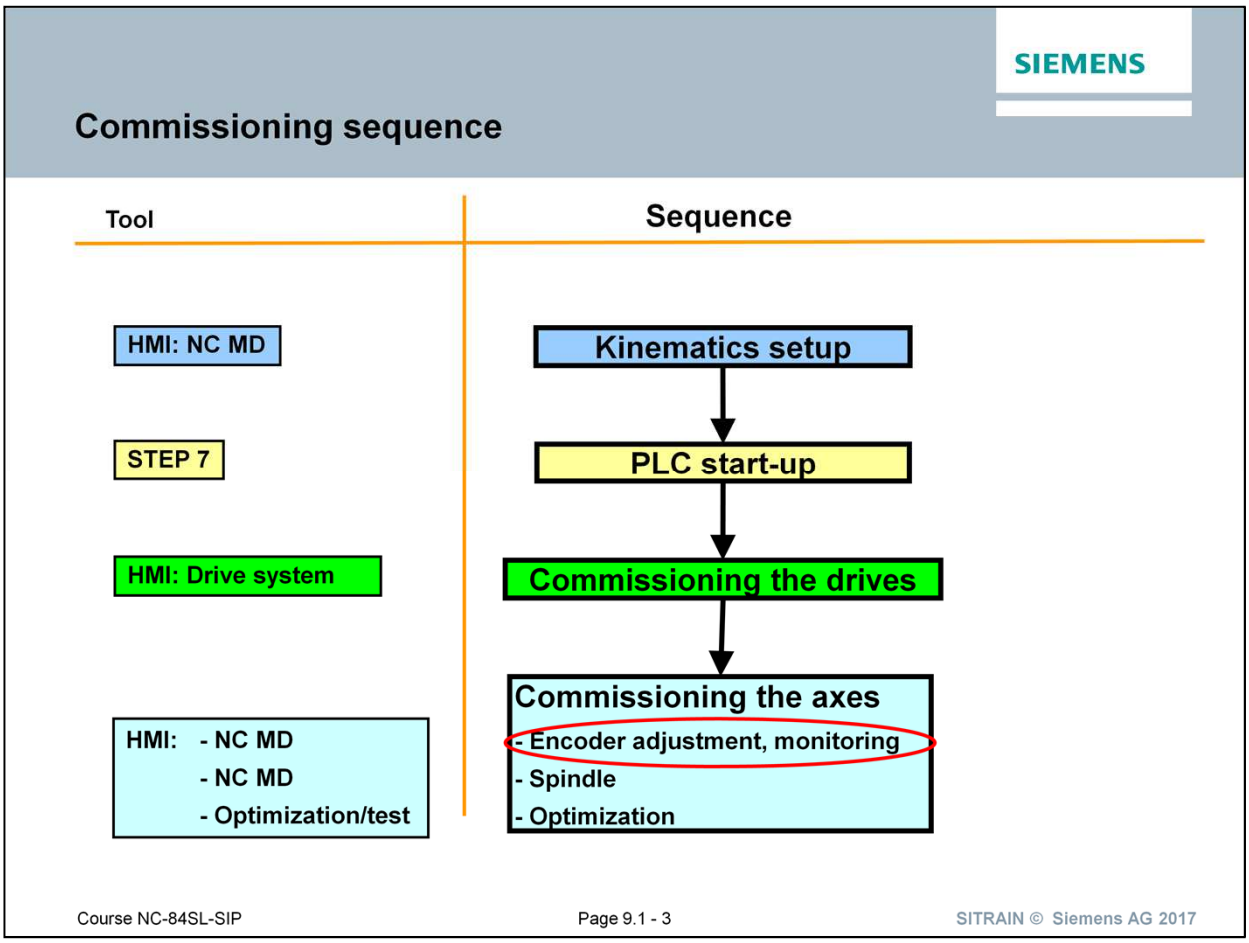
Learning objectives

- You will have an understanding of setpoint and actual value configuration
- You will be able to reference/adjust measuring systems
- You will be able to set up monitoring functions



SITRAIN © Siemens AG 2017

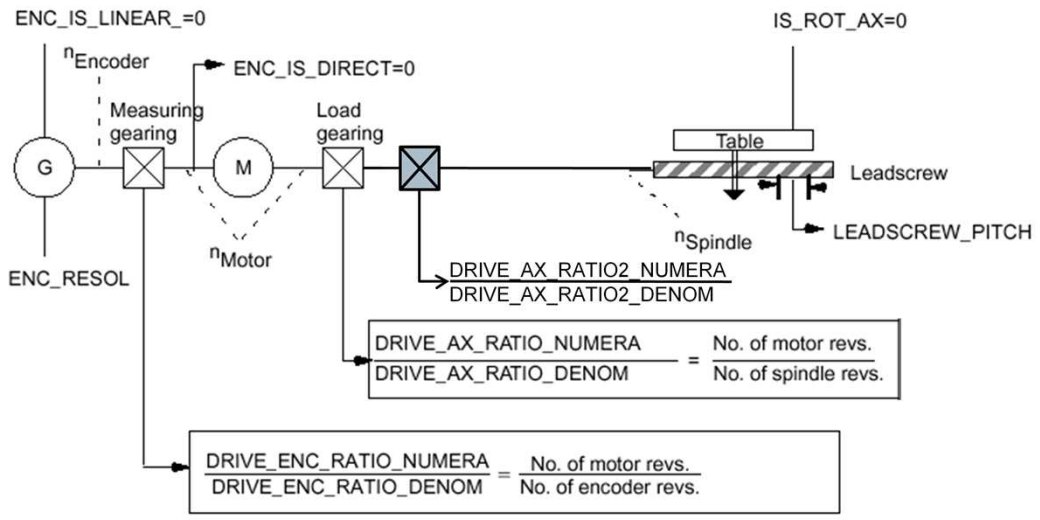
Notes



Commissioning part:

Axes particularly:
- Encoder adjustment
- Monitoring functions

Measuring system settings



Course NC-84SL-SIP Page 9.1 - 4 SITRAIN © Siemens AG 2017

Linear scale
MD 31000 ENC_IS_LINEAR [n]
1 Encoder for actual position sensing is linear (linear scale).
0 Encoder for actual position sensing is rotary.

Direct measuring system
MD 31040 ENC_IS_DIRECT [n]
1 Encoder for actual position sensing is directly on the machine.
0 Encoder for actual position sensing is directly on the motor.

Number of increments
MD 31020 ENC_RESOL [n]
Increments per encoder revolution of a rotary measuring system.
The motor measuring systems have 2,048 increments per revolution

Scale division
MD 31010: ENC_GRID_POINT_DIST
Grid spacing of linear measuring system in mm.

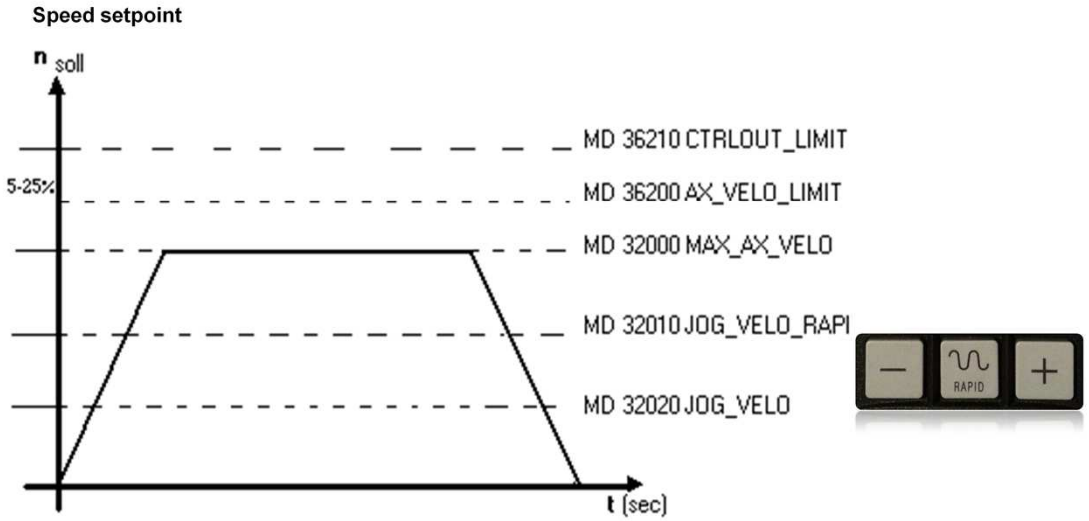
Leadscrew pitch
MD 31030 LEADSCREW_PITCH
Pitch of the leadscrew

Motor/load gearbox
MD 31060 DRIVE_AX_RATIO_NUMERA [n]
Load gearbox numerator
MD 31050 DRIVE_AX_RATIO_DENOM [n]
Load gearbox denominator

Intermediate gears
MD 31066 DRIVE_AX_RATIO2_NUMERA
MD 31064 DRIVE_AX_RATIO2_DENOM
Software version 6.4 and higher allows you to define intermediate gears; their position can be set via
MD 31044 ENC_IS_DIRECT2.

Motor/measuring system gearbox
MD 31080 DRIVE_ENC_RATIO_NUMERA [n]
Measuring gearbox numerator
MD 31070 DRIVE_ENC_RATIO_DENOM [n]
Measuring gearbox denominator

Velocities and monitoring



Course NC-84SL-SIP Page 9.1 - 5 SITRAIN © Siemens AG 2017

Velocities

The following velocities must be defined:

MD 32000: MAX_AX_VELO (maximum axis velocity G0)

MD 32010: JOG_VELO_RAPID (rapid traverse in JOG mode)

MD 32020: JOG_VELO (JOG axis velocity JOG mode)

Speed setpoint monitoring functions

MD 36210 CTRLOUT_LIMIT (maximum speed setpoint in percent)

When the limit is exceeded,
alarm "25060 Axis %1 Speed setpoint limitation" is triggered and the axis is stopped.

MD 36220 CTRLOUT_LIMIT_TIME
(delay time speed setpoint monitoring.)
The MD defines for how long the speed setpoint may exceed the value of MD 36210 before monitoring is triggered. When the limit is reached, contour errors occur.

Actual velocity monitoring functions

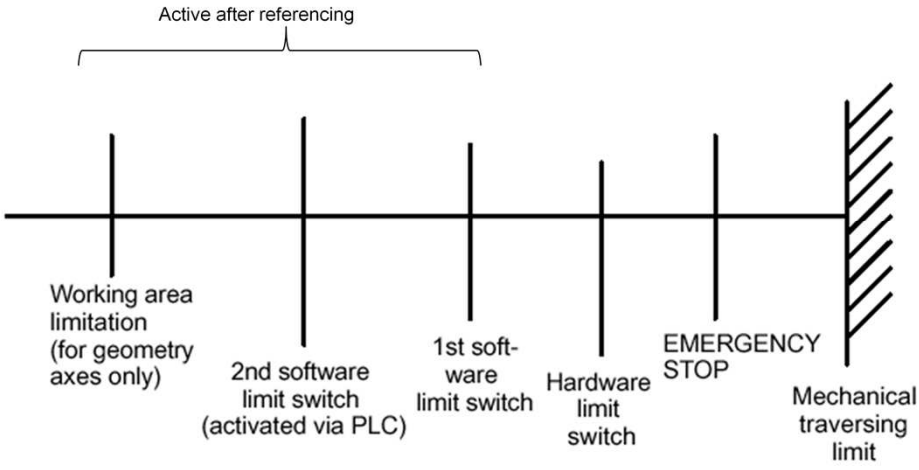
MD 36200 AX_VELO_LIMIT (threshold value for actual velocity monitoring)
When the threshold value is exceeded, the **alarm** "25030 Actual velocity alarm limit" is triggered and the axes are stopped.

Contour monitoring

MD 36400: CONTOUR_TOL (tolerance band for contour monitoring)

Monitoring is performed by comparing the difference between the NC position setpoint and the measured actual position value with the following error calculated in advance. The contour monitoring is always active in position-controlled monitoring.
If the tolerance band is exceeded,
alarm "25050 Axis [Name] Contour monitoring" is generated and the axes are decelerated via a preset braking ramp (MD 36610).

End position monitoring



Course NC-84SL-SIP Page 9.1 - 6 SITRAIN © Siemens AG 2017

Hardware limit switch

Monitoring can be implemented for every axis via the PLC interface (IS "Hardware limit switch minus/plus" DB31.DBX12.0 / 12.1). When the limit switch is reached, the axes are stopped. The braking response is set in MD 36600.

MD 36600: BRAKE_MODE_CHOICE (braking response to hardware limit switches)

1 Rapid deceleration with setpoint "0"

0 Deceleration characteristic is adhered to

Alarm "21614 Channel [Name1] Axis [Name2] Hardware limit switch [+/-]".

Software limit switch

MD 36100: POS_LIMIT_MINUS (1st software limit switch minus)
MD 36110: POS_LIMIT_PLUS (1st software limit switch plus)
MD 36120: POS_LIMIT_MINUS2 (2nd software limit switch minus)
MD 36130: POS_LIMIT_PLUS2 (2nd software limit switch plus)

The effective software limit switch is selected via PLC
(IS "2nd software limit switch minus/plus" DB31.DBX12.2 / DBX12.3).
The monitoring is only effective after reference point approach.
The monitoring function is no longer effective after a PRESET.

Alarms: 10620 / 10621 / 10720 - 10722

Working area limitation with setting data

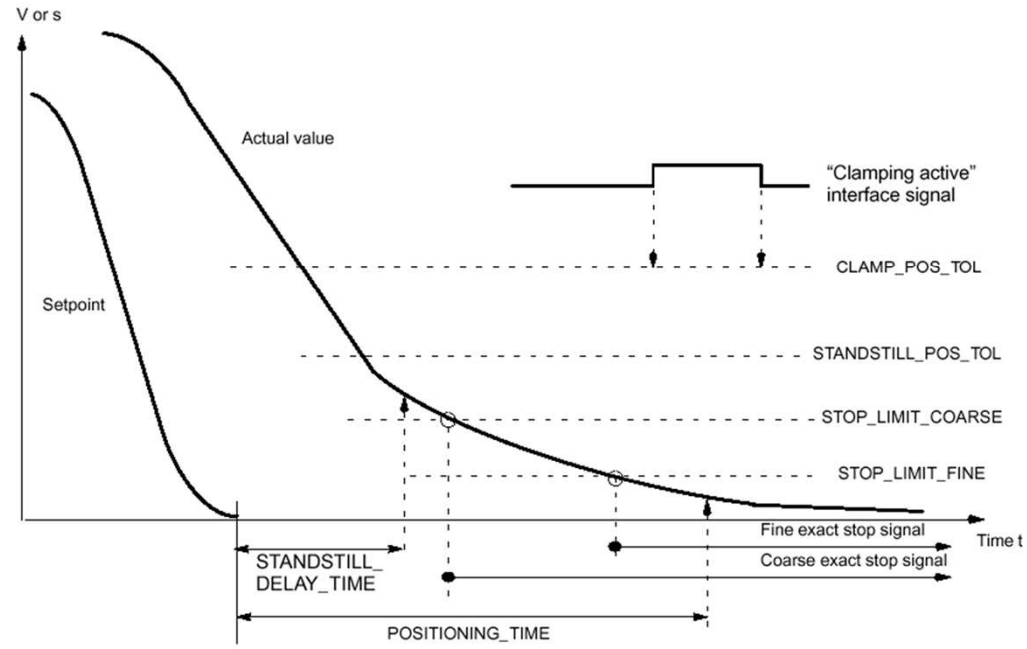
Working area limitations for geometry axes can be defined and activated via setting data ("Parameters" OA, "Setting data" softkey, "Working area limitation" softkey). Monitoring is active after reference point approach.

Working area limitation via part program

Working area limitations for geometry axes can be set by the part program with G25/G26 and activated/deactivated with the commands WALIMON and WALIMOF.

Alarms: 10630 / 10631 / 10730

Tolerance window monitoring



 SITRAIN © Siemens AG 2017

Monitoring functions for positioning

The monitoring functions for positioning monitor whether the axis reaches the position window (exact stop) and if an axis for which there is no drive command exits a specific tolerance window (standstill monitoring, clamping tolerance).

Exact stop coarse

MD 36000 STOP_LIMIT_COARSE
IS "Position reached with exact stop coarse" (DB31, ... DBX60.6)

Exact stop fine

MD 36010 STOP_LIMIT_FINE
IS "Position reached with exact stop fine" (DB31, ... DBX60.7)

Delay time exact stop fine

MD 36020 POSITIONING_TIME
Delay time after which the actual value must have reached the tolerance window "Exact stop fine" after the setpoint position was reached.
If the exact stop fine window is not reached within the specified time,
Alarm "25080 Axis [Name] Positioning monitoring" is output.

Standstill tolerance

MD 36030 STANDSTILL_POS_TOL
Position tolerance that must not be exceeded by an axis at standstill.
If the tolerance window is exceeded,
Alarm "25040 axis [name] standstill monitoring" is output.

Delay time zero-speed monitoring

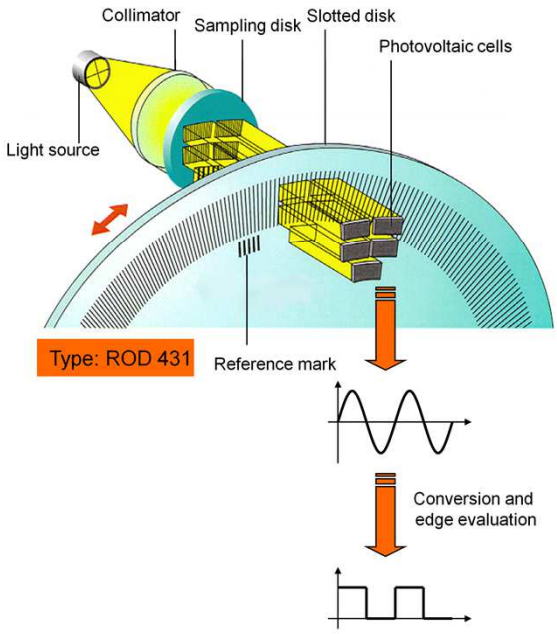
MD 36040 STANDSTILL_DELAY_TIME
Delay time after which the actual value must have reached the tolerance window "standstill tolerance" when the setpoint position is reached.
If the position tolerance is not reached,
alarm "25040 Axis [Name] standstill monitoring" is output.

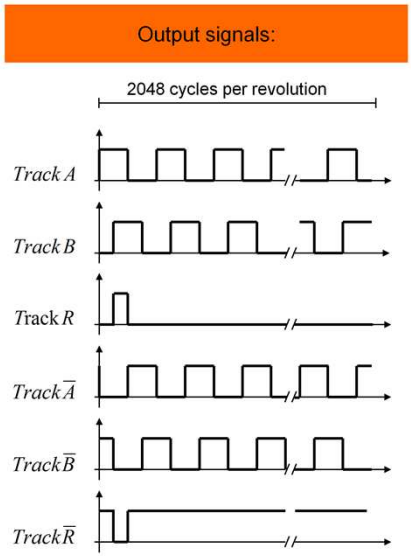
Clamping tolerance

MD 36050 CLAMP_POS_TOL (clamping tolerance)
Position tolerance that is monitored when the PLC interface signal "Clamping active" (DB31, ... DBX2.3) is pending. If the tolerance is exceeded, **alarm** "26000 … clamping monitoring" is output.

Encoder types
Incremental encoder with square wave signals

SIEMENS

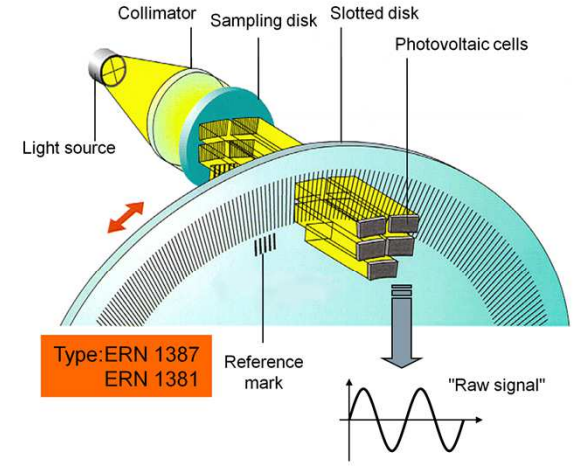


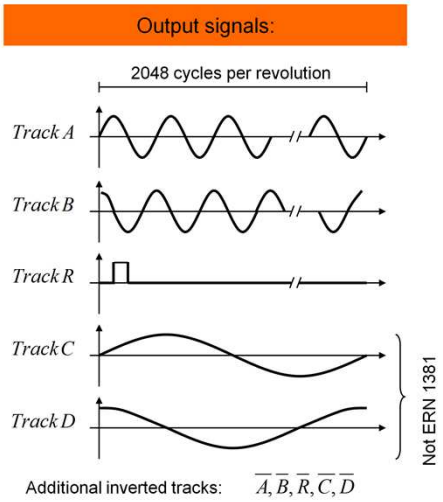


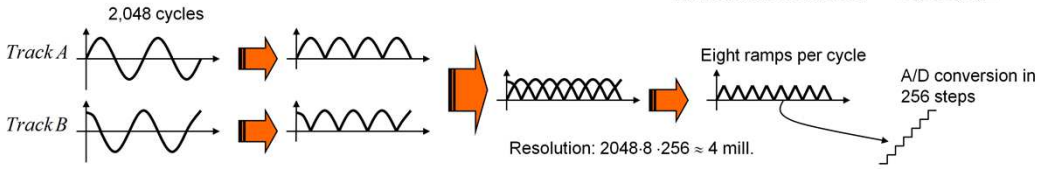
SITRAIN © Siemens AG 2017

Notes

Encoder types
Optical sin/cos encoder





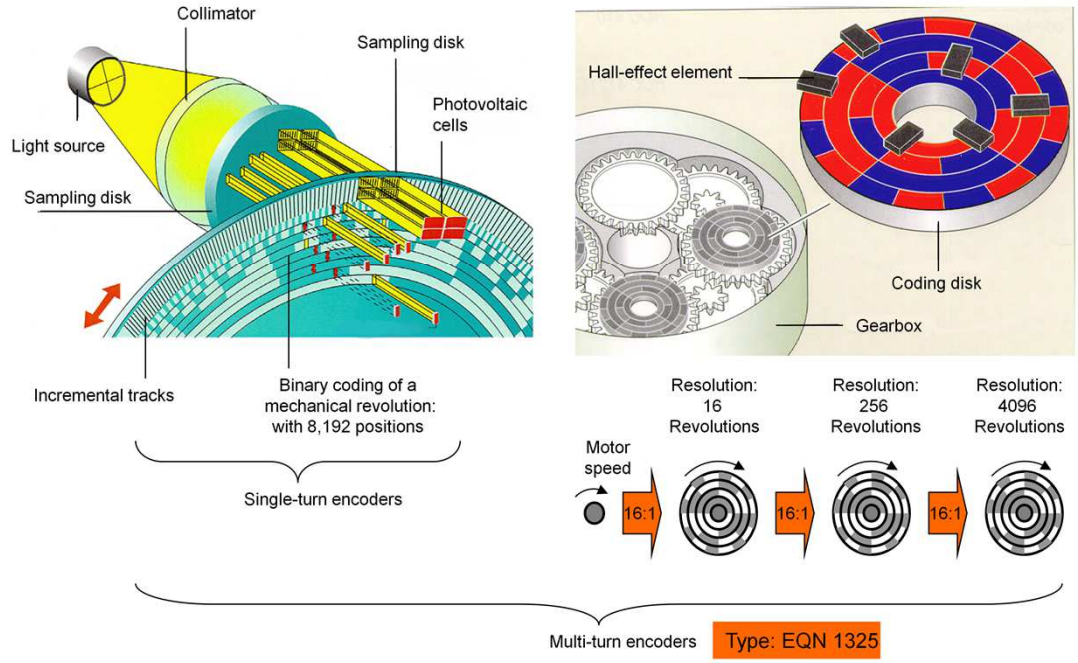


Course NC-84SL-SIP Page 9.1 - 9 SITRAIN © Siemens AG 2017

Notes

Encoder types
Absolute encoder

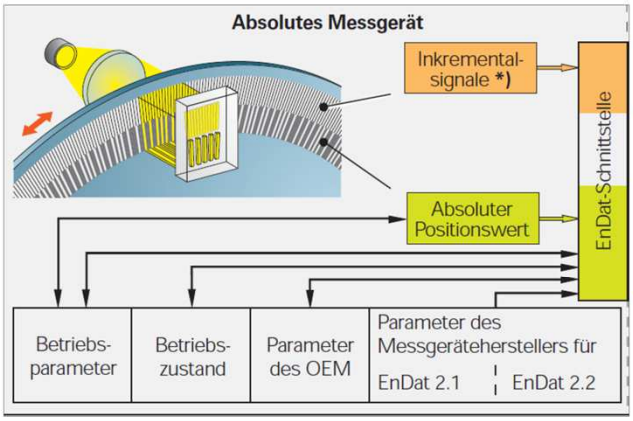
SIEMENS

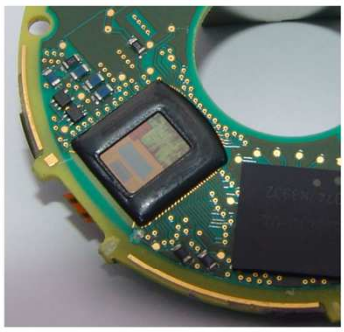


SITRAIN © Siemens AG 2017

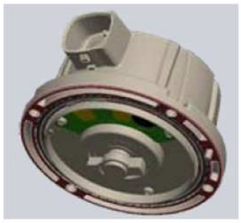
Notes

Encoder types
New absolute encoders in 1FT7 motors











Course NC-84SL-SIP Page 9.1 - 11 SITRAIN © Siemens AG 2017

Note Multi-turn encoders still have an encoder gearbox and therefore perform the revolution evaluation. Only the Hall-effect sensors are now in one chip and are no longer discretely mounted.

Absolute encoder:

Encoder AS20DQI: Absolute encoder, 20-bit, single-turn
(resolution 1048576, in the encoder 512 S/R)

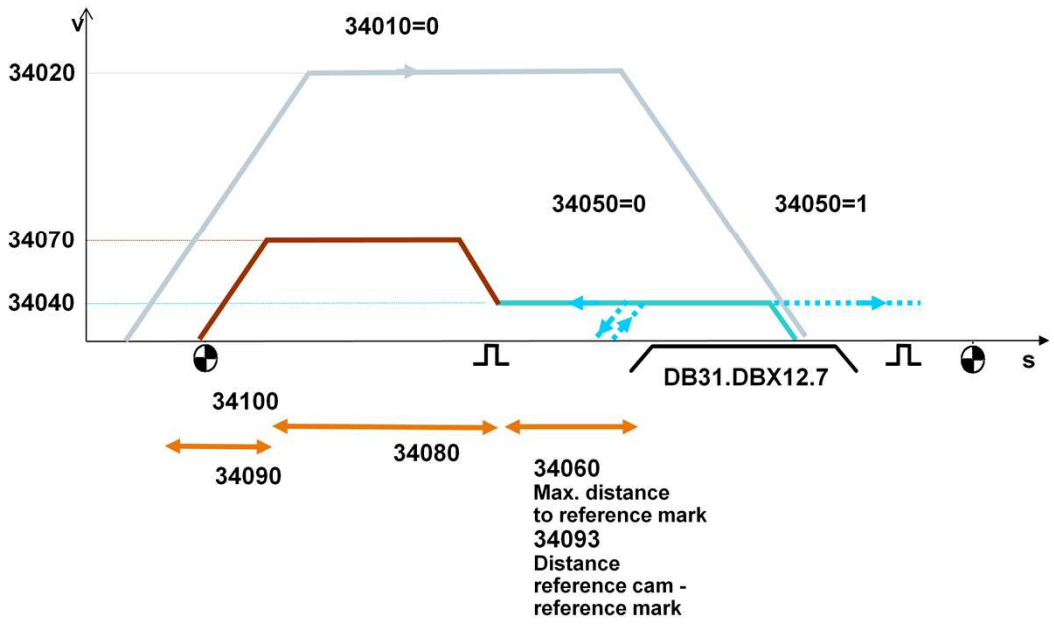
Encoder AM20DQI: Absolute encoder, 20-bit single-turn
(resolution 1048576, in the encoder 512 S/R)
+ 12 bit multi-turn with 4096 revolutions.

Encoder AS24DQI: Absolute encoder, 24-bit, single-turn
(resolution 16777220, in the encoder 2048 S/R)

Encoder AM24DQI: Absolute encoder, 24-bit single-turn
(resolution 16777220, in the encoder 2048 S/R)
+ 12 bit multi-turn with 4096 revolutions.

Referencing/adjustment
Reference point approach incremental measuring system

SIEMENS



Course NC-84SL-SIP Page 9.1 - 12 SITRAIN © Siemens AG 2017

Start of phase 1

Phase 1 is started with the plus or minus traversing keys.
MD 11300: JOG_INC_MODE_LEVELTRIGGRD specifies whether the traversing key must remain pressed for phases 1 to 3 of referencing.

If the machine axis is positioned in front of the reference cam, it accelerates to the velocity specified in MD 34020: REFP_VELO_SEARCH_CAM (reference point approach speed). The approach direction is defined in MD 34010: REFP_CAM_DIR_IS_MINUS (reference point approach in negative direction). With the IS "Delayed reference point approach" (DB31...61, DBX12.7), the control recognizes that the reference cam has been reached and decelerates the axis to standstill.

Start of phase 2

• Synchronization with falling reference cam edge **MD 34050= 0**

The machine axis accelerates to the velocity specified in MD 34040: REFP_VELO_SEARCH_MARKER in the opposite direction to phase 1. If the reference cam is exited, the control synchronizes with the first zero mark.

• Synchronization with rising reference cam edge **MD 34050= 1**

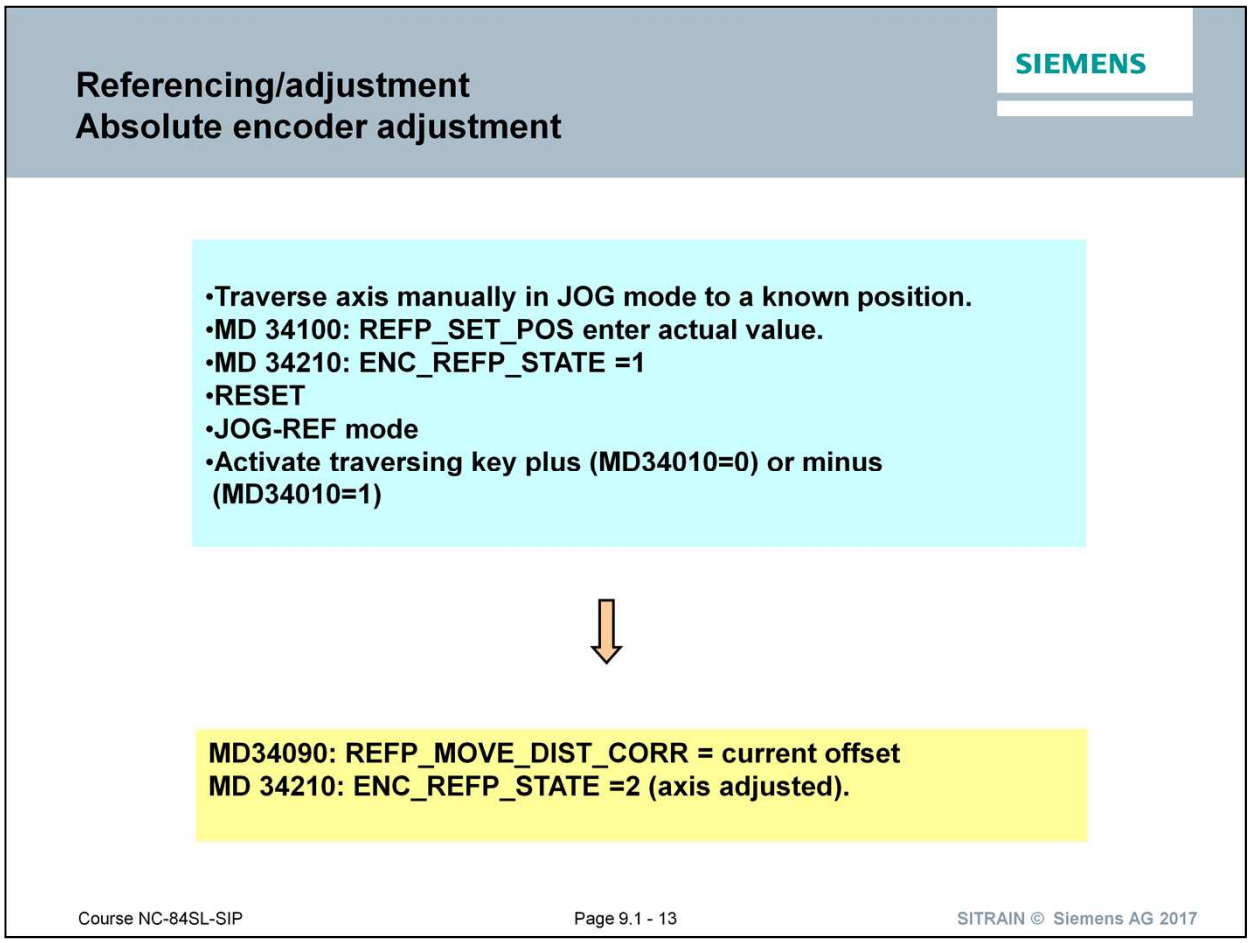
The machine axis accelerates to the velocity specified in MD 34040: REFP_VELO_SEARCH_MARKER in the opposite direction to phase 1. When leaving the reference cam, the machine axis decelerates to standstill and travels beyond the reference cam. The control system will synchronize itself with the first zero mark.

Start of phase 3

Phase 3 is started automatically. The machine axis accelerates to the velocity specified in MD 34070: REFP_VELO_POS and travels a distance resulting from addition of the paths from MD 34080: REFP_MOVE_DIST and MD 34090: REFP_MOVE_DIST_CORR.

Adjustment

1. Reference point approach
2. Moving to a mechanical reference point (measure!)
3. MD34090 (new) = MD34090 (old) + actual position - desired position
4. Check by repeating 1 and 2



Absolute encoder adjustment

1. Traverse axis manually in JOG mode to a known position.
2. Enter the actual value corresponding to the approached position in MD 34100: REFP_SET_POS. This value may be a specified design value (e.g. fixed stop) or can now be determined with a measuring instrument.
3. MD 34210: Set ENC_REFP_STATE to "1". This enables the "Adjustment" function.
4. The modified machine data becomes effective after a RESET.
5. Switch to JOG-REF mode.
6. By activating the traverse key plus (MD34010=0) or minus (MD34010=1), the current offset is entered in MD34090: REFP_MOVE_DIST_CORR and MD 34210: ENC_REFP_STATE changes to "2" (axis adjusted).
7. Exit JOG-REF operating mode. The calibration operation for this axis is now complete.

Note

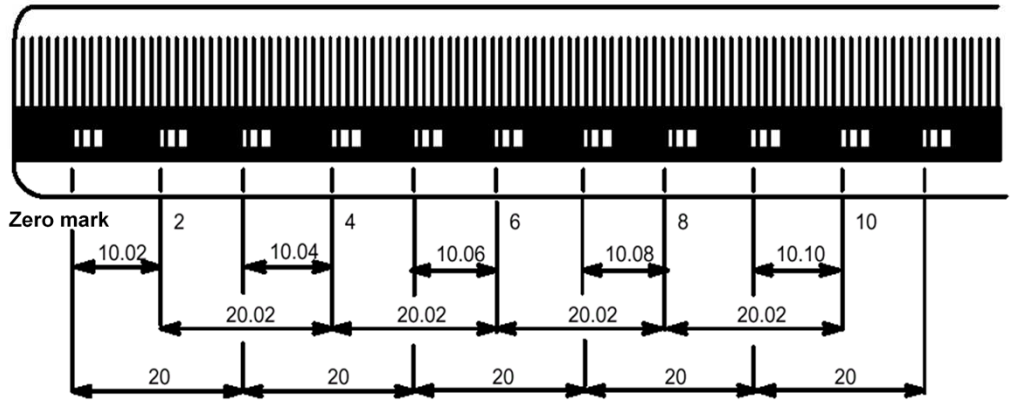
MD 34220 ENC_ABS_TURNS_MODULO[n]= 4096

The MD is only relevant for rotary encoders (on linear and rotary axes).
The default value "4096" is a setting for the most commonly used encoder types (e.g. motor measuring system).
When you use an encoder with smaller multi-turn information or when you use single-turn encoders, the value must be changed accordingly.

Referencing/adjustment
Adjustment of distance-coded measuring system

SIEMENS

Absolute offset = machine actual position + / - actual position of the length measuring system



MD 34200: ENC_REFP_MODE[n] (reference point mode = 3)
MD 34300: ENC_REFP_MARKER_DIST (reference mark distance) = 20
MD 34310: ENC_MARKER_INC (interval between two reference marks = 0.02)

SITRAIN © Siemens AG 2017

Reference cam A reference cam only has to be mounted in front of the end of the traversing area for channel-specific and part program referencing (G74).

Referencing The time sequence of referencing is divided into 2 phases:

Phase 1 Overtravel of two reference marks with synchronization.

Phase 2 Travel to a fixed end point
MD 34330: STOP_AT_ABS_MARKER (linear measurement system without end point)
MD 34100: REFP_SET_POS (end position)

Determining the absolute offset

MD 34090: REFP_MOVE_DIST_CORR reference point/absolute offset

MD 34320: ENC_INVERS length measurement system in opposite direction to the machine system

Method 1

The absolute offset can be determined by entering the absolute offset between the machine zero point and the position of the first reference mark on the linear measurement system directly in MD 34090 (often difficult to measure).

Method 2

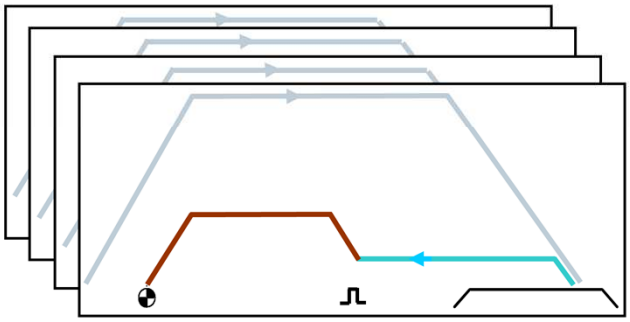
The absolute offset is calculated.
Requirement:
MD 34320 (measuring system in opposite direction) is correctly set.
To test the setting, approach the reference point once and note the position. Then traverse the axis in JOG mode in plus direction, e.g. 50 mm. After the axis has been referenced once again, the actual value must be greater than the noted value.
Now traverse the axis to the mechanical reference point and calculate MD 34090 by the following formula:
MD 34090(new)=MD 34090 (old) + desired position - displayed actual position
After MD 34090 has been changed, referencing must be performed again!

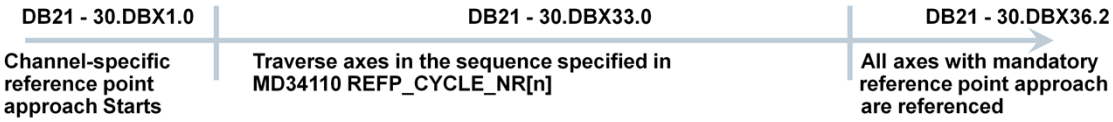
Referencing/adjustment
Channelspecific referencing

SIEMENS

MD 34110: REFP_CYCLE_NR channel-specific referencing

1 to n	Determines the axis sequence for channel-specific referencing.
0	The machine axis is not started via channel-specific referencing
-1	NC start is possible without referencing of this axis.





Course NC-84SL-SIP Page 9.1 - 15 SITRAIN © Siemens AG 2017

Axis-specific referencing

Axis-specific referencing is started separately for each machine axis with the "Traversing key plus/minus" IS (DB31, ...DBX4.7 / 4.6). If the machine axes are to be referenced in a particular sequence, the following options are available:

- The operator must adhere to the sequence during starting.
- The PLC user program checks the sequence on starting or defines the sequence itself.

Channelspecific referencing

- Configuration Via the axis-specific **MD 34110:** REFP_CYCLE_NR (axis sequence for channel-specific referencing) it can be determined in which sequence the machine axes are referenced.

- Interface Specific referencing is started by a pulse in bit 1.0 in the channel axis interface (DB21…30).

After the start of channel-specific referencing, bit 33.0 of the channel axis interface (DB21…30) is set to true until all axes that have to be referenced have been referenced.

At the end of channel-specific referencing, if all axes that have to be referenced have been referenced, bit 33.0 of the channel axis interface (DB21…30) is set to zero and bit 36.2 of the channel axis interface (DB21…30) is set to true.

Note Axis-specific referencing and channel-specific referencing are not mutually exclusive.

Referencing/adjustment
Reference point approach - general

SIEMENS

MD 34200: ENC_REFP_MODE[n] Reference point mode
0 Absolute encoder
1 Incremental measuring system with zero pulse on encoder track.
3 Distance-coded linear measurement system
5 BERO replaces reference cam

MD 34000: REFP_CAM_IS_ACTIVE
1 Reference cam is available
0 No reference cam (referencing with zero mark)

MD 20700: REFP_NC_START_LOCK
0 NC start without reference point approach possible

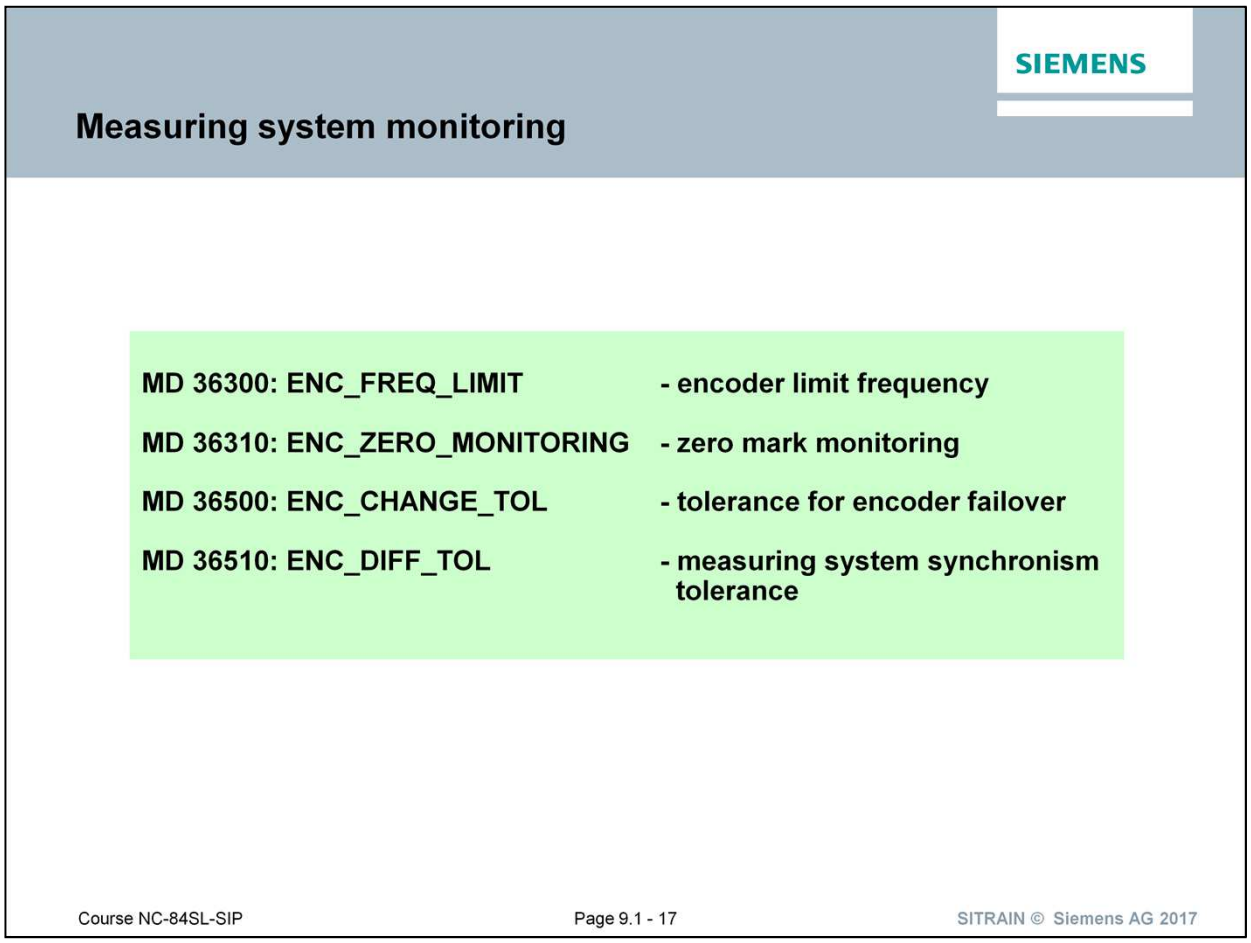
MD 11300: JOG_INC_MODE_LEVELTRIGGRD
1 JOG mode for JOG-INC and reference point approach
0 Continuous operation for JOG-INC and reference point approach

SITRAIN © Siemens AG 2017

MD 34200 Is usually automatically correctly set by the system as part of axis assignment.

MD 34000 The setting "0" is used, for example, if, for rotary axes, the motor is mounted directly on the axis, that is, a certain position of the encoder also corresponds to a certain position of the axis.

MD 20700 For machines usually always "1", in our training racks for simplification "0".



Encoder frequency monitoring

MD 36300: ENC_FREQ_LIMIT (encoder limit frequency),
The frequency entered in MD36300 is monitored.
If it is exceeded,
alarm "21610 Channel [Name] Axis [Name] Encoder [No.] frequency exceeded" is output and the axes are stopped.

Zero mark monitoring

MD 36310: ENC_ZERO_MONITORING > 0 activates zero mark monitoring.
The value indicates the number of impulses that are allowed to be lost. An error results in display of
alarm "25020 Axis [Name] Zero mark monitoring" and the axis is stopped with the ramp down (MD 36610).
MD 36310 = 100 Hardware monitoring of encoder switched off.

Tolerance in encoder failover

During failover, the position actual value difference is monitored.
If this difference is greater than the value entered in MD 36500: ENC_CHANGE_TOL,
alarm "25100 Axis %1 Measuring system switchover not possible" is output and the switchover is inhibited.

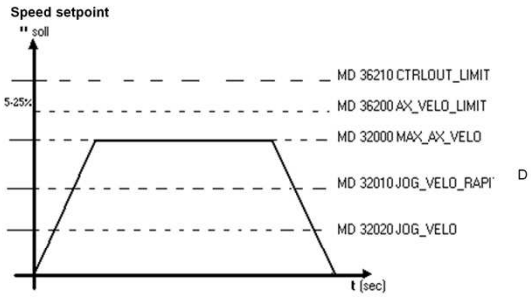
Measuring system synchronism tolerance

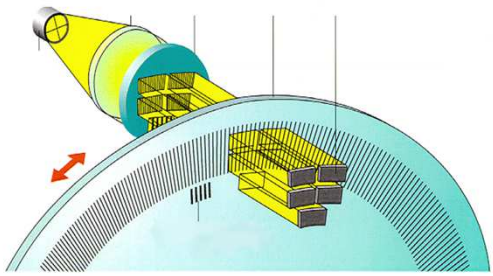
Permissible deviation between the actual values of the two measuring systems. This difference must not be exceeded during cyclic comparison of the two measuring systems used; if it is, a fault code is generated.

Exercise: Measuring system

SIEMENS







Course NC-84SL-SIP Page 9.1 - 18 SITRAIN © Siemens AG 2017

Notes

Exercise: Check the measuring system settings

Check the machine data for the measuring systems:

	Machine axis name	X	Y	Z	C
Linear scale	MD 31000 1 Linear scale 0 Rotary measuring system	.	.	.	.
Direct measuring system	MD 31040 1 Encoder on machine 0 Encoder on motor	.	.	.	.
No. of increments	MD 31020 Increments per encoder revolution	.	.	.	.
Grid spacing	MD 31010 Grid spacing of a linear measuring system				
Leadscrew pitch	MD 31030 Pitch of the leadscrew.	.	.	.	.
Gearbox **Motor/load**	MD 31060 Numerator of load gearbox	.	.	.	.
	MD 31050 Denominator of load gearbox	.	.	.	.
Gearbox	MD 31080	.	.	.	.
Motor/measuring system	Numerator of the measuring gearbox				
	MD 31070 Denominator of the measuring gearbox	.	.	.	.

Index[n] The measuring system machine data has an index [0] or [1].
[0] Values for the 1st measuring system
[1] Values for the 2nd measuring system

Exercise: Measuring system adjustment

Incremental measuring system

Calibrate the measuring system of the X axis via
MD 34090: reference point offset in such a way that the zero position of the rotating masses points upward.

Adjustment is performed in the sequence referred to earlier:

1. Approach reference point
2. Align the zero position of the centrifugal mass upward
   in JOG or JOG-INC
3. MD34090 (new) = MD34090 (old) + actual position (= display Machine OA)
   - desired position (=0 mm)
4. Check by repeating 1 and 2

Absolute measuring system

Adjust the measuring system of the Y axis as described below
so that the zero position of the centrifugal mass is pointing upward.

1. Traverse axis manually in JOG mode to a known position.
2. Enter the actual value corresponding to the approached position in MD 34100:REFP_SET_POS. This value may be a specified design value (e.g. fixed stop) or can now be determined with a measuring instrument.
3. Set MD 34210: ENC_REFP_STATE to "1". This enables the "Adjustment" function.
4. The modified machine data becomes effective after a RESET.
5. Switch to JOG-REF mode.
6. By activating the traverse key plus (MD34010=0) or minus (MD34010=1), the current offset is entered in MD34090: REFP_MOVE_DIST_CORR and MD 34210: ENC_REFP_STATE changes to "2" (axis adjusted).
7. Exit JOG-REF operating mode. Adjustment of this axis is now complete.

Exercise: Creating a user view

Task

Create a user view with the machine data for adjustment offset when the encoder is changed at the X axis and spindle.

Steps

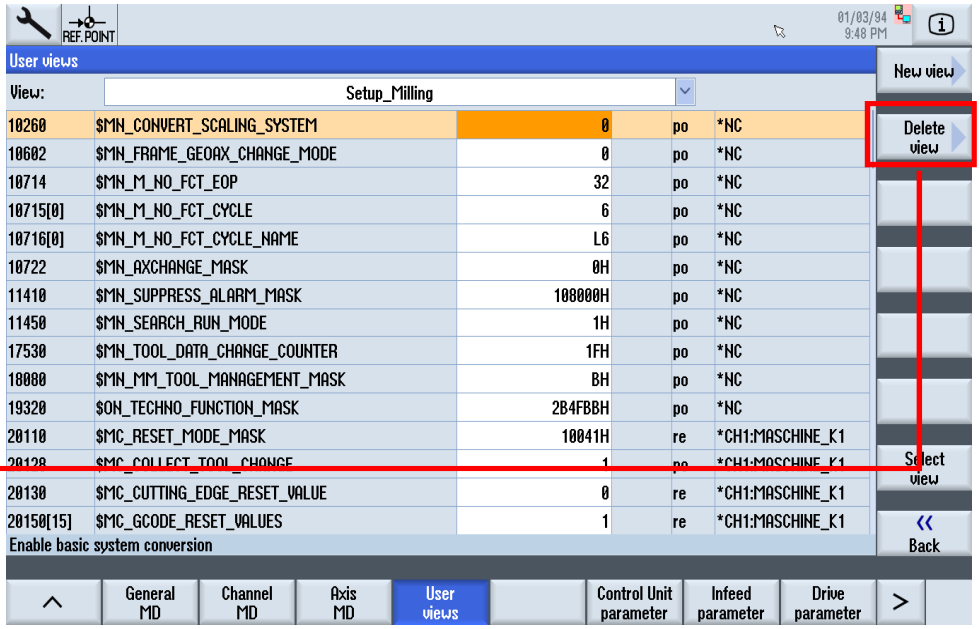
Select the user views using:

OA: Setup -> softkey: Mach. data

-> Softkey: User views -> softkey: New view

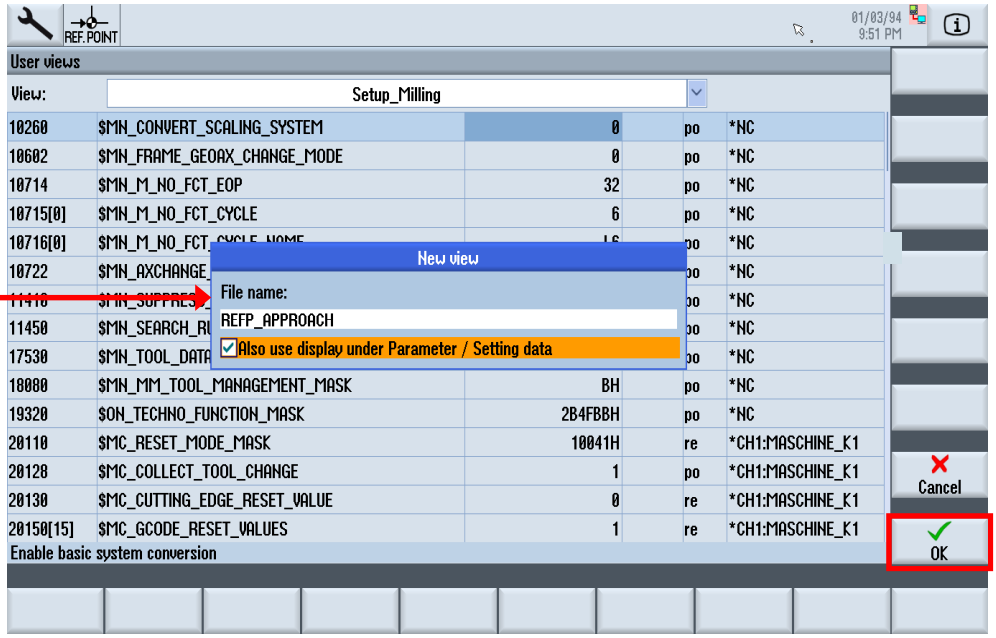
After you have pressed

--> softkey: "User views" the following menu is displayed



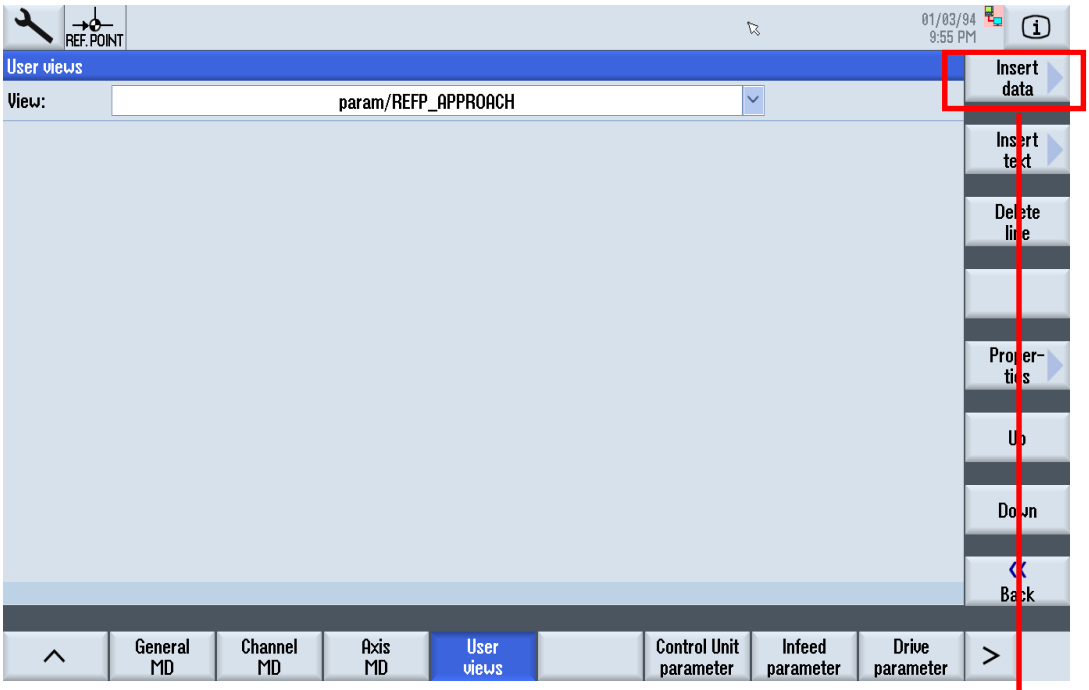
Here, press the

softkey: "New view" and assign an appropriate file name:

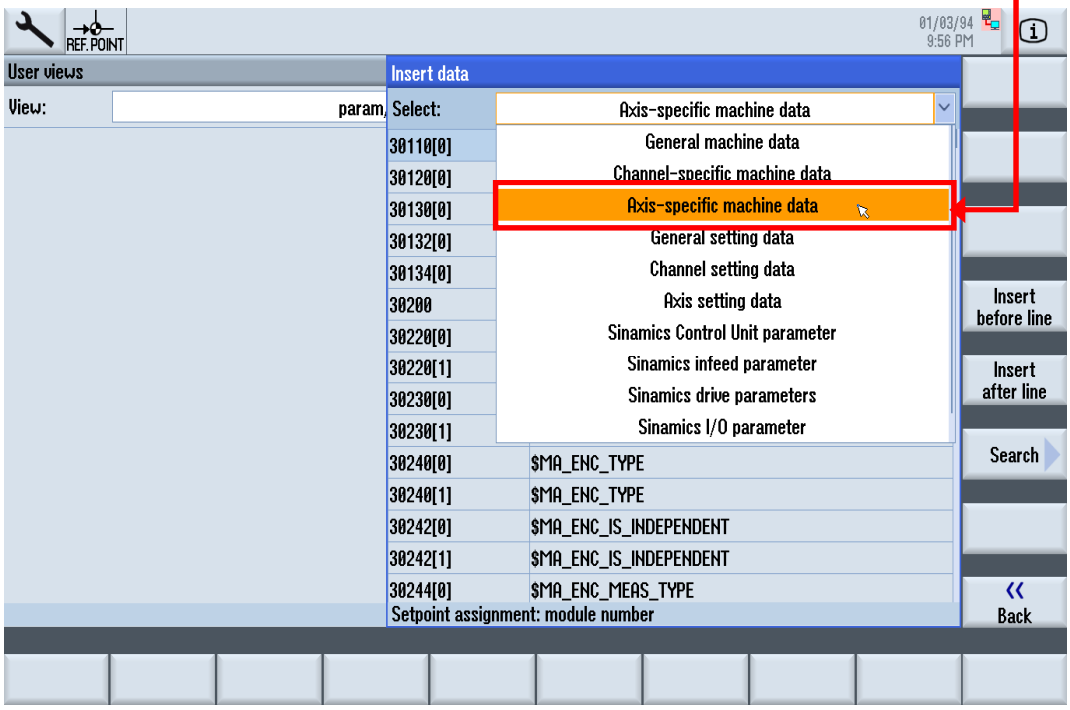


Confirm the screen form with **softkey: "OK"**

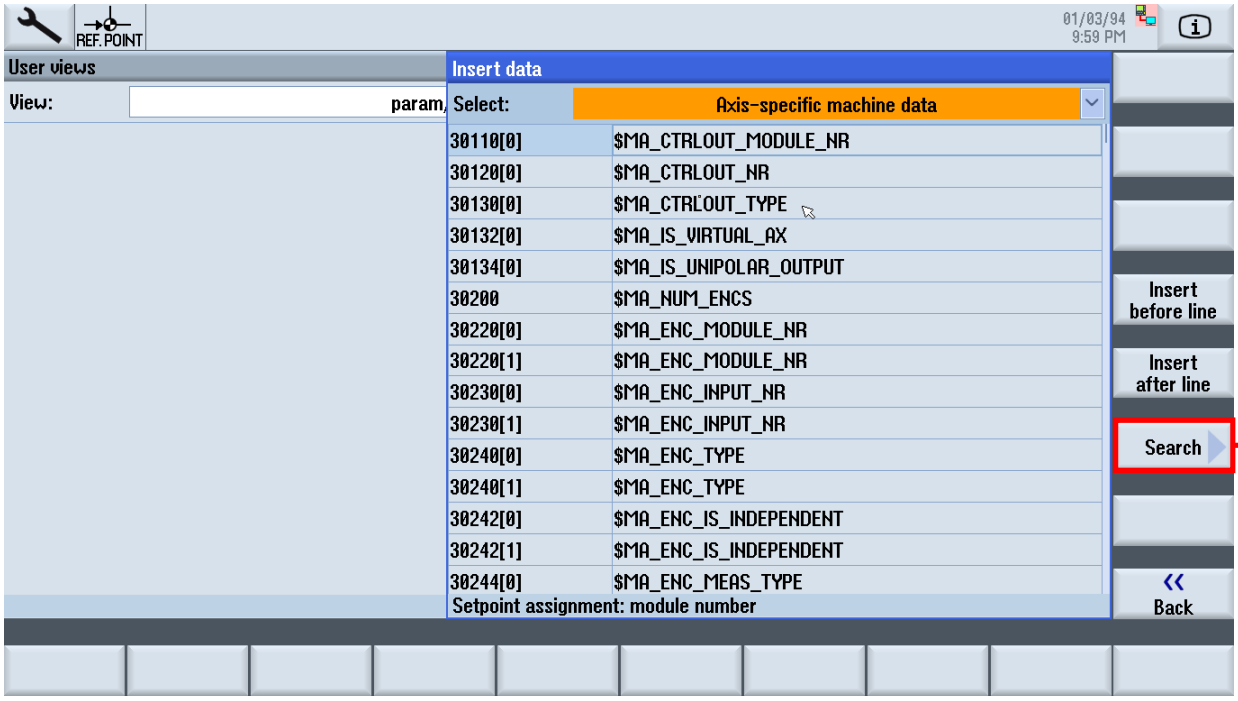
With **softkey: "Insert data",** you can select machine data and setting data:



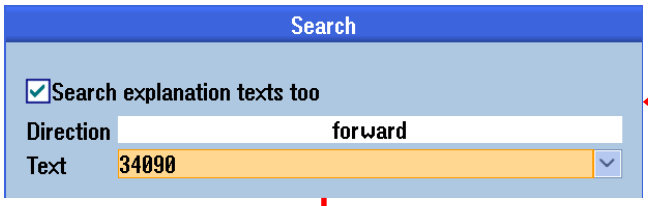
To start, first select the area **"Axis-specific machine data"**:

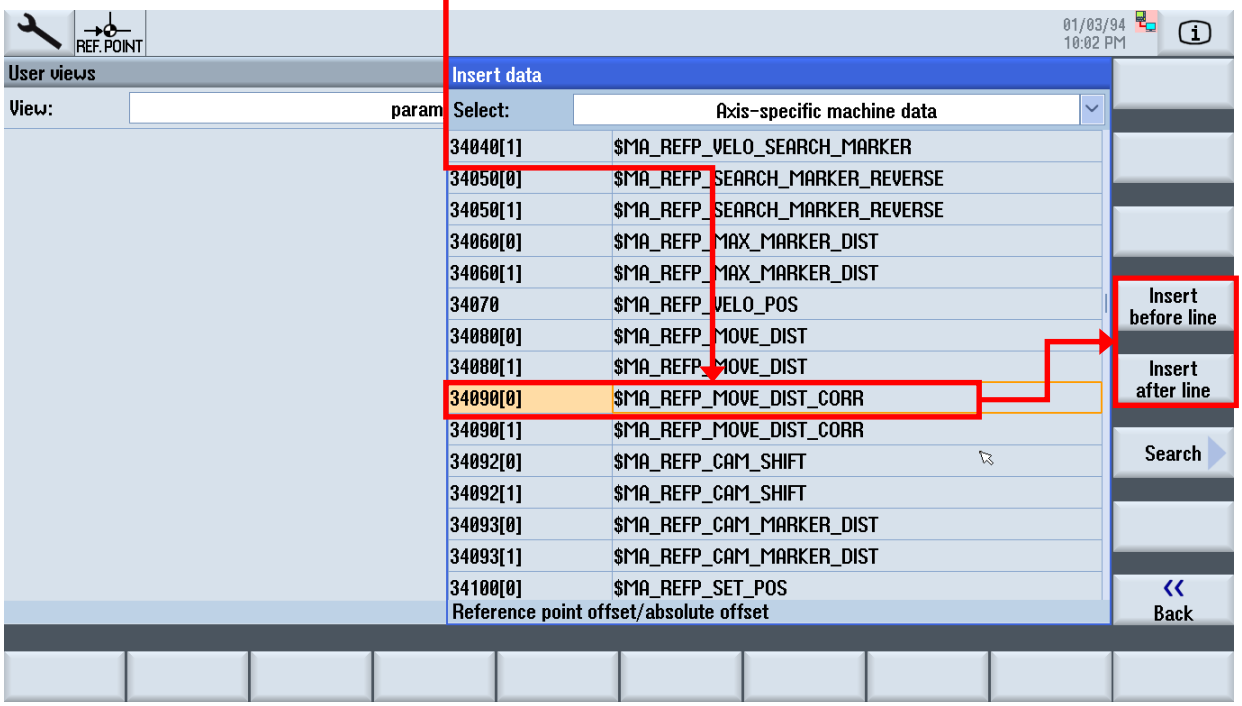


In the user views, you can mix the machine data and setting data of the various areas.



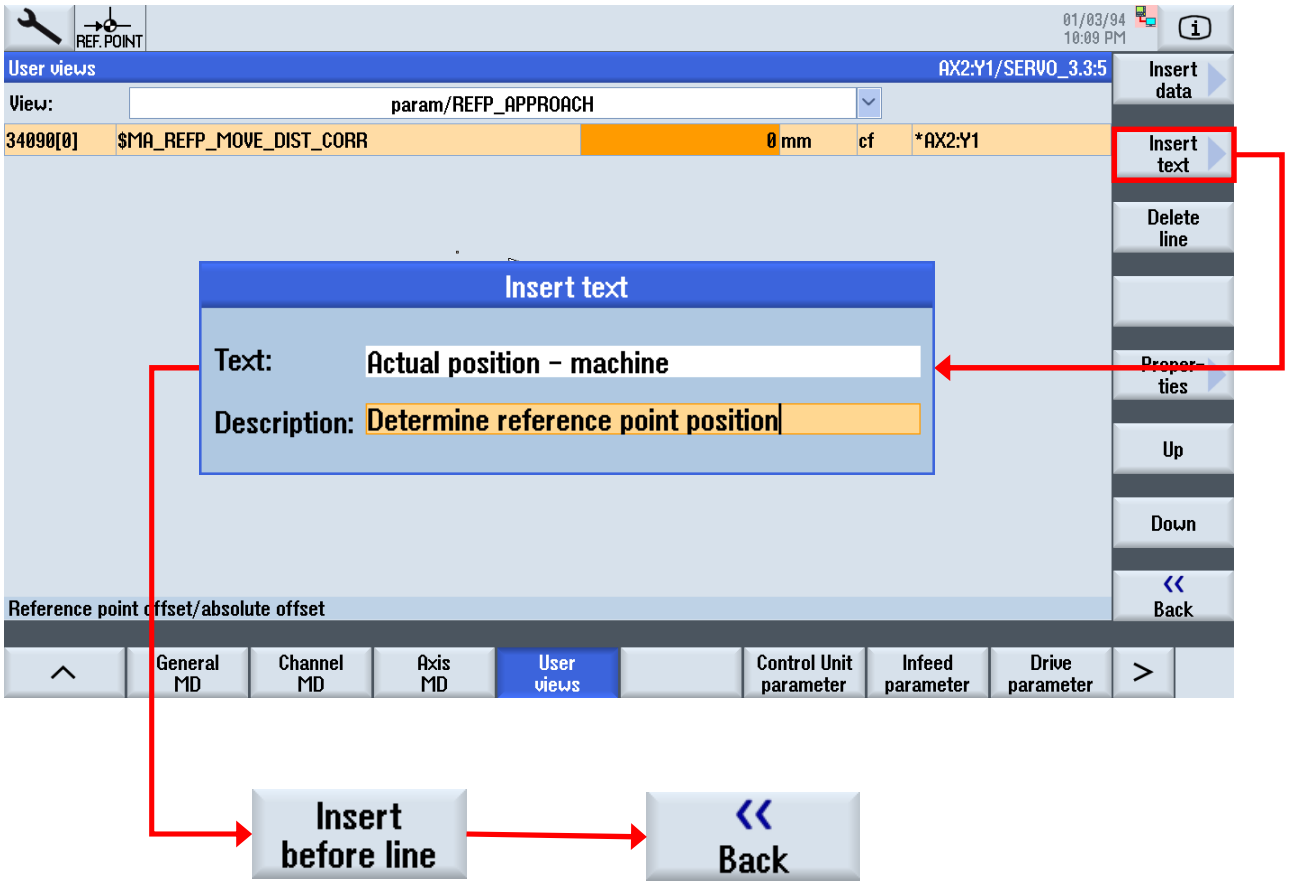
Select the required machine data, e.g. using the **softkey "Search"**.



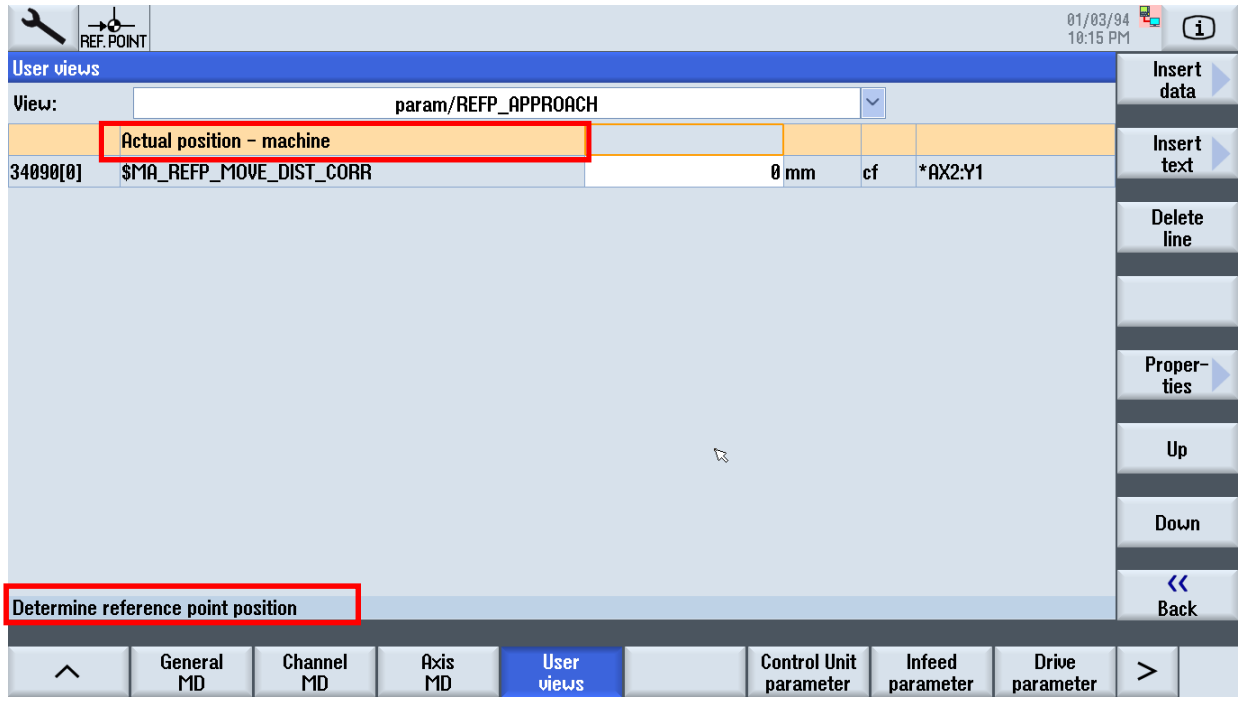


Then, with **softkey "Insert before line"/"Insert after line"**, place the machine data in the user view. With softkey: "Back", you return to the basic menu to edit the user view.

Now you can enter a text for the variable and plain text display. This is activated with the **softkey "Insert before line"/"Insert after line"**
and **softkey: "Back"**



The result looks like this:

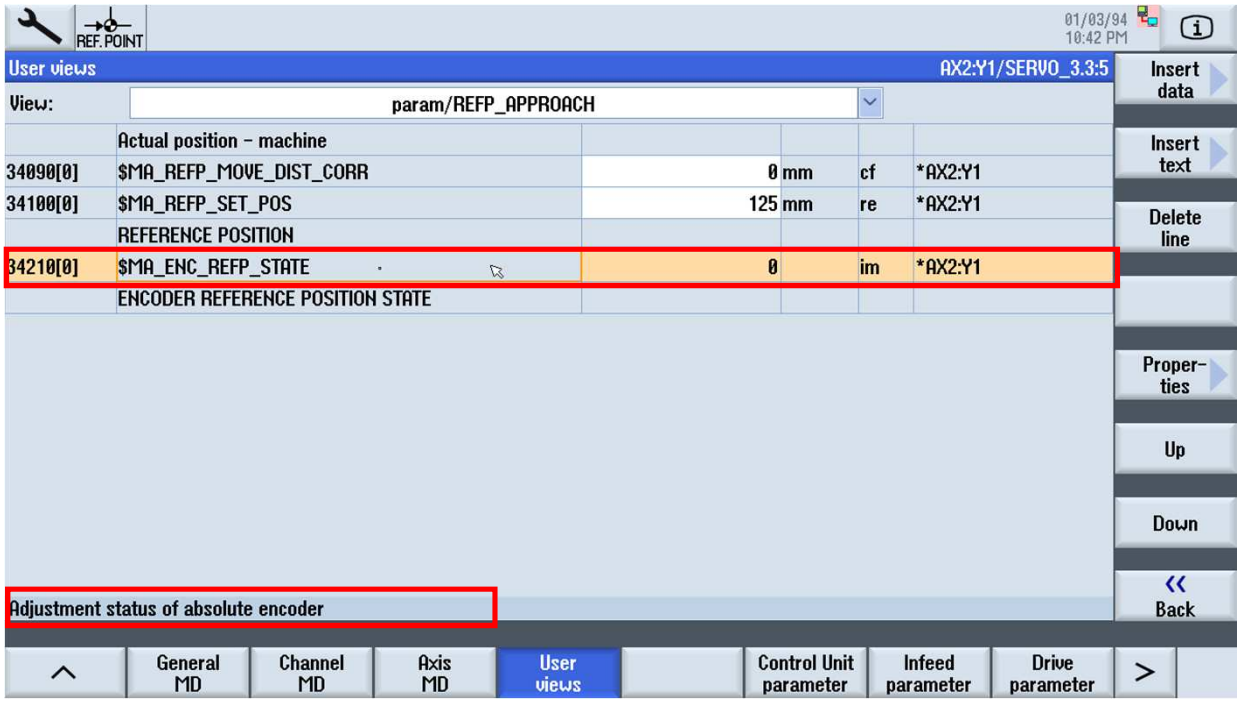


If you wish, you can now supplement this by adding the following MD:

MD 34100 Reference point position or adjustment position, absolute encoder

MD 34210 Adjustment status, absolute encoder

and add the corresponding text, e.g. including the adjustment technique under the topic 'Description', the text 'Absolute encoder adjustment' etc:



Using the user view

The following example shows how to use this user view:
To start, you should adjust the X axis encoder.

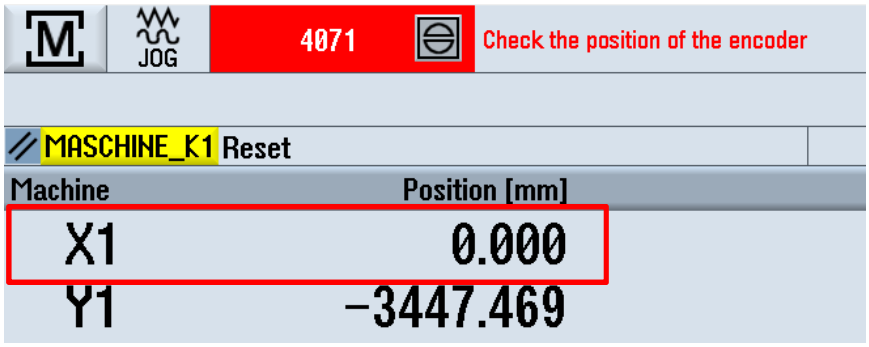
Adjustment X

To do this, select MD 34090 and select the X axis.

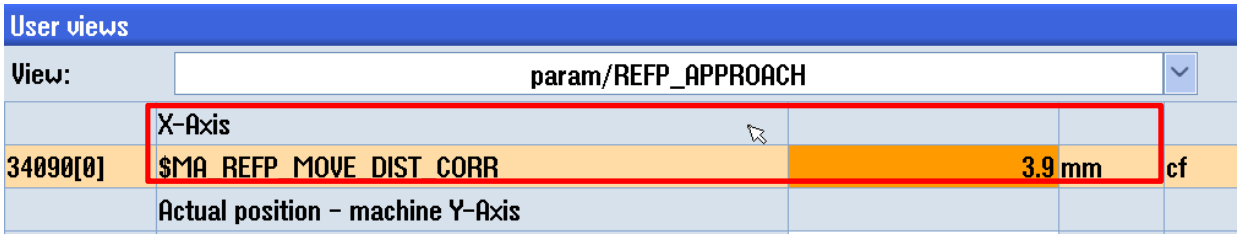
Determine the offset using the reference point approach and approach the mechanical adjustment position.

Example: Set position = 75 mm

Actual position (machine) = 78.9 mm



-> correction, MD 34090 by 3.9 mm



Adjustment Y (Z)

Now select MD 34210 and select the Y (Z) axis
with absolute encoder. Traverse to the mech. adjustment position,
enter the required position value in MD 34100
and confirm using the JOG-REF and + key on the MCP.

	Actual position - machine Y-Axis				
34090[0]	$MA_REFP_MOVE_DIST_CORR	-636.001624856572	mm	cf	*AX2:Y1
34100[0]	$MA_REFP_SET_POS	125	mm	re	*AX2:Y1
	REFERENCE POSITION				
34210[0]	$MA_ENC_REFP_STATE	2		im	*AX2:Y1
	ENCODER REFERENCE POSITION STATE				

-> MD34090 is automatically determined,
MD 34210 changes to 2 (encoder adjusted).

Additional exercises:

1. Channel specific referencing

With the NC Start key, the configured channels are to be referenced in Jog Ref mode. For this, search for the suitable interface signals in DB11 (mode group) and DB21 ff. (channel interface) on DocOnCD.

2. MCS/WCS switchover

The MCS/WCS key of the operator interface still has no function.
The key is put into operation using a small PLC program.
For this, search for the suitable interface signals in DB 19 (DocOnCd).

3. Override switch

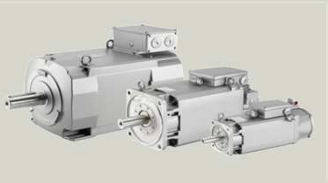
The rotary override switches still do not have the expected effect.
Examine the function for Jog mode, for program operation, and for rapid traverse overlay.
Set the necessary bits in the channel and axis interface to achieve the correct function. For this, search for the suitable interface signals in DB 21..30 and DB31..61 (DocOnCd).

SIEMENS









Chapter 9.2

Axis start-up: Spindle settings

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Objectives 2
Start-up sequence 3
Spindle modes 4
Definition and configuration of a spindle 5
Parameter sets 6
Gear stages 7
Spindle monitoring functions 8
Spindle gear stage change sequence 9
Calibrating the measuring system 10
Instructions: Machine data for spindle 11

Objectives

The participant will ...

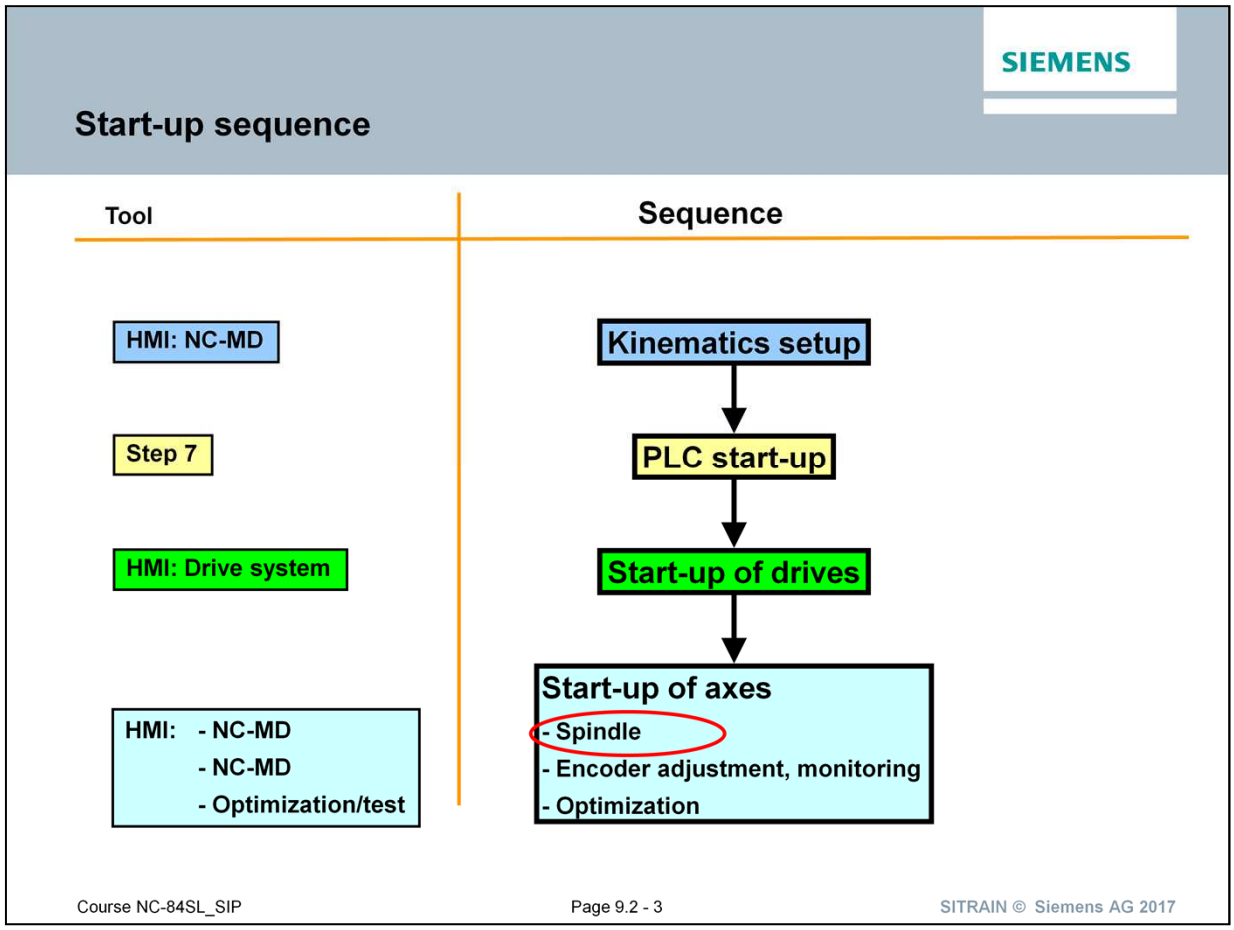
... Become familiar with spindle modes

... Become familiar with spindle start-up

... Become familiar with calibrating the measuring system

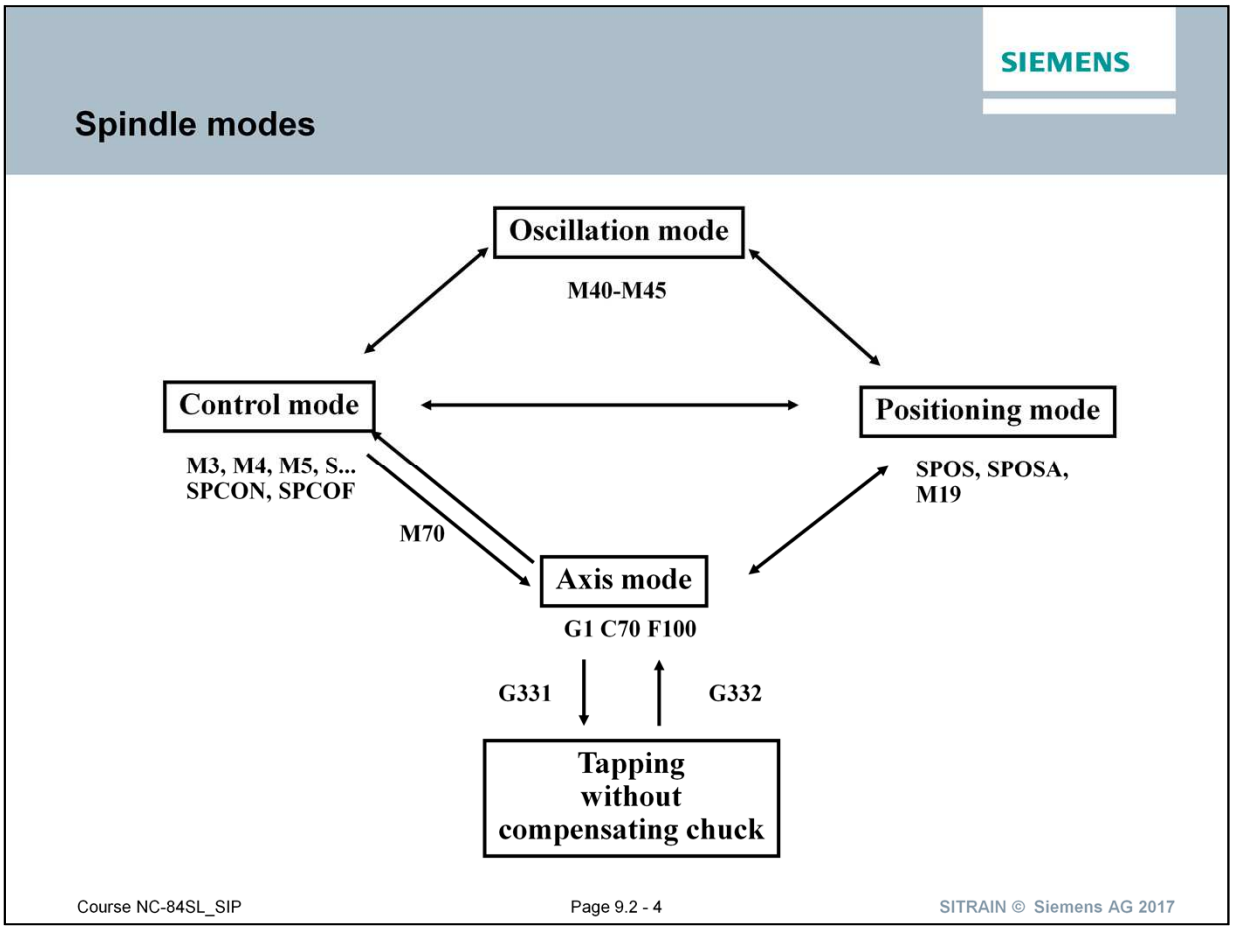
SITRAIN © Siemens AG 2017

Notes



Start-up part

- Axes Special settings for a spindle



Spindles

The primary function of a spindle is to set a tool or workpiece in rotary motion to facilitate machining.

Control mode

After specifying a spindle rotation direction (M3, M4) and a spindle speed (S....), the spindle rotates in control mode, i.e. the actual speed is not constant but fluctuates around the setpoint speed.

SPCON/SPCOF

If a constant speed is required, the SPCON command can be used to activate position control. The command is deselected with SPCOF.

Notice: With SPCON, the maximum speed is automatically limited to 90%!

Positioning mode

The SPOS, SPOSA, and M19 commands are available for positioning the spindle, e.g. for tool change. The positioning data for M19 is stored in the axis-specific setting data SD 43240: M19_SPOS[n] (position) and SD 43250: M19_SPOSMODE[n] (position approach mode). M19 is available with software 5.3 and higher.

Example:		
	SPOS=90	Position spindle 1 to 90º
	SPOSA[2]=30	Position spindle 2 to 30º
	M19	Position spindle 1 to SD 43240

Oscillation mode

Oscillation means that the spindle motor rotates alternately in the clockwise and counterclockwise directions. This oscillation movement makes it easy to engage a new gear stage. Oscillation can be implemented via machine data, the PLC program, or FC18.

Gear stage change

The spindle changes to oscillation mode if a new gear stage has been specified using automatic gear step selection (M40) in conjunction with a new S value or by M41 to M45. The spindle only changes to oscillation mode if the new gear step is not equal to the current actual gear step.

Axis mode

For certain machining tasks (e.g.:on turning machines with end face machining) the spindle can be addressed in the part program as an axis with its axis identifier (for example: C).If axis mode is active and the rotary axis is homed, all axis functions can be used.

Encoderless operation

Encoderless operation is possible for spindles with increased speed

Definition and configuration of a spindle

Definition:

MD 30300 =1 rotary axis (IS_ROT_AX)

MD 30310 =1 rotary axis with modulo programming (ROT_IS_MODULO)

MD 30320 =1 display relative to 360 degrees (DISPLAY_IS_MODULO)

MD 35000 =1 declaration of axis as 1st spindle (SPIND_ASSIGN_TO_MACHAX)

=2 declaration of axis as 2nd spindle

SITRAIN © Siemens AG 2017

Rotary axis

With SINUMERIK, the spindle is an endlessly turning rotary axis. Machine data MD30300, 30310, 30320 =1 must be set to define the spindle.

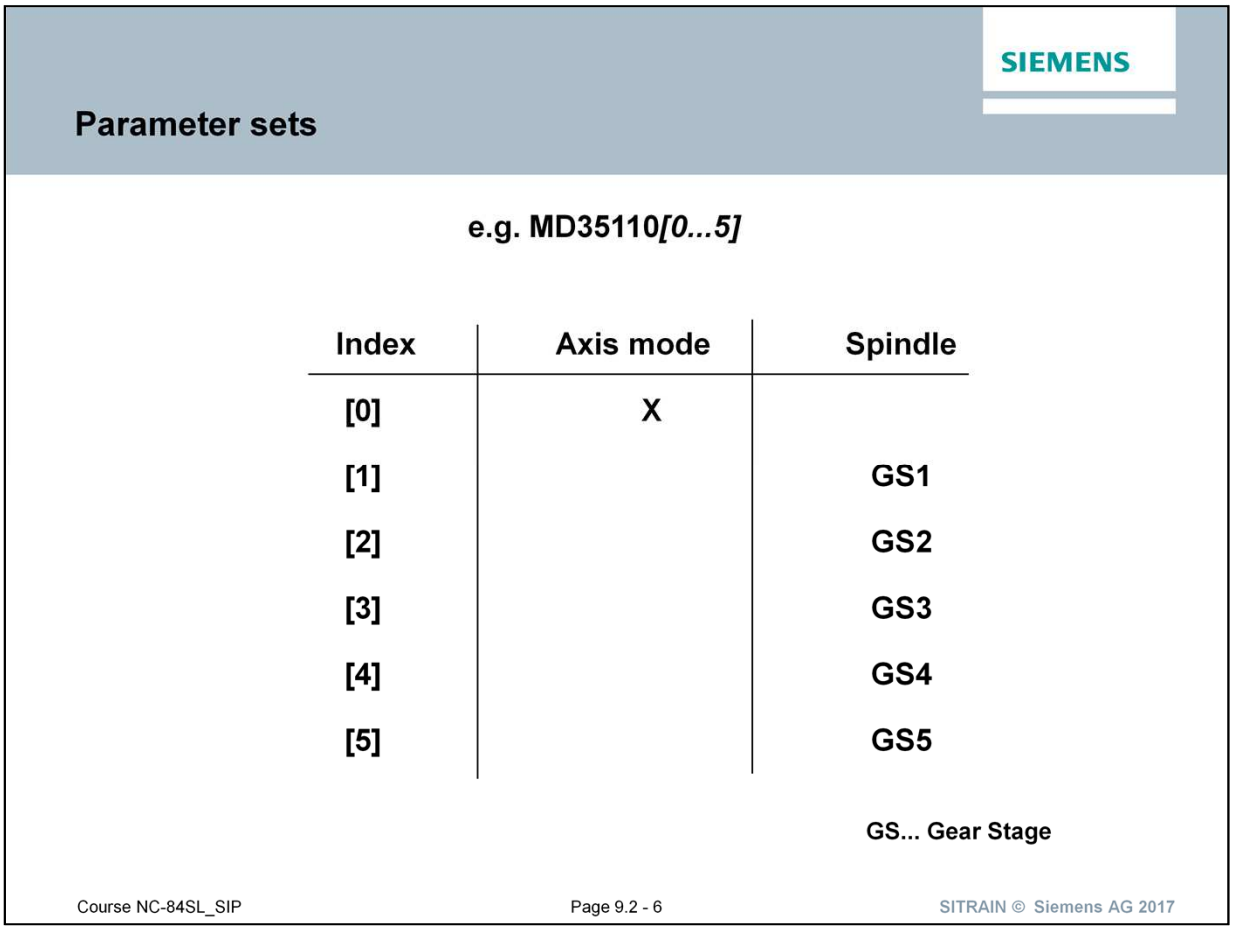
You need to perform an NCK reset after the rotary axis has been defined.

Spindle

The spindle number for the axis is defined in axis-specific machine data MD35000. The spindle number is used for programming parts.

Example:

M3 S1000	Spindle 1 rotates clockwise at 1,000 rpm
M2=3 S2=1000	Spindle 2 rotates clockwise at 1,000 rpm



Parameter sets

For the following machine data, the index [n] stands for axis operation [0] or for gear stages 1 [1] to 5 [5].

MD 35110: Maximum speed for gear stage change (GEAR_STEP_MAX_VELO[n])

MD 35120: Minimum speed for gear stage change (GEAR_STEP_MIN_VELO[n])

MD 35130: Maximum speed for gear stage (GEAR_STEP_MAX_VELO[n] _LIMIT)

MD 35140: Minimum speed for gear stage (GEAR_STEP_MIN_VELO_LIMIT[n])

MD 35200: Acceleration in speed control mode (GEAR_STEP_SPEEDCTRL_ACCEL[n])

MD 35210: Acceleration in position control mode (GEAR_STEP_POSCTRL_ACCEL[n])

MD 31050: Denominator load gearbox (DRIVE_AX_RATIO_DENOM[n])

MD 31060: Numerator load gearbox (DRIVE_AX_RATIO_NUMERA[n])

MD 32200: Servo gain factor (POSCTRL_GAIN[n])

MD 36200: Threshold value for velocity monitoring (AX_VELO_LIMIT[n])

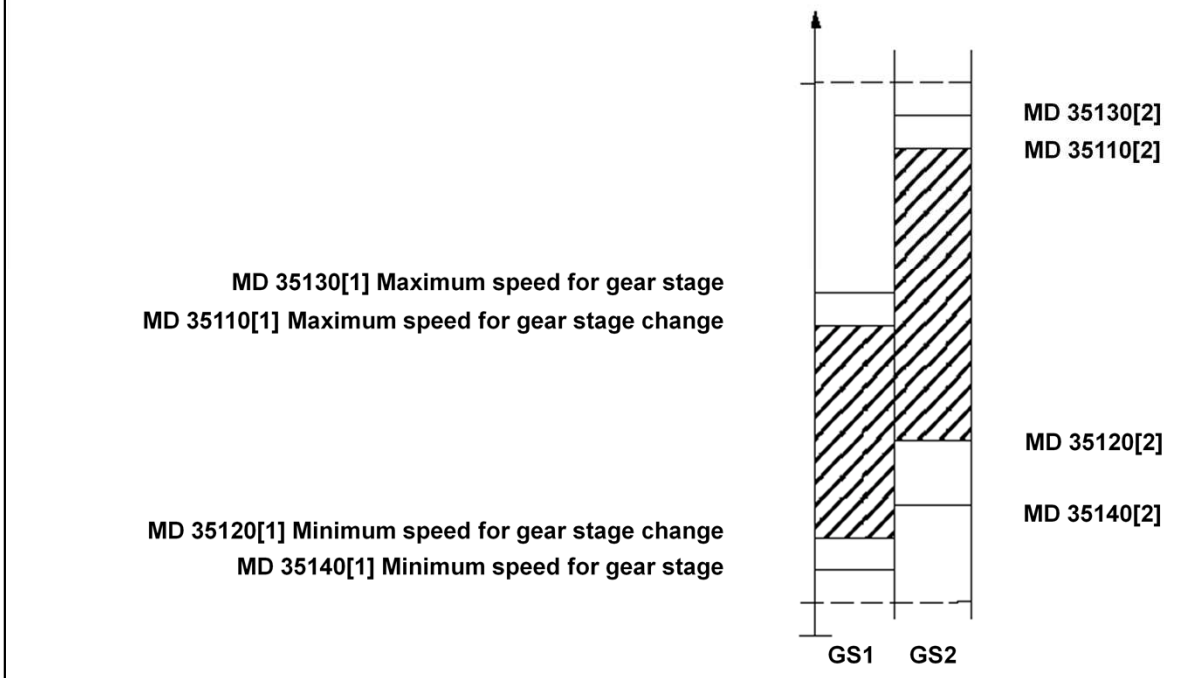
MD 35590

This machine data is used for axes to define whether or not switching between these parameter sets is possible via the PLC (DB DB31... DBB9).

Note

The spindle **Setting Data** should also be checked!

Gear stages



Course NC-84SL_SIP Page 9.2 - 7 SITRAIN © Siemens AG 2017

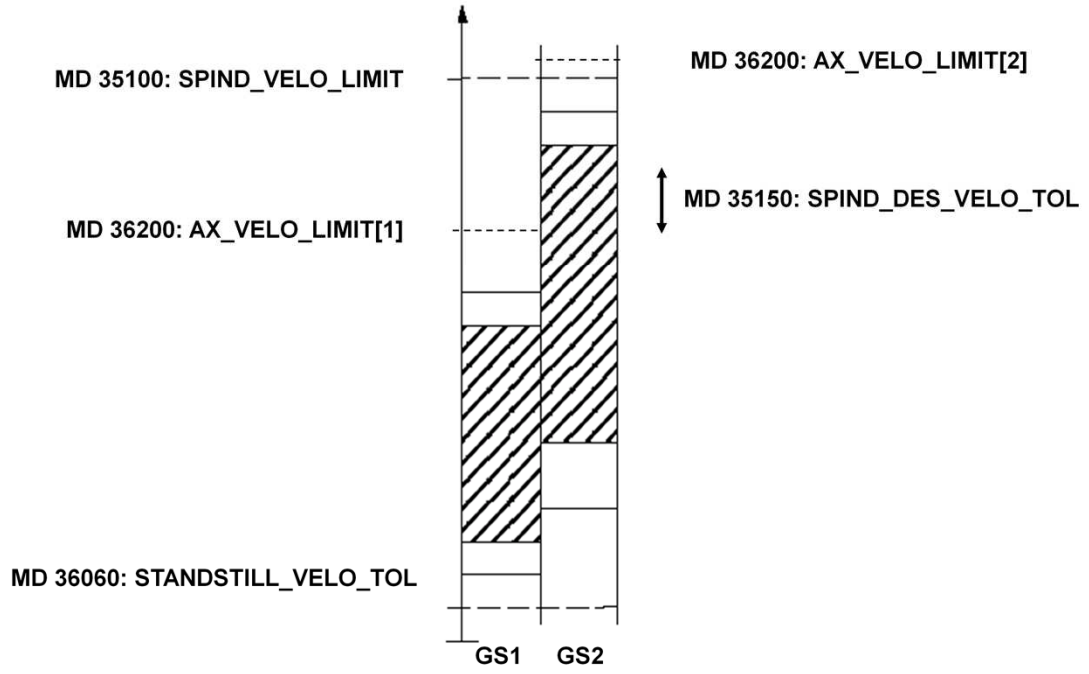
MD35110[n] The maximum speed of the gear stage for automatic gear stage change (M40) is specified in the maximum speed for gear change MD (GEAR_STEP_MAX_VELO)

MD35120[n] The minimum speed of the gear stage for automatic gear change (M40) is specified in the minimum speed for gear change MD (GEAR_STEP_MIN_VELO).

MD35130[n] The maximum speed for gear stage is specified in the maximum speed for gear stage MD (GEAR_STEP_MAX_VELO_LIMIT). This speed can never be exceeded in the gear stage.

MD35140[n] The minimum speed for gear stage is specified in the minimum speed for gear stage MD (GEAR_STEP_MIN_VELO_LIMIT). The speed cannot fall below this value, even if a very low S value is programmed.

Spindle monitoring functions



SITRAIN © Siemens AG 2017

Maximum spindle speed

The maximum spindle speed is specified in **MD 35100**: SPIND_VELO_LIMIT.The NCK limits the speed to the specified value. If the speed is still exceeded by the speed tolerance (drive error), alarm "22150 channel [name] block [number] spindle [number] maximum chuck speed exceeded" is output.

MD 36200: AX_VELO_LIMIT[1...5] is used for gear-stage dependent monitoring of the spindle speed; if the speed is exceeded, alarm "25030 axis [name] actual speed alarm limit" is generated.

Progr. spindle speed limitation

The functions G25 S... and G26 S... are used to preset a minimum spindle speed and a maximum spindle speed respectively via the part program. The limitation is active in all operating modes.

Spindle setting data

The operator or machine setter can set an additional limit for the spindle speed in the "Parameters" area, "Setting data" softkey, "Spindle data" softkey.
Notice: If you set the speed to "0", the spindle is motionless in all operating modes and no error message is issued!

Axis/spindle standstill

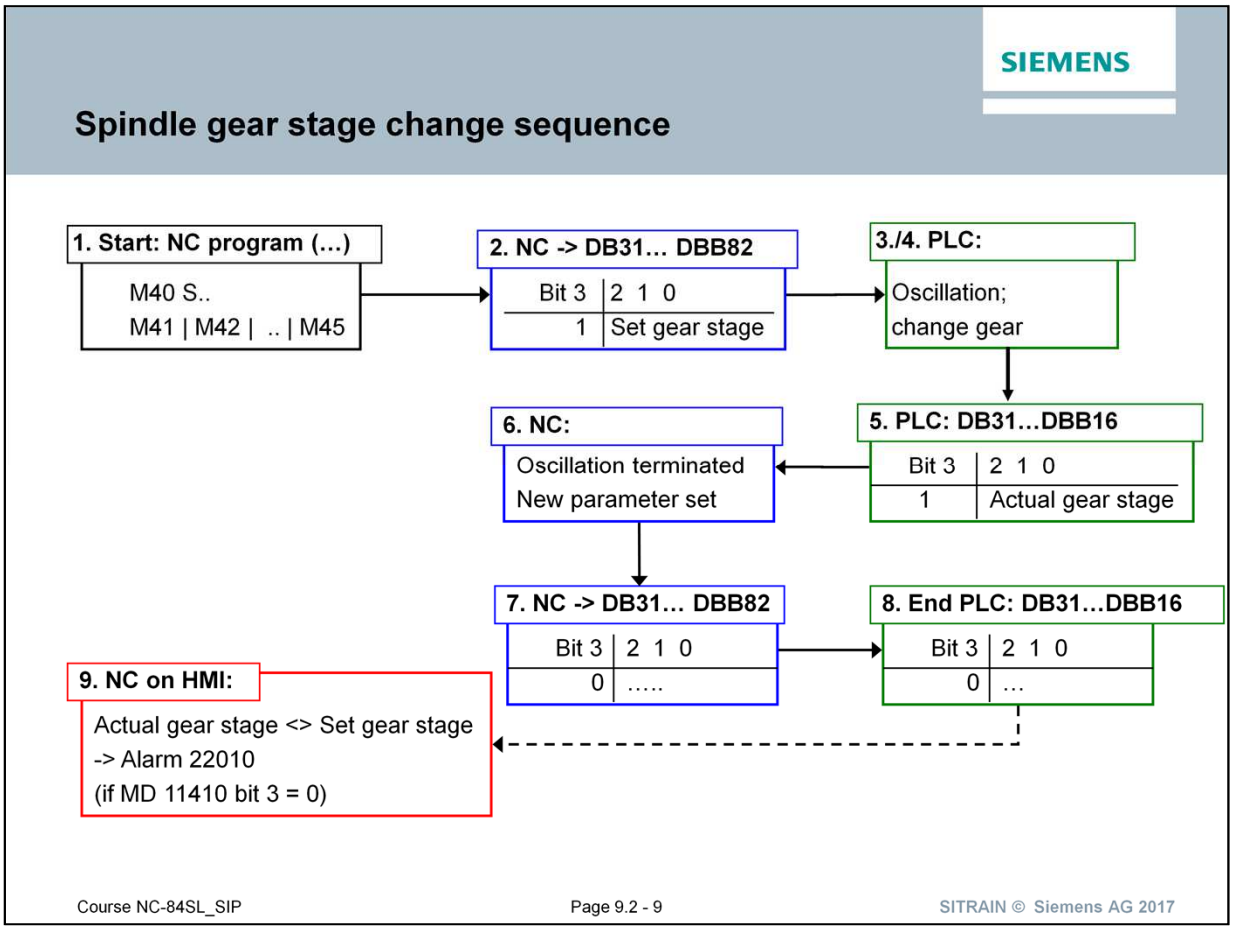
If the speed is below the speed set in **MD 36060**: STANDSTILL_VELO_TOL, this is displayed via interface signal DB31.DBX61.4 "Axis/spindle standstill".
If **MD 35510**: SPIND_STOPPED_AT_IPO_START is set, the path feed is then enabled e.g. after M5.

Spindle in setpoint range

If the spindle reaches the tolerance range specified in **MD 35150**: SPIND_DES_VELO_TOL, the interface signal DB31.DBX83.5 "Spindle in setpoint range" is output. If **MD 35500**: SPIND_ON_SPEED_AT_IPO_START is set, path feed is then enabled.

Notice!

Contour precision MD36400, zero-speed tolerance MD36030, and exact stop window MD36000, MD36010 are also monitored for the spindle in position control.



Gear stage change sequence

A complete gear stage change consists of the following steps:

1. Gear stage change is triggered by programming M40 S.. or M41 … M45 (and other variants, see Chapter S1 Spindles of the Function Manual)
2. The NC makes the following settings in the basic program:

   DB31..DBX82.3=1 Gear stage change requested and
   DB31.. DBX82.2|1|0 Set gear stage
3. If applicable, the PLC user program sets oscillation DB31..DBX18.5. The spindle motor is then decelerated by the NC and set to oscillation mode to disengage/engage the gear.
4. The PLC user program is responsible for the mechanical changing of the gear.
5. The gear change is acknowledged by the PLC user program with:

   DB31…DBX16.3=1 Gear has changed
   DB31…DBX16.2|1|0 Actual gear stage
6. Oscillation mode is terminated by the NC and change-over to the new servo parameter set takes place in the stop state.
7. The NC acknowledges the change-over with:

   DB31..DBX82.3->0 End of gear step change
8. This must be confirmed by the PLC user program in turn:

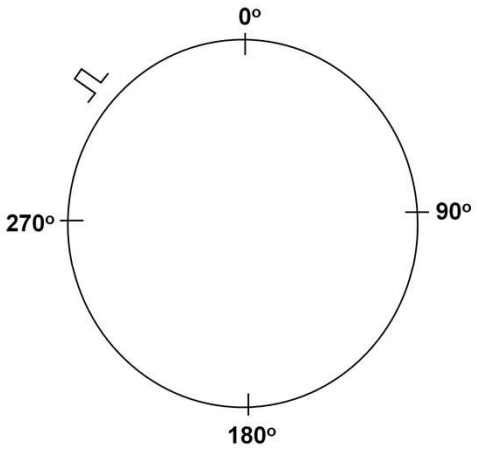
   DB31..DBX16.3->0 End of gear step change

For logical reasons, now is also the latest point in the process at which the interface signal for oscillation DBX18.5 is reset.

9. If the actual gear stage does not match the set gear stage, alarm 22010 is generated.

This alarm can be suppressed by setting MD 11410 bit 3 = 1.

Calibrating the measuring system



Requirements:
MD 34060=360 Max. distance to reference mark (REFP_MAX_MARKER_DIST[n])

The measured correction value is written in
MD 34090: Reference point offset (REFP_MOVE_DIST_CORR).
Notice: This value only becomes effective after an NCK reset.

SITRAIN © Siemens AG 2017

Synchronization

When the control is switched on, the spindle can be synchronized as follows:

- The spindle is started with a spindle speed (S value) and a spindle direction of rotation (M3 or M4) and synchronizes with the next zero marker of the position measuring system or with the next BERO signal.
- The spindle is positioned from standstill using SPOS, M19 or SPOSA or from the motion. The spindle synchronizes with the next zero mark of the position measuring system or with the next BERO signal. Then It is then positioned at the programmed position.
- Overtravel of the zero mark in JOG mode using the direction keys in speed-controlled operation.

Calibrating the zero mark

When the spindle is synchronized, the position of the zero mark is defined as actual position = 0. If this position does not match the actual zero position, the position must be corrected via the machine data.

Variant 1
MD 34060=360 Max. distance to reference mark (REFP_MAX_MARKER_DIST[n]) must be set as a prerequisite for the calibration.

The measured correction value is written in MD 34090: Reference point offset (REFP_MOVE_DIST_CORR).

Notice: This value only becomes effective after an NCK reset.

Variant 2
SPOS=0
Program and run M70,
this sets the spindle to axis operation allowing it to be calibrated using reference point approach.

Instructions: Machine data for spindle

Definition

MD 30300 =1 rotary axis (IS_ROT_AX)
MD 30310 =1 rotary axis with module programming (ROT_IS_MODULO)
MD 30320 =1 display relative to 360 degrees (DISPLAY_IS_MODULO)
MD 35000 =1 axis is 1st spindle (SPIND_ASSIGN_TO_MACHAX)

Calibration of measuring system

MD 31000=0 ENC_IS_LINEAR
MD 31040=0 ENC_IS_DIRECT
MD 31020=1024 ENC_RESOL steps/rev.
MD 31080=1 DRIVE_ENC_RATIO_NUMERA motor rev.
MD 31070=1 DRIVE_ENC_RATIO_DENOM encoder rev.
MD 31060=1 DRIVE_AX_RATIO_NUMERA motor rev.
MD 31050=1 DRIVE_AX_RATIO_DENOM load rev.

Most of these MD are already preassigned with the 'Assign axis' function!

Gear stages

Enter the minimum and maximum speeds for gear stage 1
MD35130[1] =2500 maximum speed for gear stage (GEAR_STEP_MAX_VELO_LIMIT)
MD35140[1] =0 minimum speed for gear stage (GEAR_STEP_MIN_VELO_LIMIT)

JOG speeds

MD 32010 =2500 rapid traverse in jog mode (JOG_VELO_RAPID)
MD 32020 =500 JOG axis feedrate (JOG_VELO)

Monitoring functions

MD 35100 =3000 SPIND_VELO_LIMIT
MD 36200 =3000 AX_VELO_LIMIT[1]

Pay attention to the maximum spindle speed in the setting data!

Increase the tolerance window for contour precision MD36400, zero-speed tolerance MD36030, and exact stop window MD36000, MD36010 as the spindle has not been optimized.

Exercise: Reference point spindle

Set MD 34060=360 max. distance to reference mark (REFP_MAX_MARKER_DIST[n]).

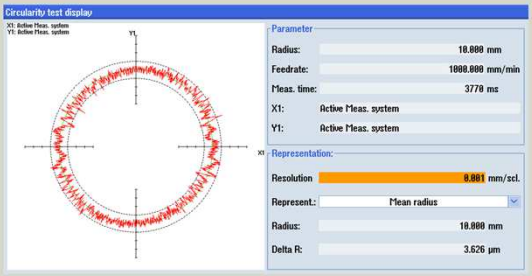
Adjust the zero mark by setting MD 34090: reference point offset (REFP_MOVE_DIST_CORR) accordingly until the centrifugal mass marker is indicating up 0°. You can use the SPOS=0 command for this purpose.

Once the setting has been adjusted correctly, the 0° marker for the centrifugal mass will be pointing upwards.

If it is not, move the spindle accordingly withe JOG or JOG-INC and read the correction for MD 34090 from the actual position in the Machine OA.
A power ON is required after adjusting MD 34090: SK: Reset (po)

.

SIEMENS



Chapter 9.3

Axis optimization

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
System clocks 3
Commissioning sequence 4
Control loops 5
Speed controller optimization - Setpoint step change 6
Position controller optimization 7
Automatic servo tuning (AST) 8
Circularity test with AST 13
Jerk limitation 16
Feedforward control 17
Trace 19
Exercises for SINUMERIK 840D sl commissioning 22
Exercise: Conventional optimization in time range 23
Tips on AST: Options 25
Tips on AST: Strategy 26
Tips on AST 28
Exercise for auto servo tuning 38
Additional exercise: Interpolation path 43
Exercise: Circularity test with AST 44

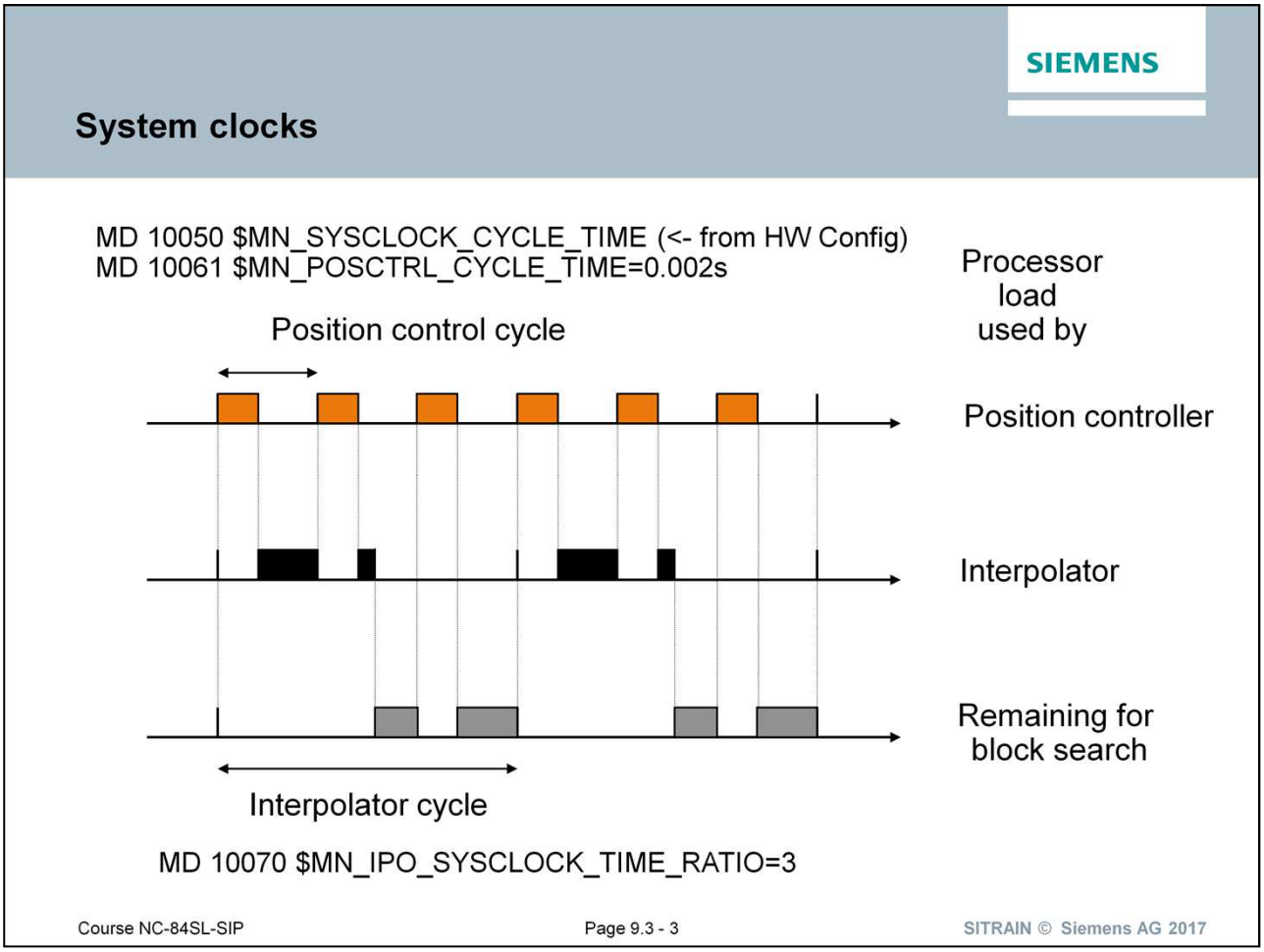
Learning objectives

- You will be familiar with the Auto Servo Tuning function
- You will be able to perform controller optimization.



SITRAIN © Siemens AG 2017

Notes



System clocks

As shown in the above diagram, the remaining block cycle time results from the intensity of the tasks called (entered time). The greatest load is placed on the system if the position control cycle is shortened on systems with many axes. The remaining available computing power makes up the lead time (preparation) and, therefore, the block cycle time that can be achieved on average.

Block change time

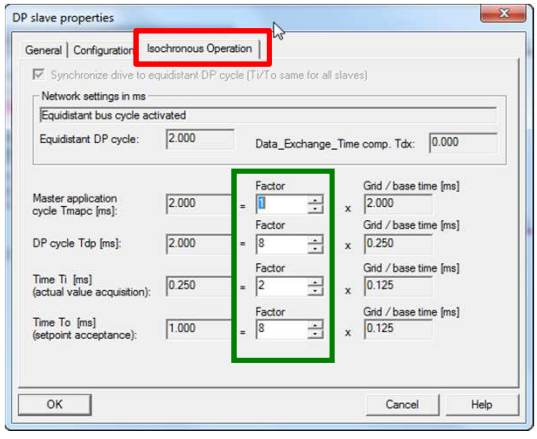
Stable execution of a block-intensive program is only guaranteed if the resulting lead time is <= IPO cycle +10%; otherwise, the temporarily empty interpolator buffer would result in random path feed in the following lines and distort the contour.

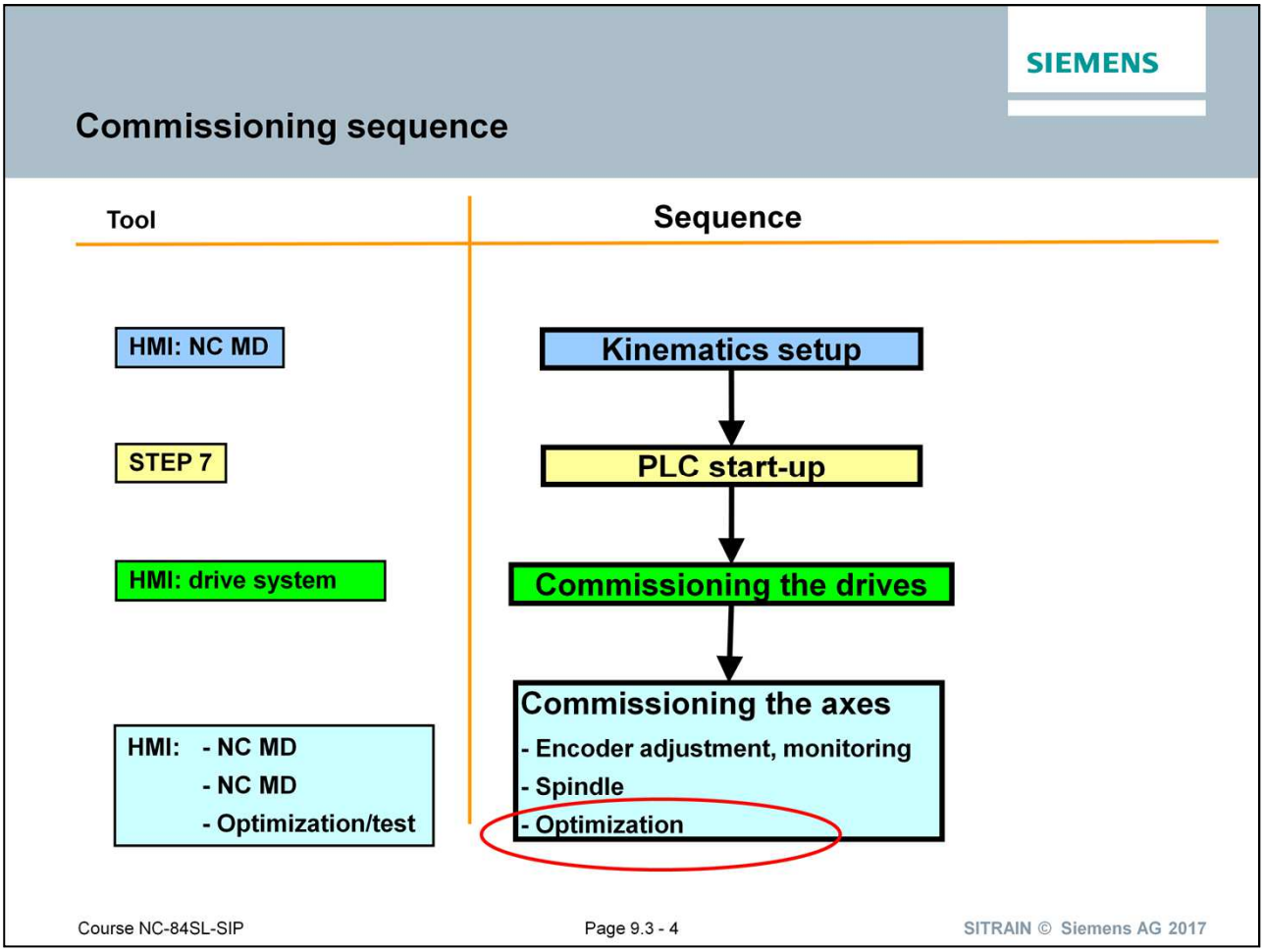
Note

On SINUMERIK 840D sl, the system clock is taken over from the setting **isochronous DP cycle**, to be set under:

DP slave properties -> tab: Isochronous operation

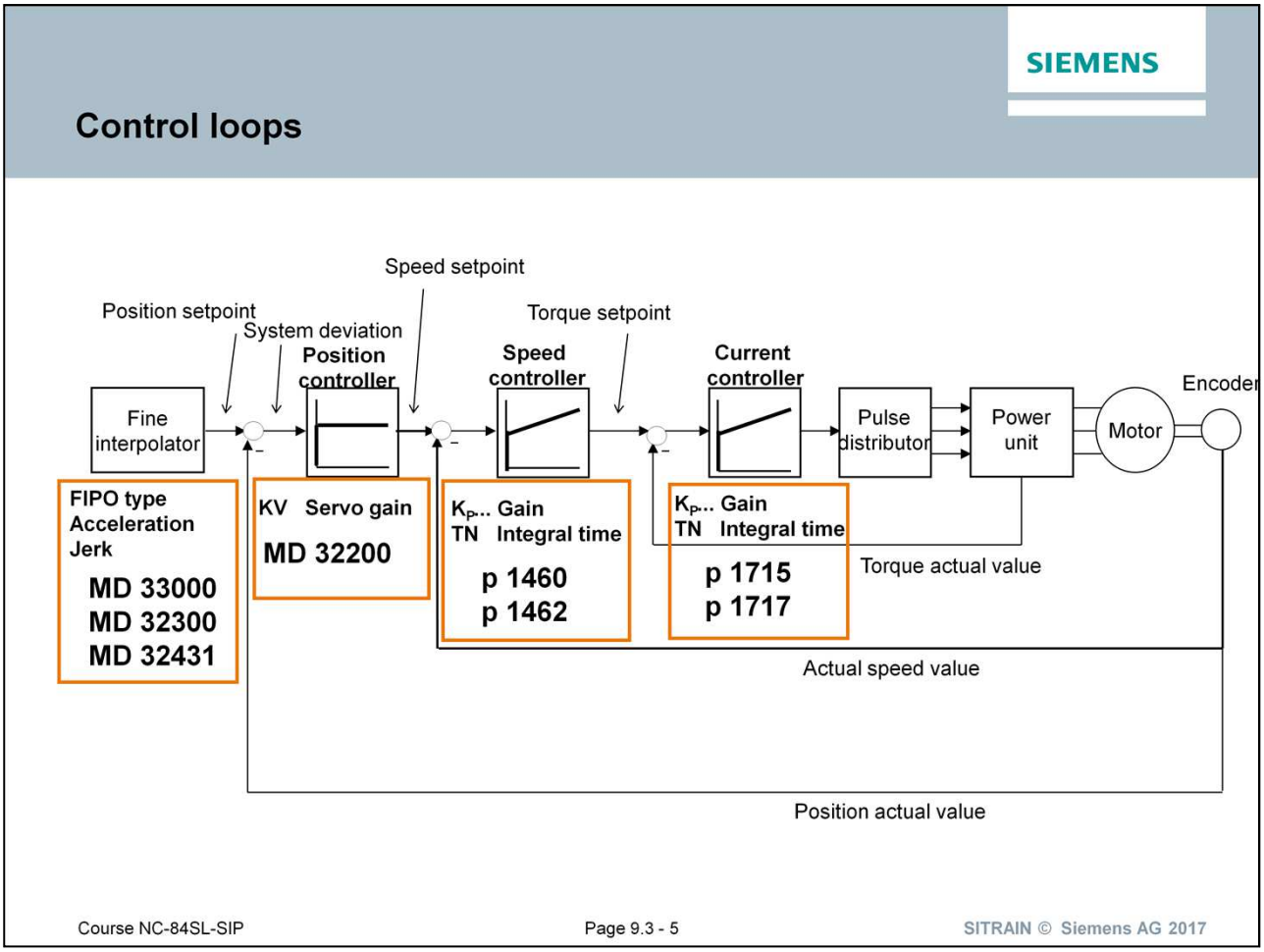
and it is determined by the set factors:





Commissioning part:
Axes

In this case the part:
- Controller optimization



Control loop structure The control loop of a feed axis is structured as a controller cascade. The innermost control loop is the current controller, which is secondary to the speed controller. Another primary controller, the position controller, supplies the speed setpoint.

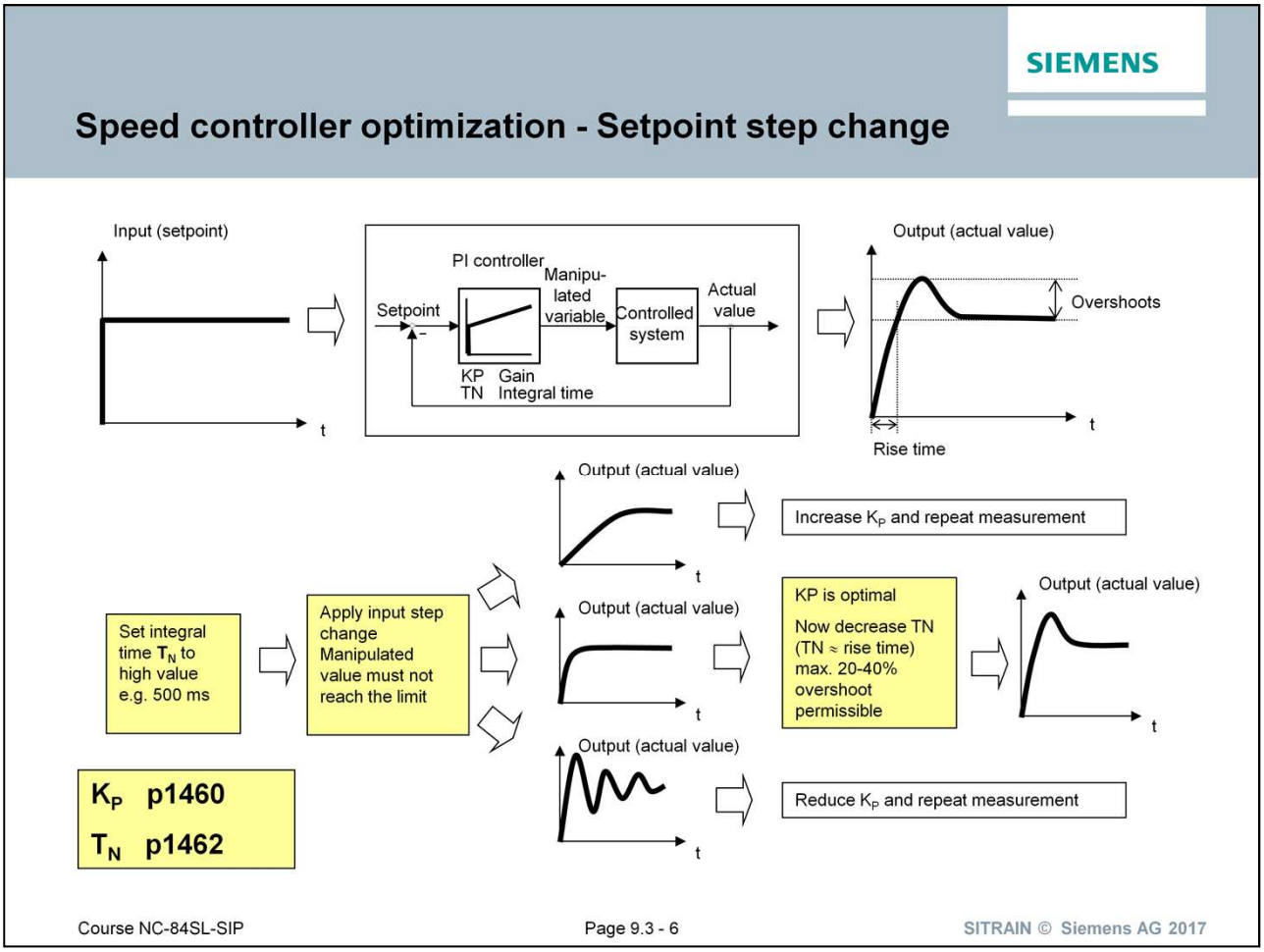
Position controller parameters The following are important interpolator and position controller parameters:

MD 10050 Position control cycle (= system clock): **is set in HW Config!**
MD 10070 Interpolation cycle/position-control cycle ratio
MD 33000 FIPO type recommended: = 3
MD 32300 acceleration
MD 32200 Kv proportional gain of position controller
MD 32431 axial jerk limitation

Cycle times SINAMICS

With SINAMICS S120, software blocks are exclusively used to implement the controllers with their typical responses. The following generally applies: The shorter the cycle time can be selected for a control loop, the better the results achieved. The cycle times are set with drive parameters or machine data.
p115[0]: Sampling time current controller default value 125 µs
p115[1]: Sampling time speed controller default value 125 µs

Parameters for speed and current controller The following are important speed controller and current controller parameters:
p1460: Kp speed controller
p1462: Tn speed controller
p1715: Kp current controller
p1717: Tn current controller



Speed controller

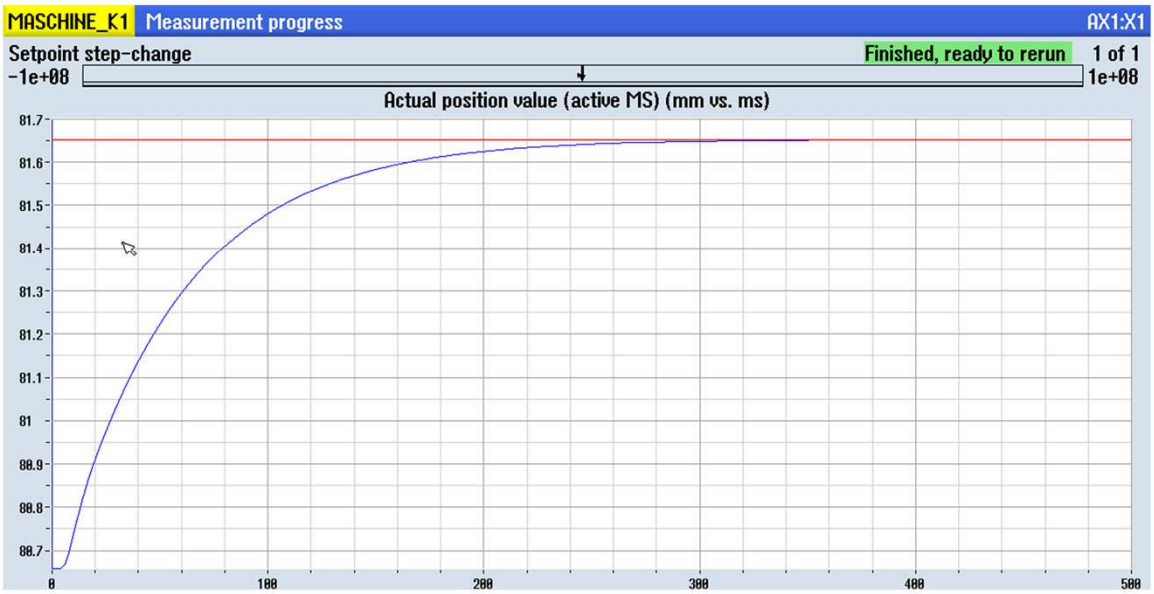
The controlled system of the speed controller is the mechanics for which the controller has to be optimized. Classic optimization with the symmetrical optimum is usually sufficient.

SINUMERIK 840D sl also offers the possibility of optimization with frequency response analyses.

Current controller

The current controller is correctly preset with the selection of a motor type if a SIEMENS motor with a DRIVE-CLiQ interface is used.

Position controller optimization



 SITRAIN © Siemens AG 2017

Servo gain factor

In MD32200 POSCTRL_GAIN (servo gain factor), you enter the desired servo gain for the axes. The aim is to achieve a large servo gain.
This results in:

- small following error/system deviation
- small circle radius error
- faster block change
- greater contour accuracy

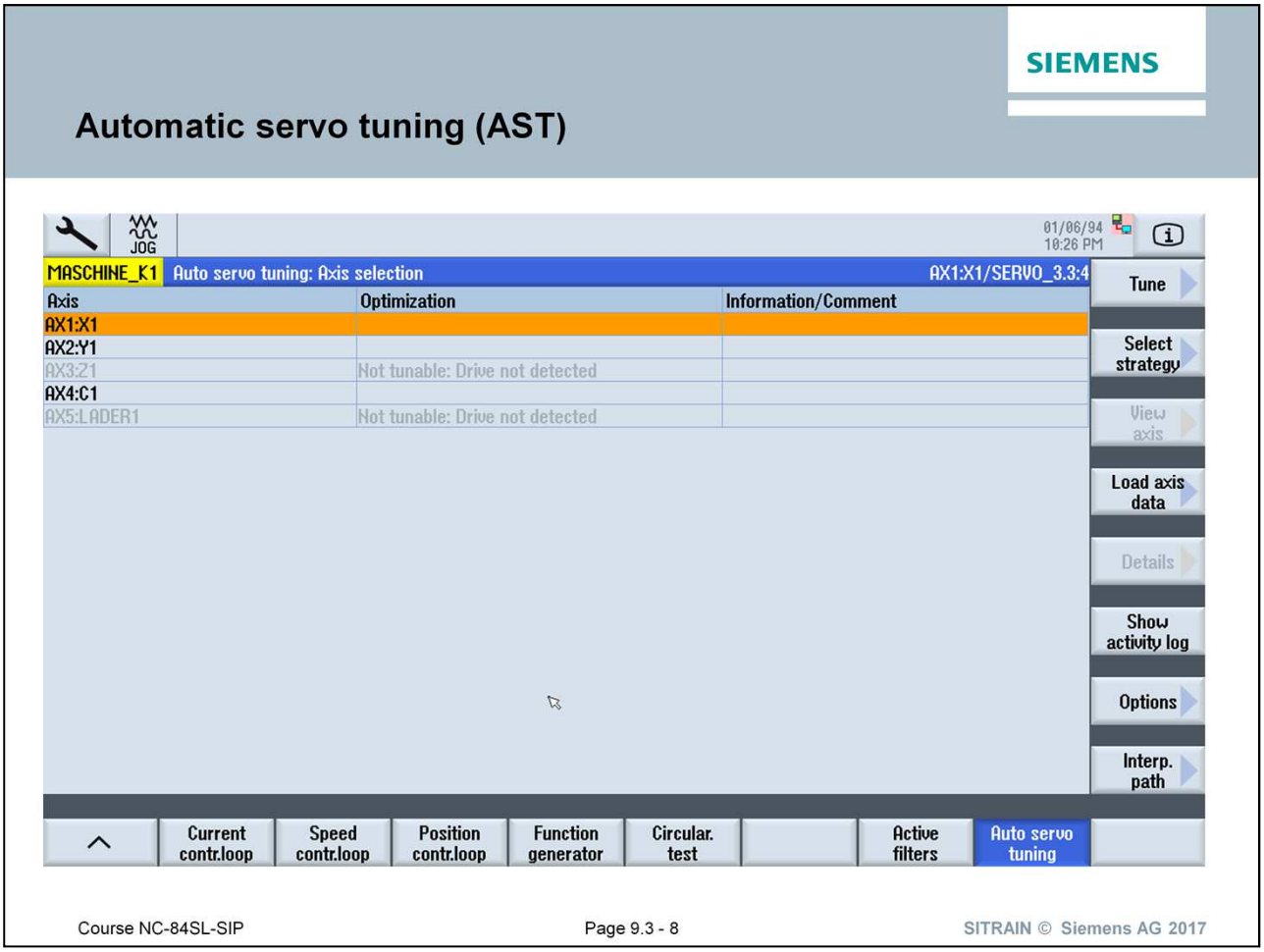
The magnitude of the servo gain factor is limited. There must be no overshooting when a setpoint ramp is specified nor during travel to the position.
To avoid contour errors, all axes participating in the interpolation must have the same servo gain factor.
The servo gain is calculated by the

$$K_v = \frac{\text{Speed}[\text{m/min}]}{\text{Followingerror}[\text{mm}]}$$

formula.

Acceleration

The axes are accelerated and decelerated with the value entered in MD32300: MAX_AX_ACCEL (axis acceleration). This value should allow the axes to be accelerated and positioned rapidly and accurately while ensuring that the machine is not unduly loaded. The default values for acceleration are in the range from 0.5 m/s^2 to 2 m/s^2.



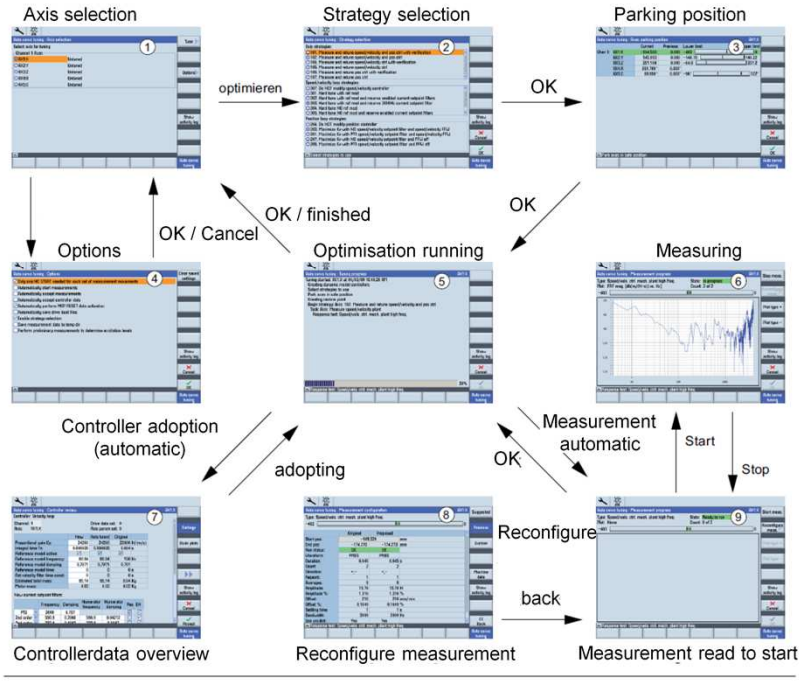
Auto servo tuning (AST)

With the SINUMERIK Operate user interface under menu **"Setup" > "Optimiz.Test",** you have following options for automatic axis optimization:

- Selection of a single axis for optimization
- Selection of a strategy among several options
- Reconfiguration of the measurement conditions
- Progress display and activity log for optimization
- Display of the present and, at the same time, the average prior measurement
- Checking and processing of the optimization results for speed and position controller
- Application or rejection of the results

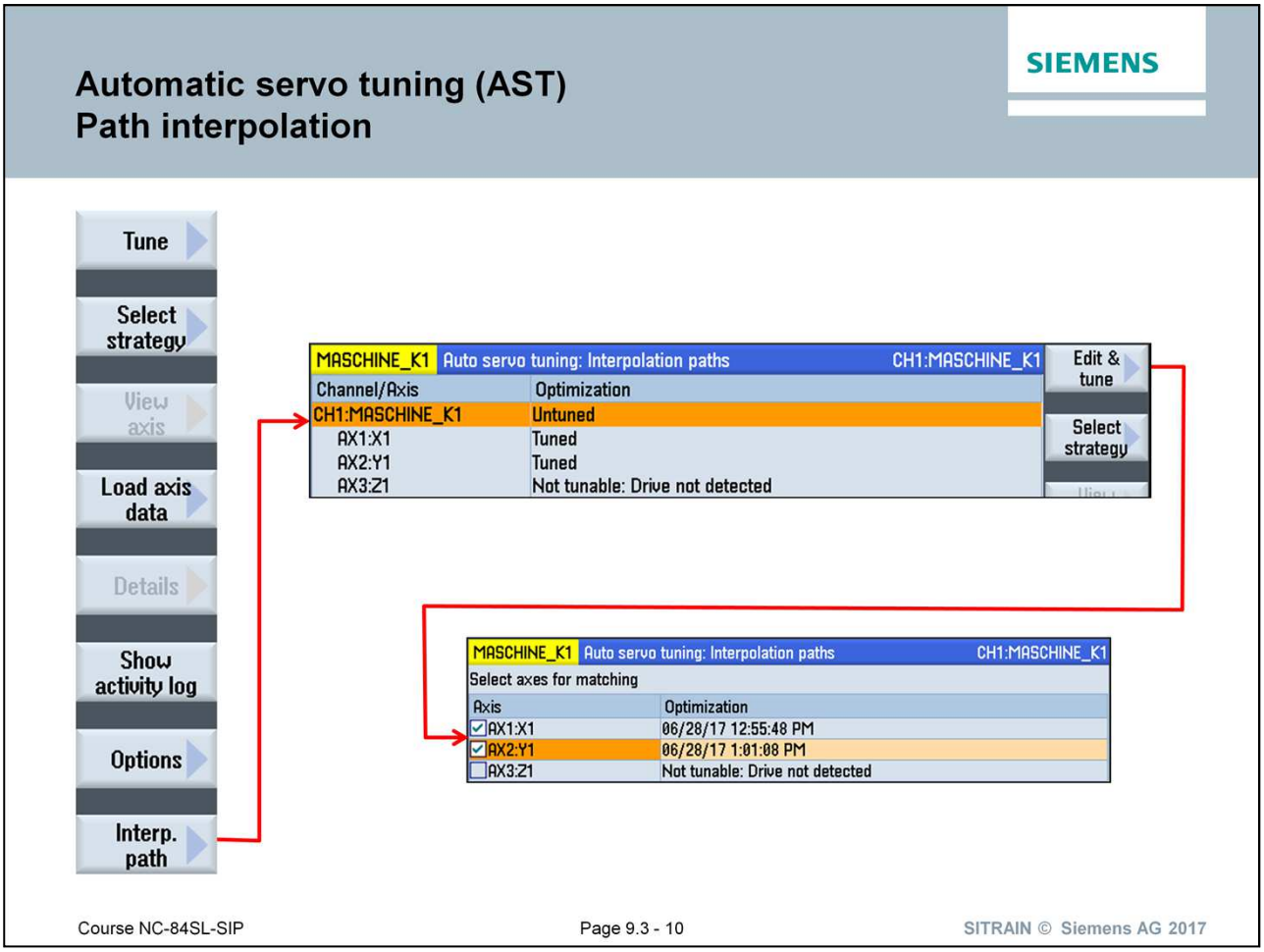
The "optimization" function can be used for all single axes.

Automatic servo tuning (AST)
Overview of the optimization workflow



SITRAIN © Siemens AG 2017

1. In the "Setup" operating area, press the "Auto. Servo Tuning" softkey. The "Axis selection" dialog screen appears.
2. Use the cursor keys to select an axis or a gantry axis grouping for optimization.
3. In the "Axis selection" dialog screen, press the "Optimize" softkey.
4. In the "Strategy selection" dialog screen, select the strategy for optimization. This is how the mechanical controlled system of the speed controller is measured, for example, and the gains and filters for an optimized dynamic response are defined.
5. In the "Axis parking position" dialog screen, move the axes to a safe position for optimization on the machine.
6. Press the "OK" softkey to start optimization. If measurement can only be continued after the inputs have been made, this is signaled by input prompts (screen forms). This is required because you need to initiate certain machine processes. You can abort the coordination process at any stage.
7. Once the optimization process has completed certain optimization routines for a control loop, a "Controller data overview" dialog screen appears. You can modify and check the results as well as accept or reject the proposed parameters.
8. Select "Accept" to apply the calculated values.
9. If you select "Accept" to apply the speed control settings, the drive data is updated and the strategy continues with the next step of taking the measurements for position control,
"Measurement in progress" dialog screen.
10. Once the optimum values for position control have been selected, the data is sent to the CNC and the drives, and the strategy continues with the next step.



Interpolation path

This menu (**-> softkey: Path interpolation**) enables you to adapt the position controller of interpolating axes.
Depending on the strategy, the following machine data or parameters are adapted completely or partially:

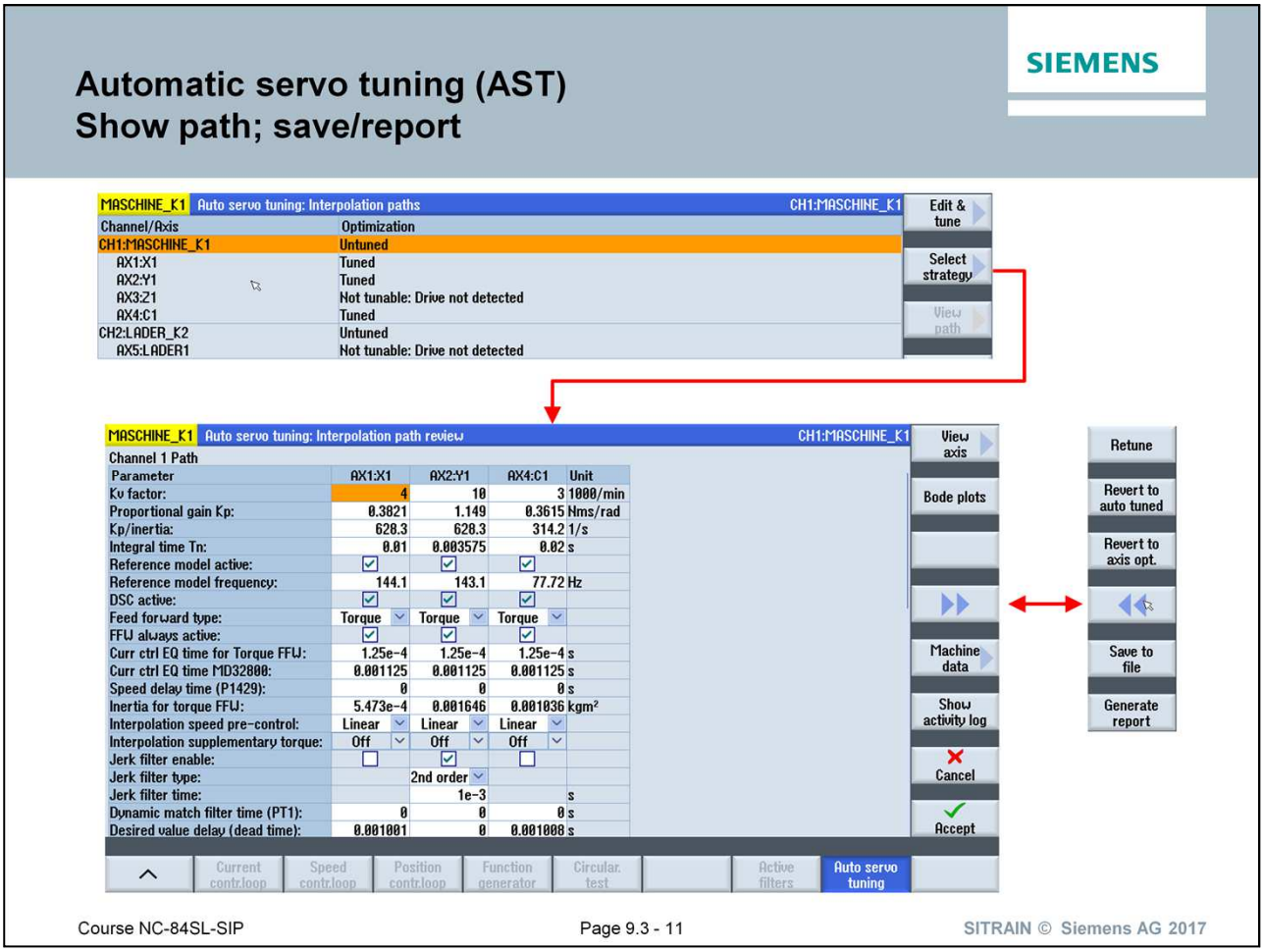
Servo gain factor:	MD 32200
Feedforward control:	MD 32610/32620/32630/32810
DSC active	MD 32640
Kp / moment of inertia	p1460 / p341
Integral time of speed controller:	p1462
Reference model data:	p1400 Bit 3; p1433, p1434

The data are displayed under softkey: "Show path data".

Note

If an error occurs at the start of this optimization routine, try the following:
1. Softkey: Display controller
2. Softkey: Accept
for all axes to be interpolated and restart optimization of the interpolation path.

Automatic servo tuning (AST)
Show path; save/report



 SITRAIN © Siemens AG 2017

Show path After optimization of the path interpolation, the results can be displayed with the **softkey: Show path data.**

Save to file After switching to a second vertical softkey bar with **>>** , the results can be saved to an xml file with **softkey: Save to file.**

Generate report An optimization report can also be generated in the form of an rtf file with: **softkey: Generate report.**
In this report, the measurement curves and the controller parameters are documented.

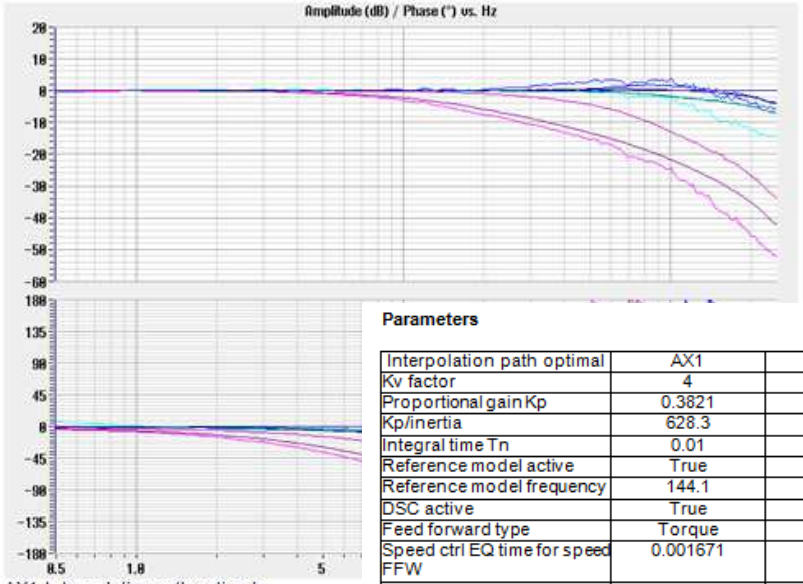
Example report (excerpt)

Machine

Session date	
Session time	
Machine axis name	
SAT version	04.07.03.00.014
Serial number	
Report date	1994-01-07
Report time	04:07:07
XML file	

Interpolation Path

Bode Plots



AX1 Interpolation path optimal:

AX2 Interpolation path optimal:

AX4 Interpolation path optimal:

.....

Parameters

Interpolation path optimal	AX1	AX2	AX4	
Kv factor	4	10	3	1000/min
Proportional gain Kp	0.3821	1.149	0.3615	Nms/rad
Kp/inertia	628.3	628.3	314.2	1/s
Integral time Tn	0.01	0.003575	0.02	s
Reference model active	True	True	True	
Reference model frequency	144.1	143.1	77.72	Hz
DSC active	True	True	True	
Feed forward type	Torque	Torque	Torque	
Speed ctrl EQ time for speed FFW	0.001671	0.001667	0.003063	s
Speed ctrl EQ time MD32810	0.002796	0.002792	0.004188	s
Curr ctrl EQ time for Torque FFW	0.000125	0.000125	0.000125	s
Curr ctrl EQ time MD32800	0.001125	0.001125	0.001125	s
Speed delay time (P1429)	0	0	0	s
Interpolation speed pre-control	1	1	1	
Interpolation supplementary torque	0	0	0	

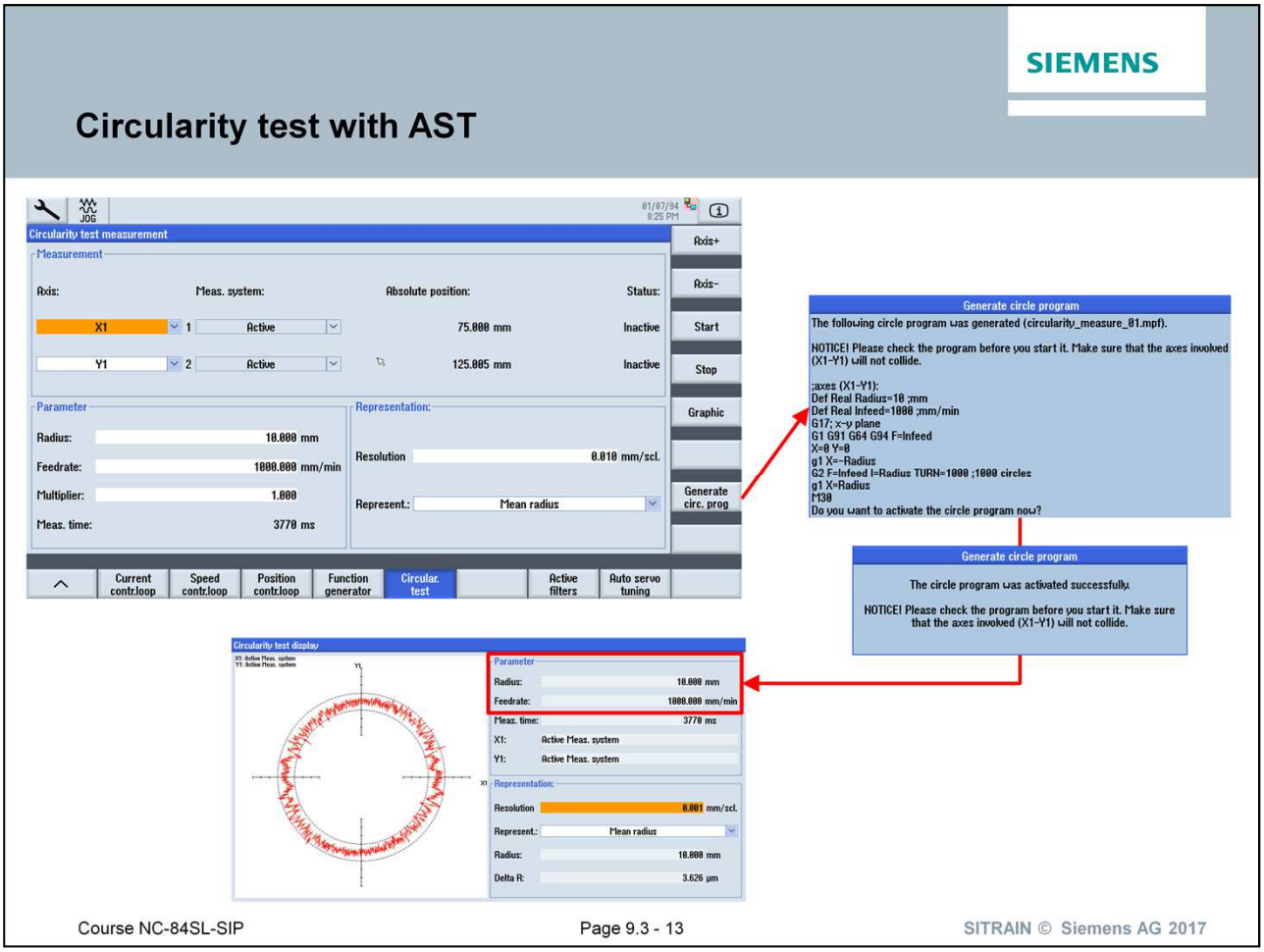
Axis optimal	AX1	AX2	AX4	
Kv factor	4	10	3	1000/min
Proportional gain Kp	0.3821	1.149	0.3615	Nms/rad
Kp/inertia	628.3	628.3	314.2	1/s
Integral time Tn	0.01	0.003575	0.02	s
Reference model active	True	True	True	
Reference model frequency	144.1	143.1	77.72	Hz
DSC active	True	True	True	
Feed forward type	Torque	Torque	Torque	
Speed ctrl EQ time for speed FFW	0.001671	0.001667	0.003063	s
Speed ctrl EQ time MD32810	0.002796	0.002792	0.004188	s
Curr ctrl EQ time for Torque FFW	0.000125	0.000125	0.000125	s
Curr ctrl EQ time MD32800	0.001125	0.001125	0.001125	s
Speed delay time (P1429)	0	0	0	s
Interpolation speed pre-control	1	1	1	
Interpolation supplementary torque	0	0	0	

AX1 Interpolation path optimal:

		Frequency	Damping	Num freq	Num damp	Res	EN
SS	PT1	1e+200					
SS	PT1	1e+200					

AX2 Interpolation path optimal:

		Frequency	Damping	Num freq	Num damp	Res	EN
SS	PT1	1e+200					



Circularity test

The circularity test function is used to check the speed controller and position controller optimization and for error analysis.
A circle is programmed and traversed by the axes; the encoder values of the axis are displayed.

Generate circle program

With this softkey, a circle program is generated into which only the radius and feedrate to be used for measurement have to be entered.
These values are automatically taken over into the parameters of the measurement function.

Selection

OA "Setup",
softkey "Optimiz./ Test", and
softkey "Circular. test"

Mean radius display

The "Mean radius" display is used to check the optimization of the speed controller.

Programmed radius display

The programmed radius display is used to check the optimization of the position controller and feedforward control.

Display

To switch between parameterization and display, press
-> softkey: Back

Optimization

Direct access to the machine data of the drive and axis

Save

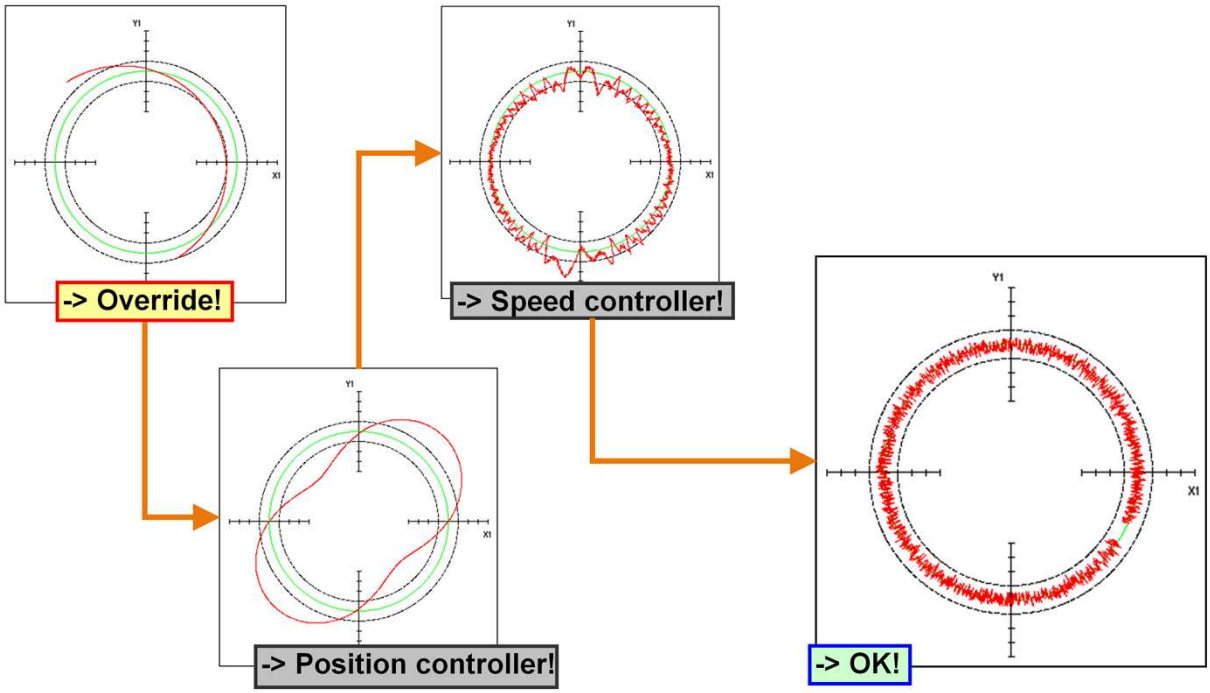
This function saves the measurement data on which the graphic is based.

Load

This function restores the graphic from the saved data.

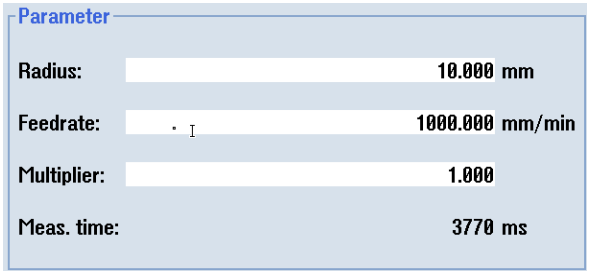
Circularity test with AST
Analysis of optimization 1

SIEMENS



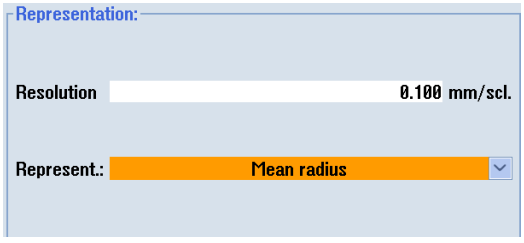
Course NC-84SL-SIP Page 9.3 - 14 SITRAIN © Siemens AG 2017

Parameters



Tests

Only measure with coarse resolution:

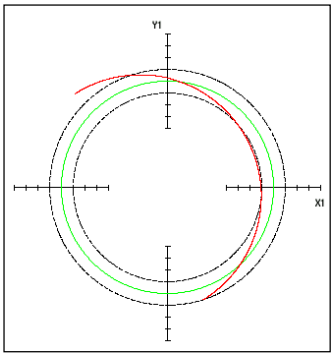


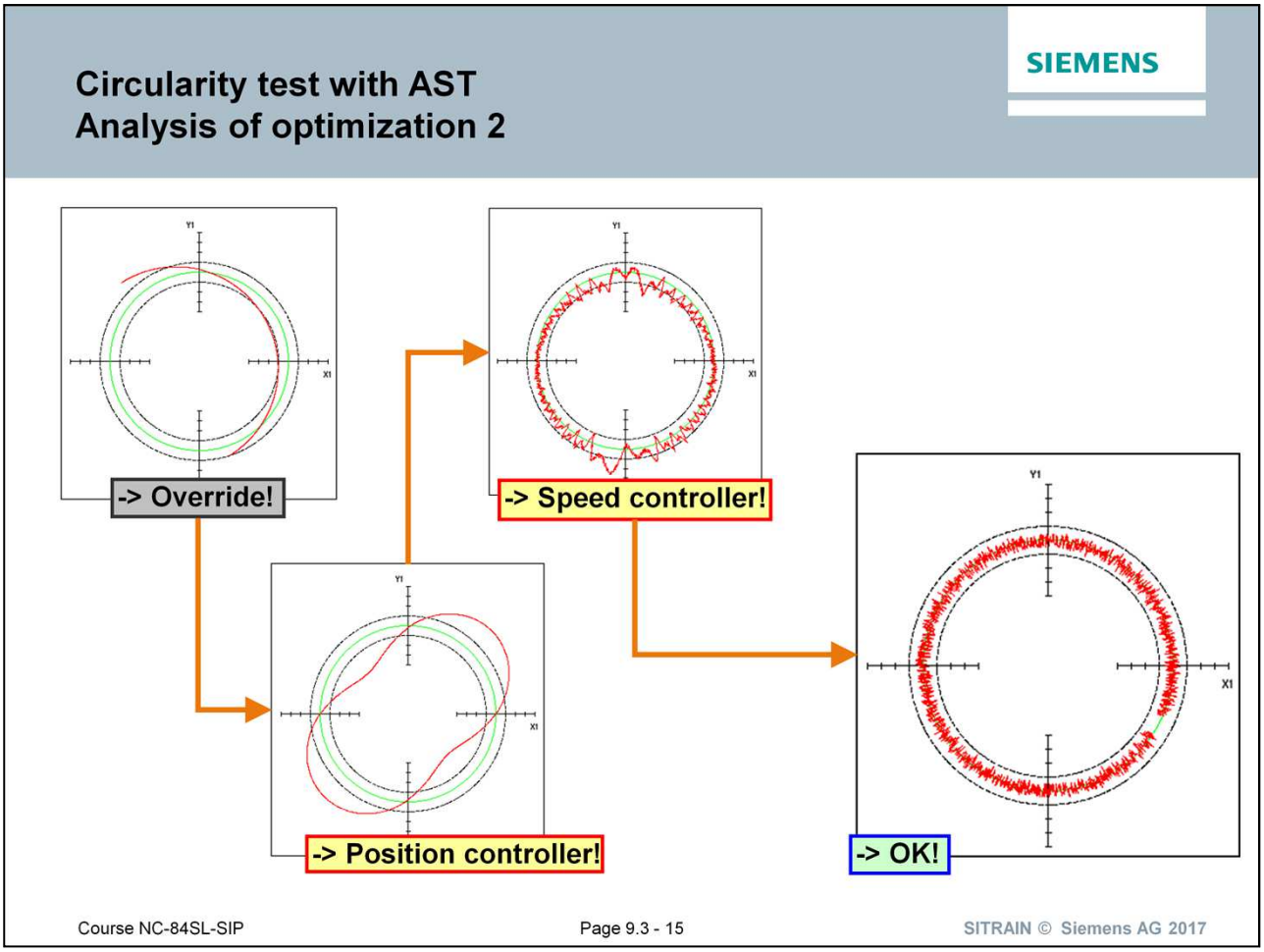
Override, feedrate

Not a full circle?

-> check override = 100 %

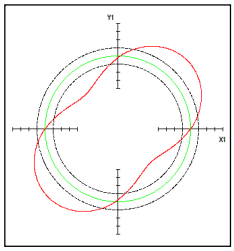
-> check feedrate





Position controller (servo gain and acceleration)

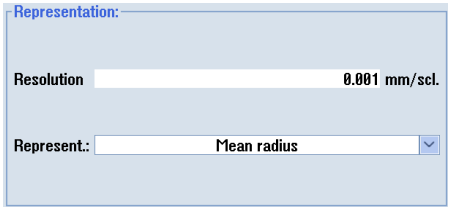
• Ellipse or similar (global deviation from circular shape)



-> Check position controller: **Servo gain**, acceleration, **jerk**, **feedforward control**

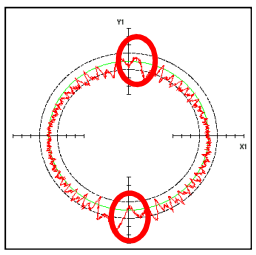
Interpolating axes must have the same settings!

High resolution Now measure with high resolution:



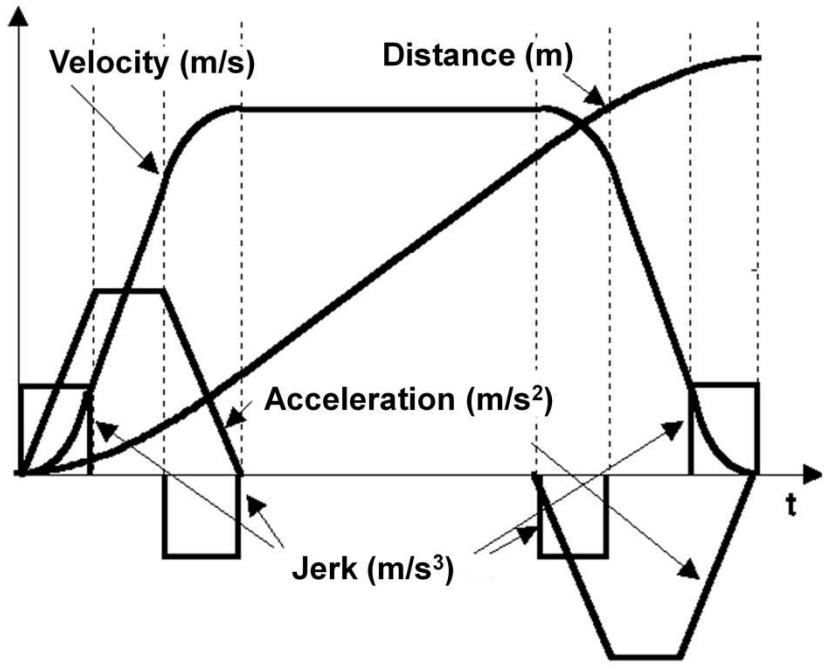
Speed controller optimization

• Local deviations from circular shape



-> Check e.g. speed controller optimization in Y!

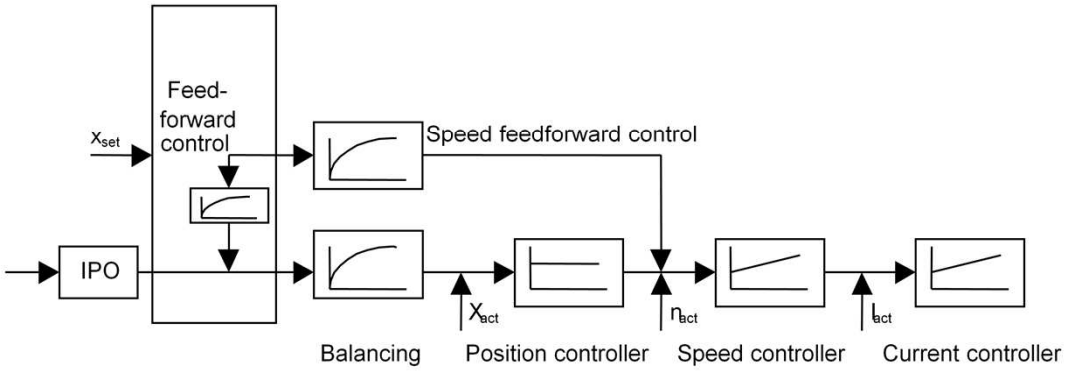
Jerk limitation



Course NC-84SL-SIP Page 9.3 - 16 SITRAIN © Siemens AG 2017

Velocity	The change of path over time.
Acceleration	The change of velocity over time.
Jerk	The change of acceleration over time.
Jerk limitation	Is activated via SOFT and deactivated via BRISK.
Path jerk	The path-specific jerk limitation is set in MD 20600 MAX_PATH_JERK. However, the oscillations caused by the jerk are mainly axis-specific; therefore, it is best to set this MD to a high value (1,000,000 m/s³)
Axis jerk	Axis-specific jerk limitation is set in MD 32431 MAX_AX_JERK.

Feedforward control
Overview



SITRAIN © Siemens AG 2017

The following feedforward control types are available for "Following error compensation".

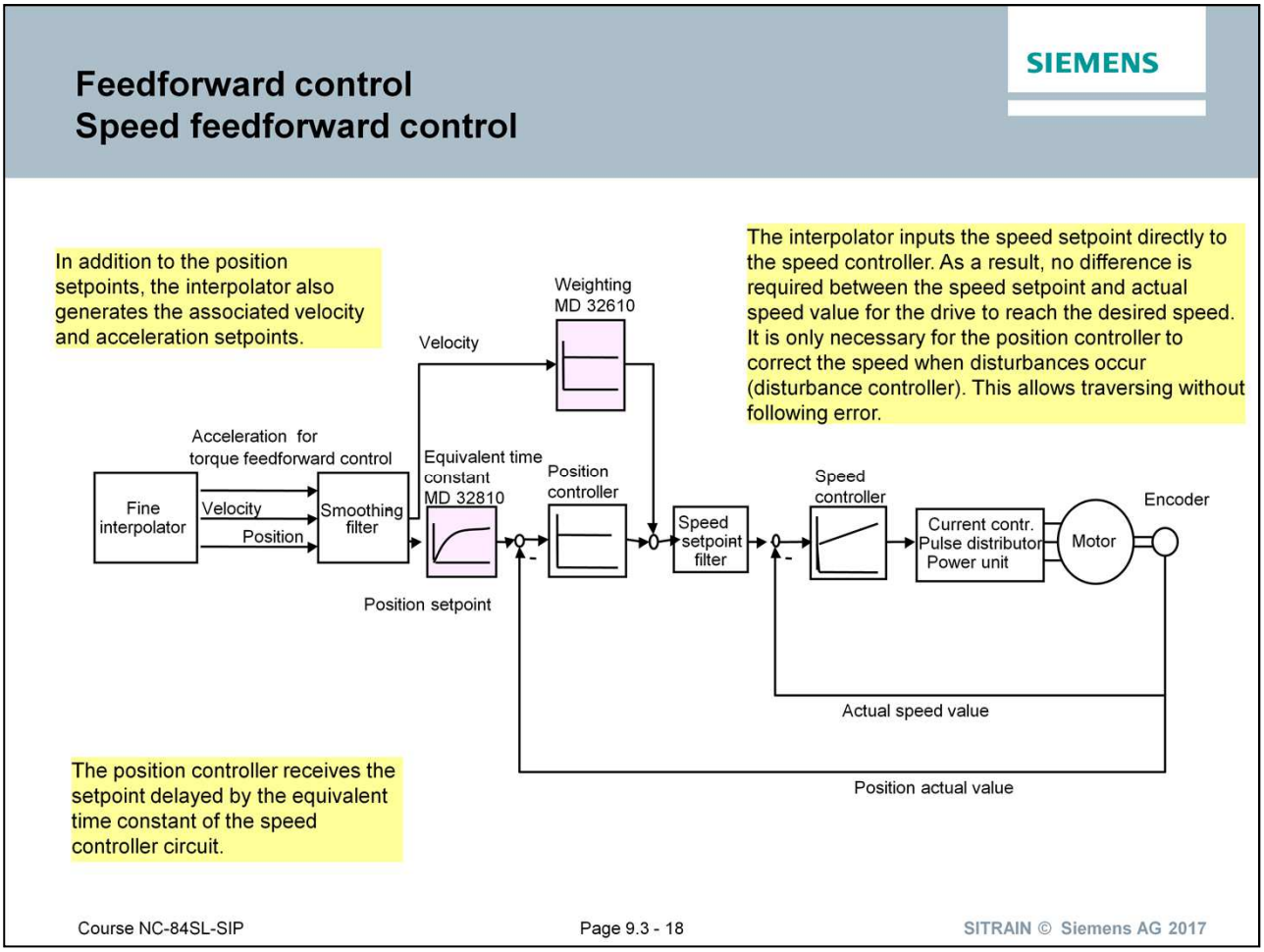
Switching ON/OFF

Feedforward control can also be switched on and off from within a part program with the following high-level language elements:

FFWON	feedforward control ON
FFWOF	feedforward control OFF

Note

To prevent jerk, feedforward control should only be switched on or off while the axis/spindle is stationary. The programmer must take this into account and ensure compliance.



Feedforward control mode MD 32620 FFW_MODE

0: feedforward control is deactivated
3: **Velocity feedforward control**
4: Torque feedforward control (840D sl SW 4.4 and higher)

Activation mode MD 32630 FFW_ACTIVATION_ MODE = 1
Feedforward control for all axes/spindles of the channel can be activated/deactivated from the part program with FFWON/FFWOF. For axes that are interpolating together, MD 32630 FFW_ACTIVATION_MODE should, therefore, be set to identical values.

FIPO type MD 33000 FIPO_TYPE = 3
(recommended for the new values 3 and 4 of MD FFW_MODE)

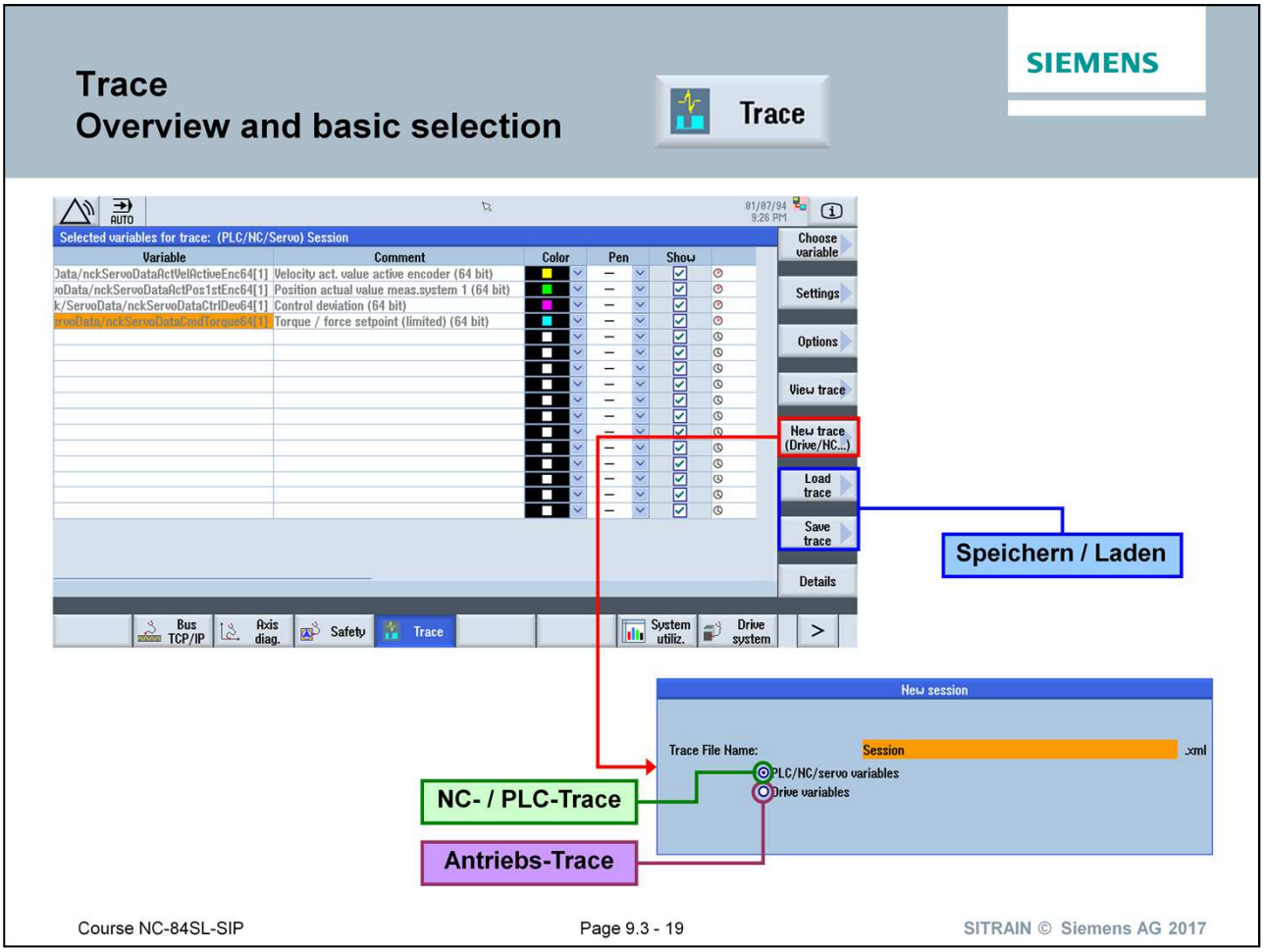
Equivalent time constant for the speed control loop

MD 32810 EQUIV_SPEEDCTRL_TIME
Equivalent time constant of the speed control loop for velocity feedforward control (FFW_MODE=3)
The time is set correctly when it is possible to reach the position without overshooting.

Weighting MD 32610 VELO_FFW_WEIGHT defines the weighting factor for feedforward control. It must not be set to any value other than 1.

Activation mode default

MD 20150 GCODE_RESET_VALUES[23] =1 (FFWOF)
Definition of the standard setting of the G group. In machine data MD 20152 GCODE_RESET_MODE and MD 20112 START_MODE_MASK, you define the conditions in which this default value is applied.



Access In the Diagnostics operating area **>** Trace

Trace: Function overview

With Trace, you are provided with an oscilloscope function.
NC, PLC, and drive signals can be selected.

Start The trigger conditions for the start of recording are:

- No trigger
- Triggering with freely selectable variable

Stop Recording can be stopped with a softkey, when the time expires, or if the memory limit is reached.

Cycle time Both cyclic events (e.g. in the IPO cycle/position control cycle/PLC cycle) and non-cyclic events can be recorded.

Graphical display and analysis

The recorded data is optically displayed in the form of curves, the so-called characteristics graphs.
To analyze the result, you can use the functions:

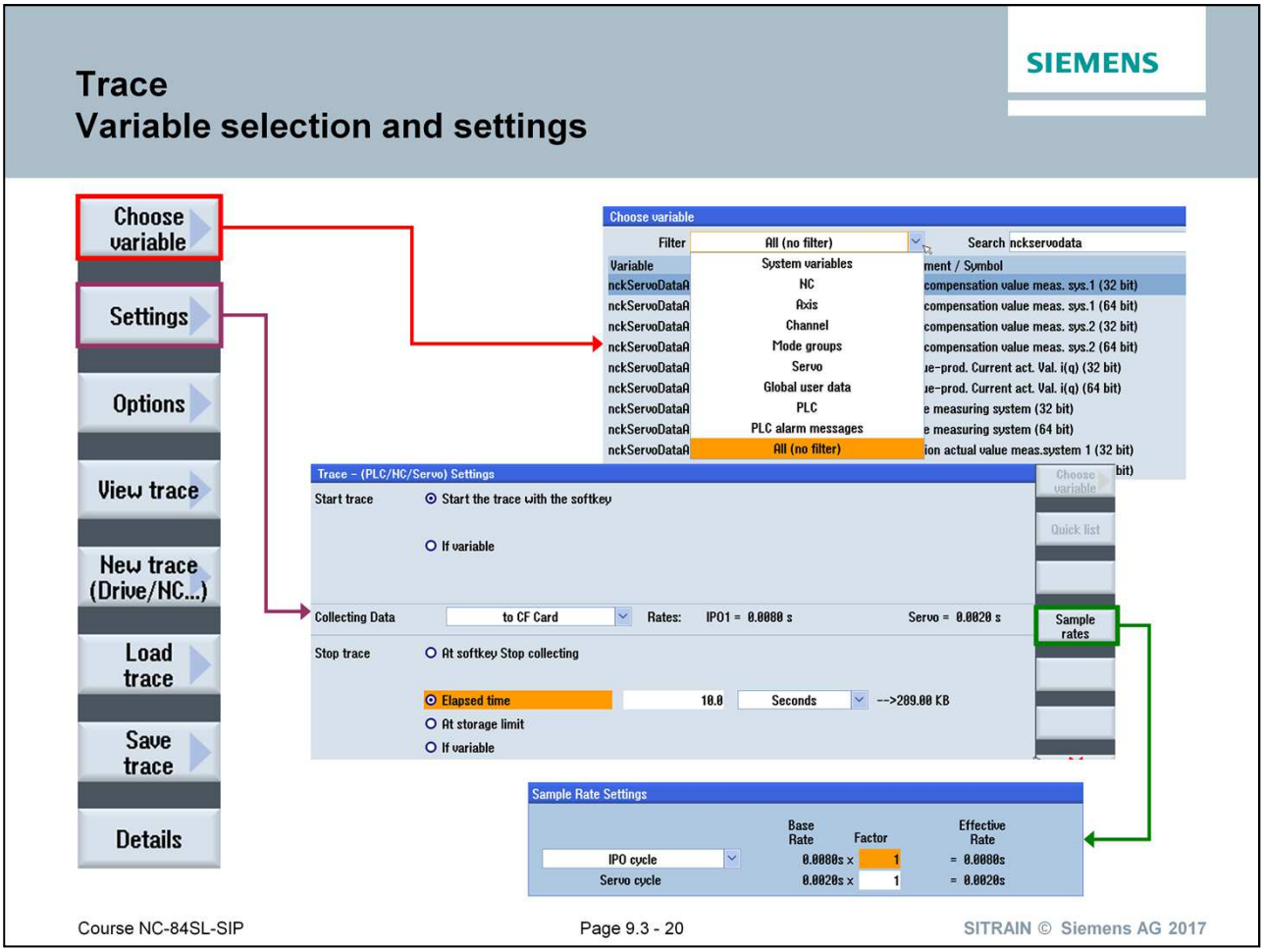
- Scale / zoom traces
- Define a traced range using markers.

Save and download data

- You can save the recorded data in an XML with **softkey: Save trace**.
- You can reload the saved data for evaluation at a later point with **softkey: Load trace**.

Basic selection Signal selection of PLC and NC signals (NC/PLC trace) or SINAMICS signals (drive trace).

This is done with **-> softkey: New trace (Drive/NC)**



Choose variable

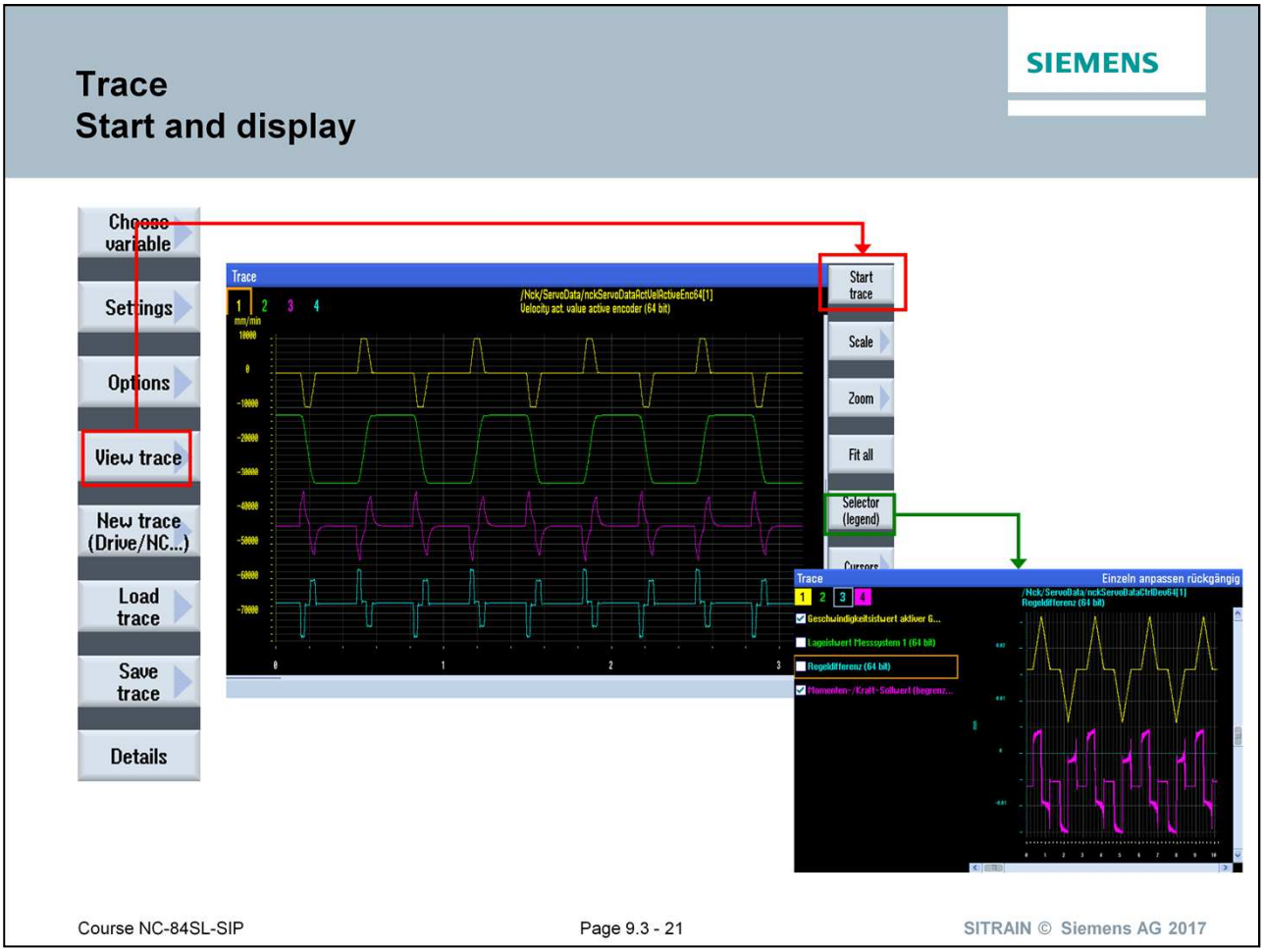
You can select
the variable with
softkey: Choose variable.

Filter, search

The selection is supported by filters (e.g. servo data) and a search function.

Settings

The start and stop conditions (triggers) are selected in menu
softkey: Settings
By default, the CF card (Operate on NCU) or the hard disk (Operate on PCU) is used as the storage medium.
In this menu, the recording duration can be increased by selecting a factor > 1 for the recording cycle with
softkey: Sampling rates



Start / display

The following softkeys are used to start and display the trace
Softkey: View trace
and
Softkey: Start trace.

Scaling

The simplest form of scaling is achieved with
softkey: Fit all

Part selection

For part selection of the curves displayed and to display the legend, select
softkey: Selector (legend)

Scale

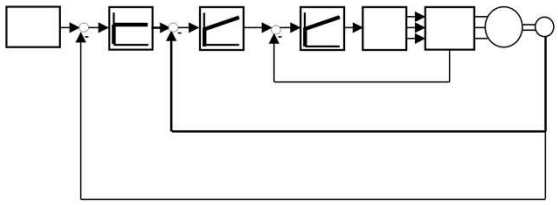
Other changes to the scaling can be made manually with
softkey: Scale

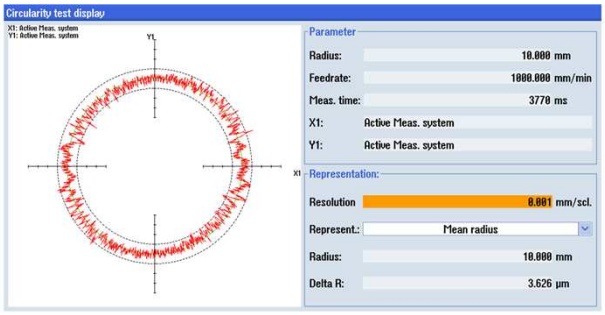
Zoom

This function enables the display of subareas to be enlarged in conjunction with markers. The function is selected with
softkey: Zoom

Exercises for SINUMERIK 840D sl commissioning







SITRAIN © Siemens AG 2017

Notes

Exercise: Conventional optimization in time range

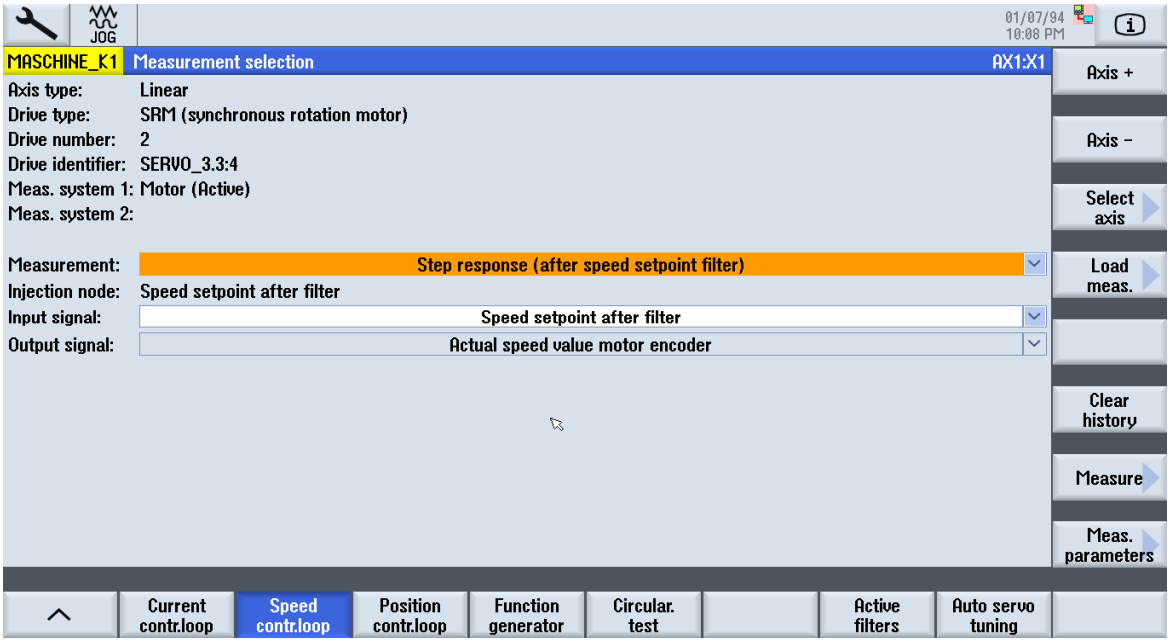
Requirement: The current controller is already preset correctly by having selected the motor type.

For measuring in the speed control loop, select

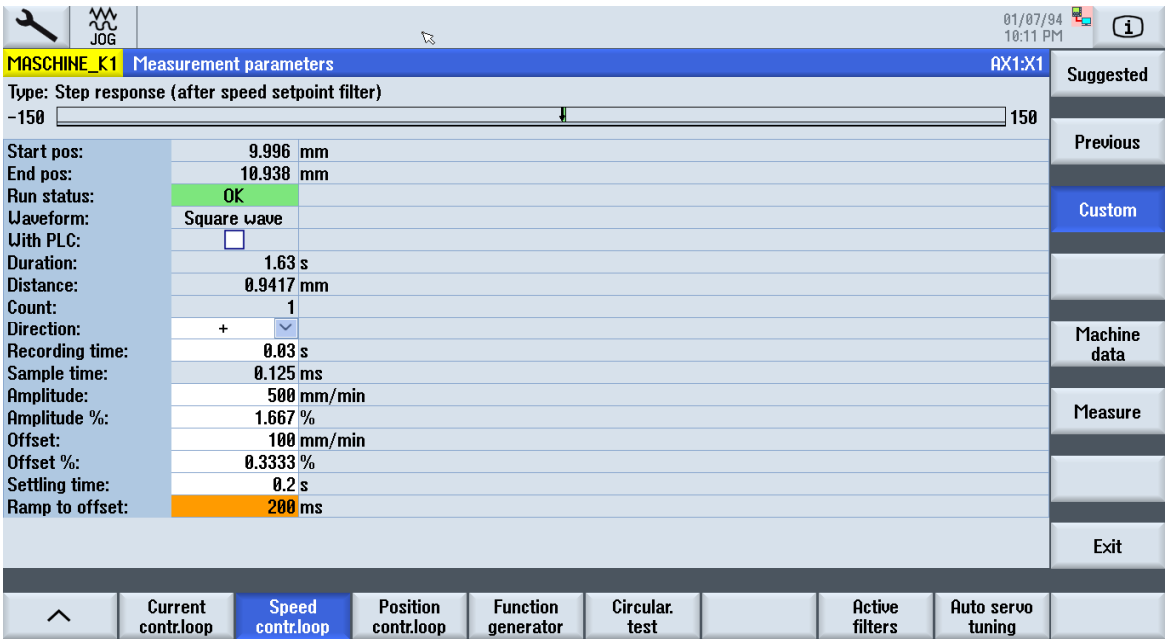
→ **Softkey "Setup", → Softkey "Optimiz./Test",**

→ **Softkey "Speed contr.loop"**

Select the parameters as shown in the image.



Enter the required values via softkey "Measuring parameters":

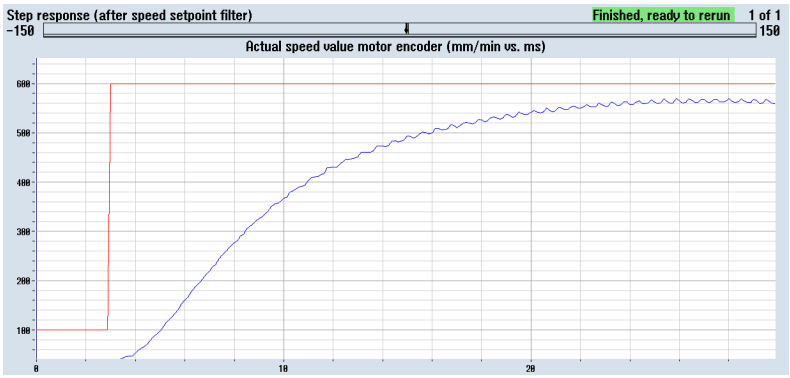


For the first measurement, change the integral time TN (MD 1462) to 0 (softkey "Drive MD"), which switches the I controller off.

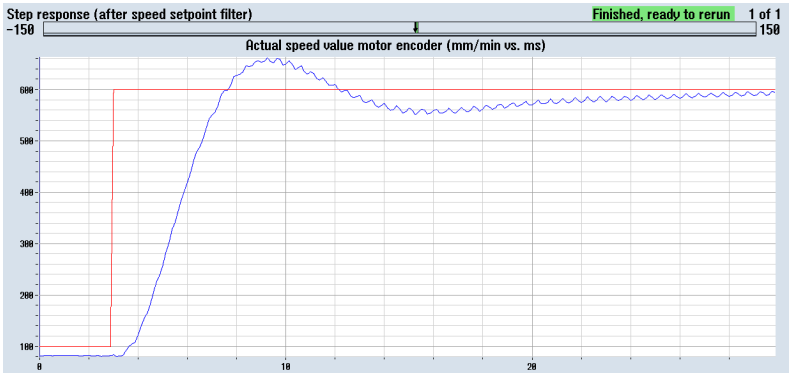
Measurement is started with
→ **softkey Measurement → softkey Start meas.**
→ **NC START**
Now change the MD 1460 (proportional gain) so that the **steepest possible rise of the step response results without any overshooting**.

Some examples:

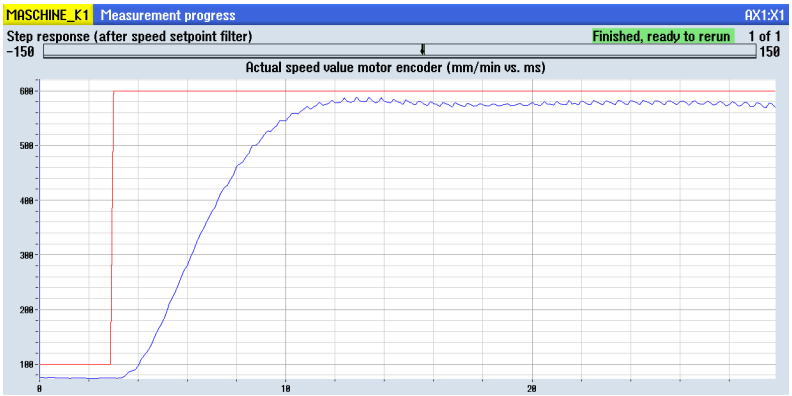
MD1462 TN = 0
MD1460 Kp too small:



MD1462 TN = 0
MD1460 Kp too large:

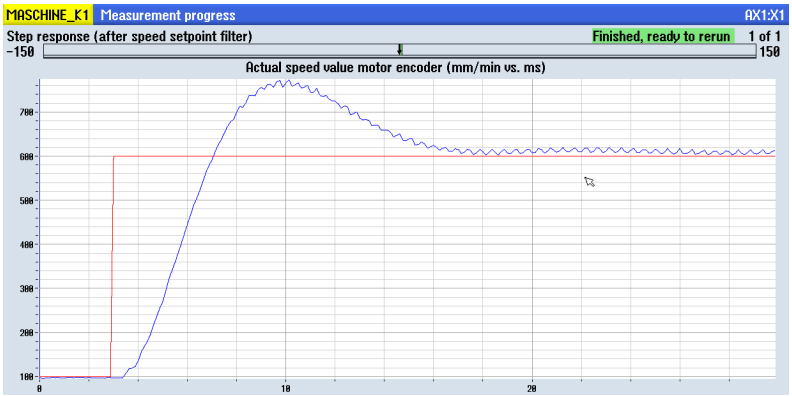


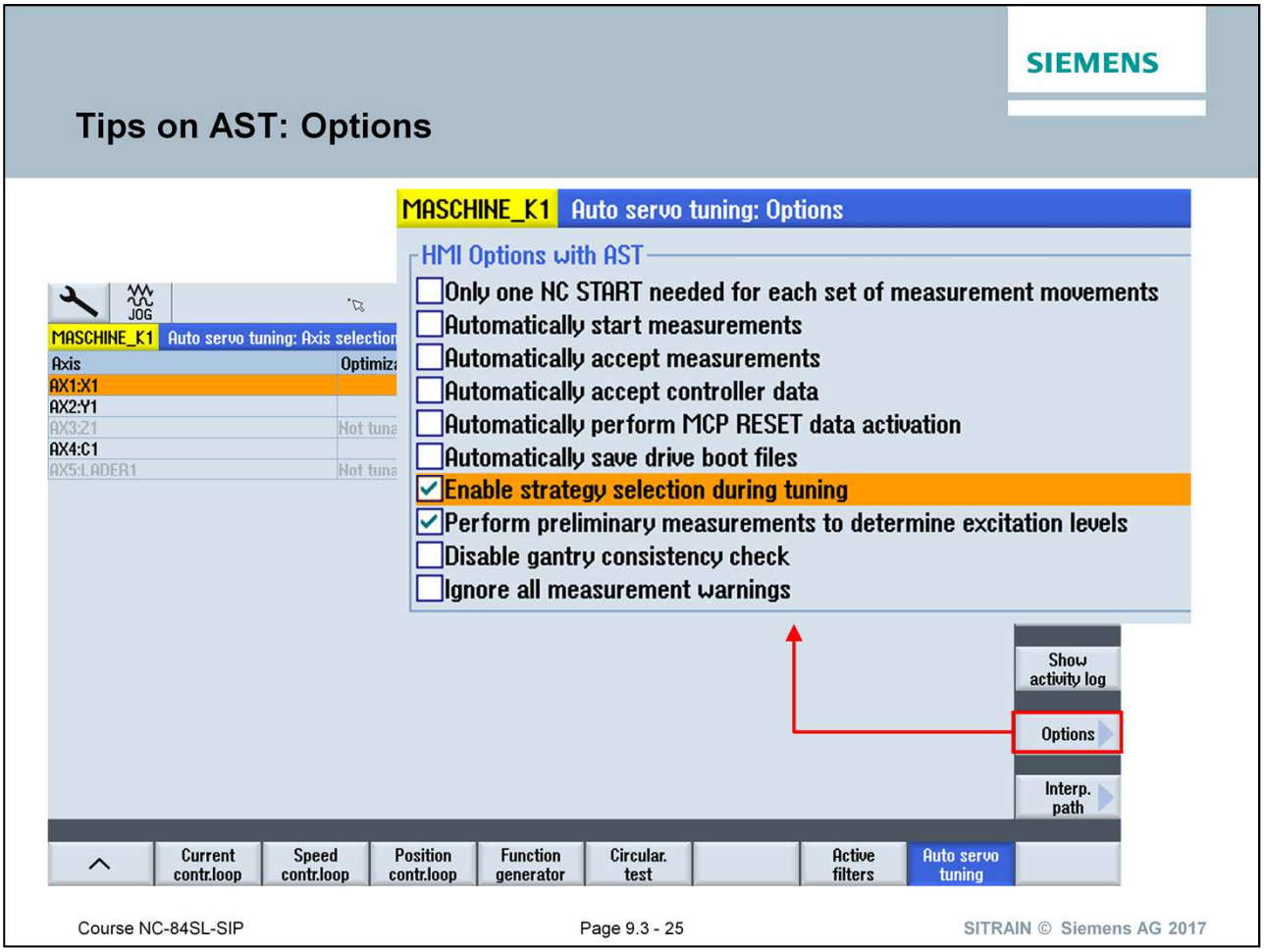
MD1462 TN = 0
MD1460 Kp ok:



MD1462 TN ok
MD1460 Kp ok:

Now Tn is reduced until the overshoot is 20-40% of the amplitude setpoint.

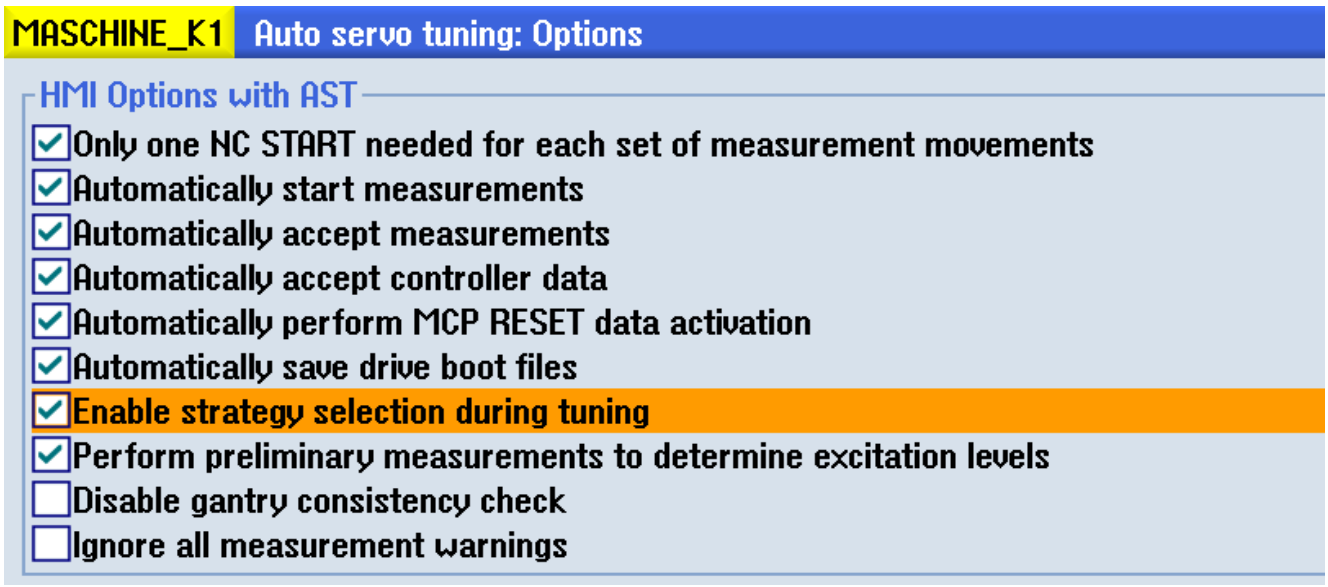


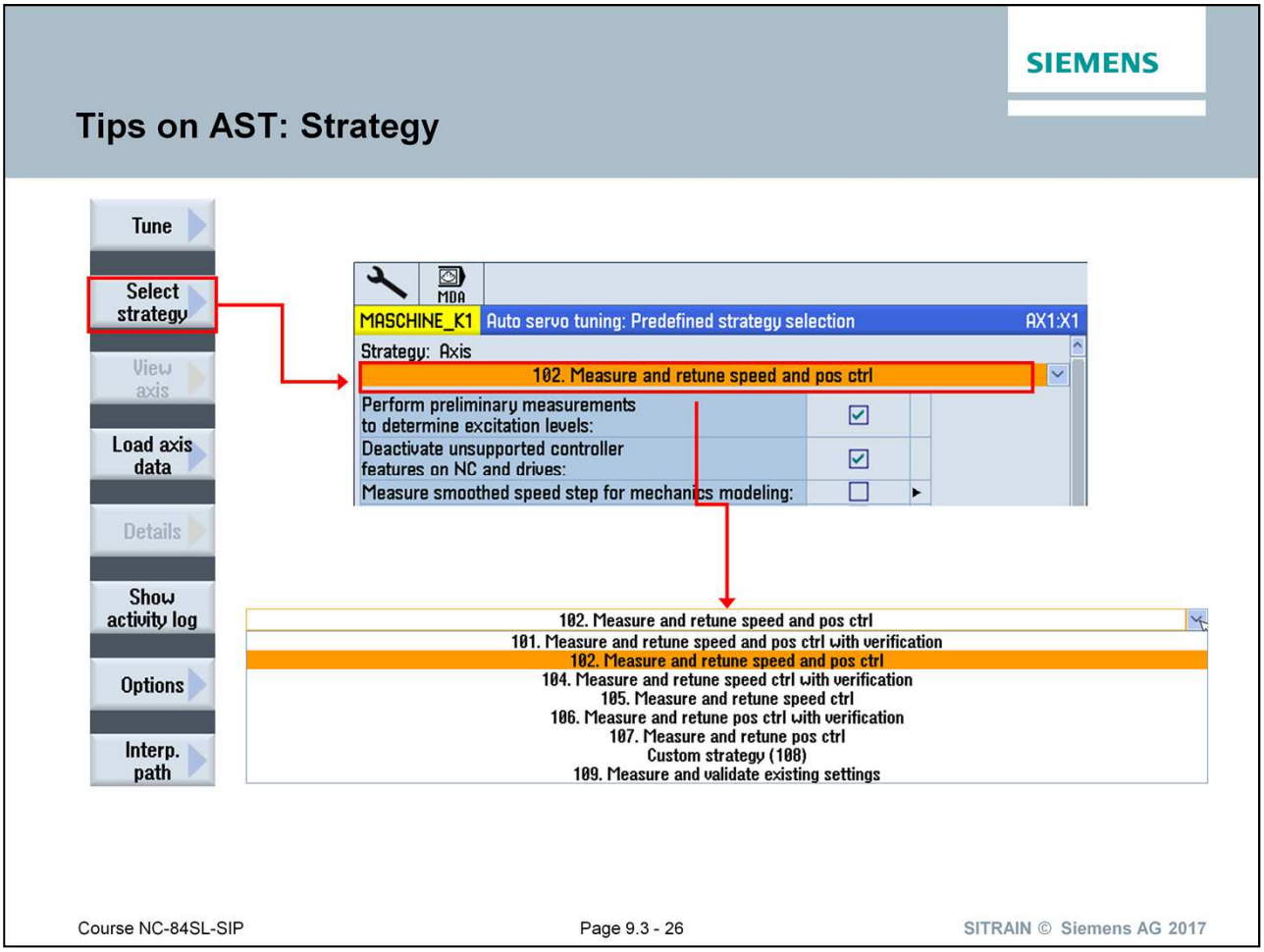


Options

To be able to see all intermediate steps if possible, we recommend the measurement option settings shown above at the start

Later you can set the options like this:



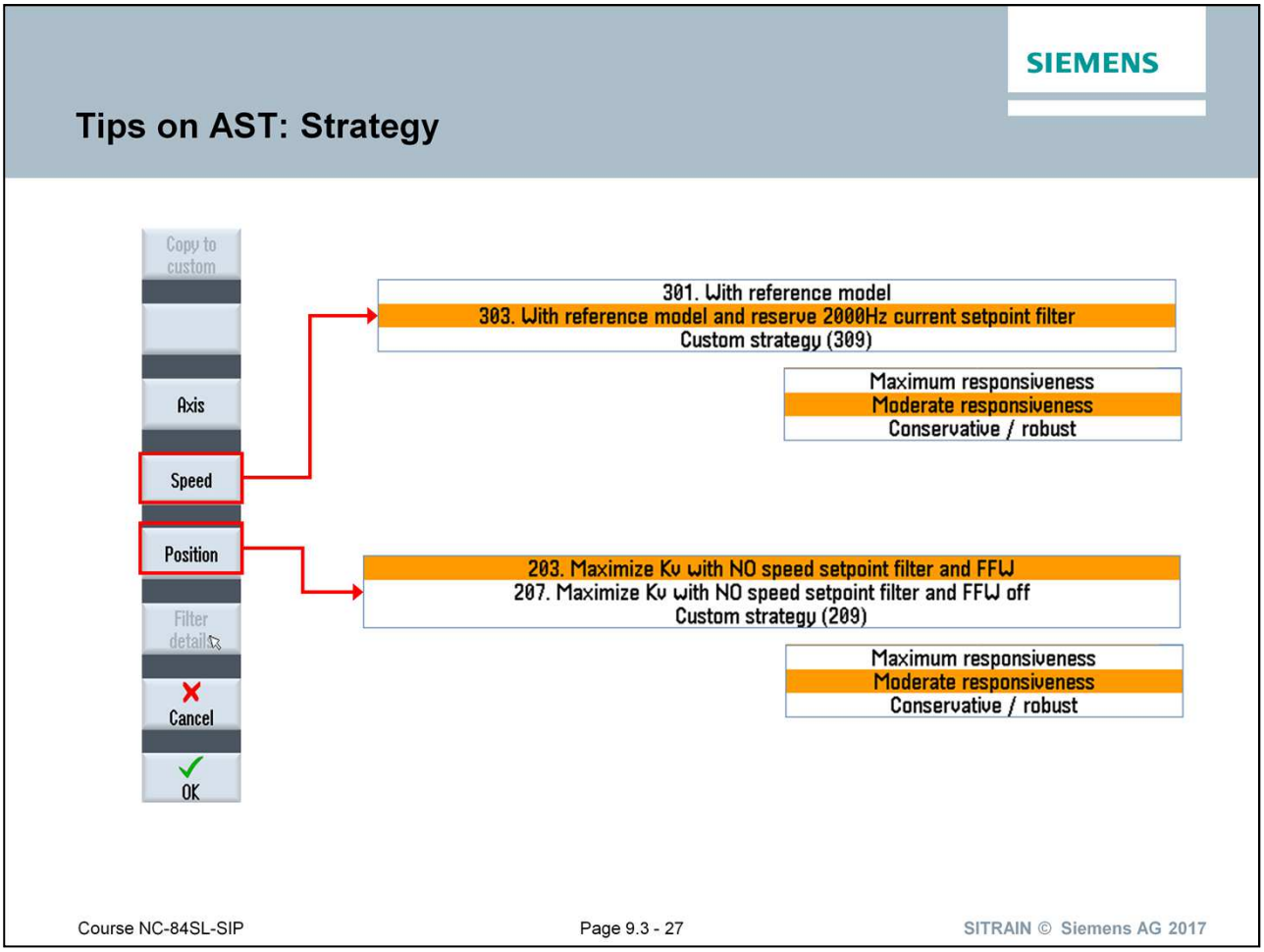


Strategy axis

Here, you can first choose whether the position controller and the speed controller or just one of the two will be retuned.
Also, whether a final measurement with display of a Bode diagram of the relevant closed control loop will be performed or not.

Optimization target

The selection made here influences both the position and the speed controller. For example, the maximum permitted servo gain factors of the position controller or the desired phase margin of the speed controller are preset.



Speed controller

→ Reference model In the optimization run, a model of the controlled system is generated by measurement.

→ 2KHz filter The current setpoint filter activated in the default data at 2000Hz can be retained.

Position controller

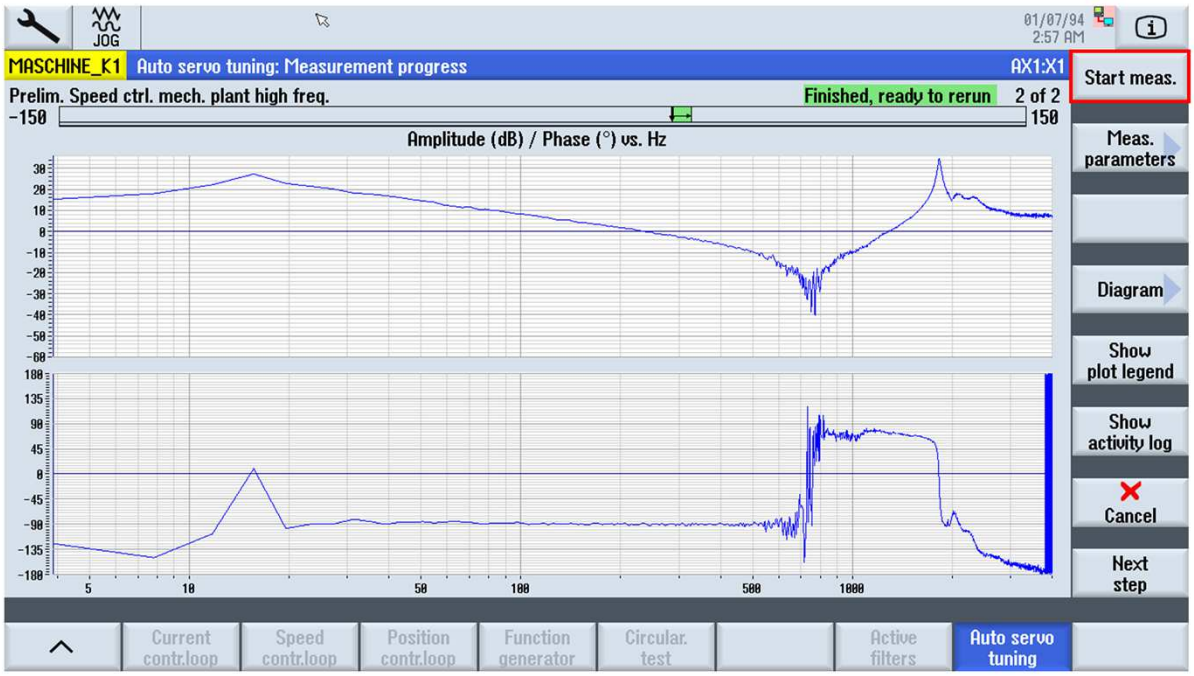
→ FFW The feedforward control (speed/current) can be activated and therefore also optimized.

Recommended setting:

The strategies **303 and 203** should be retained if possible.

Tips on AST
Performing measurements

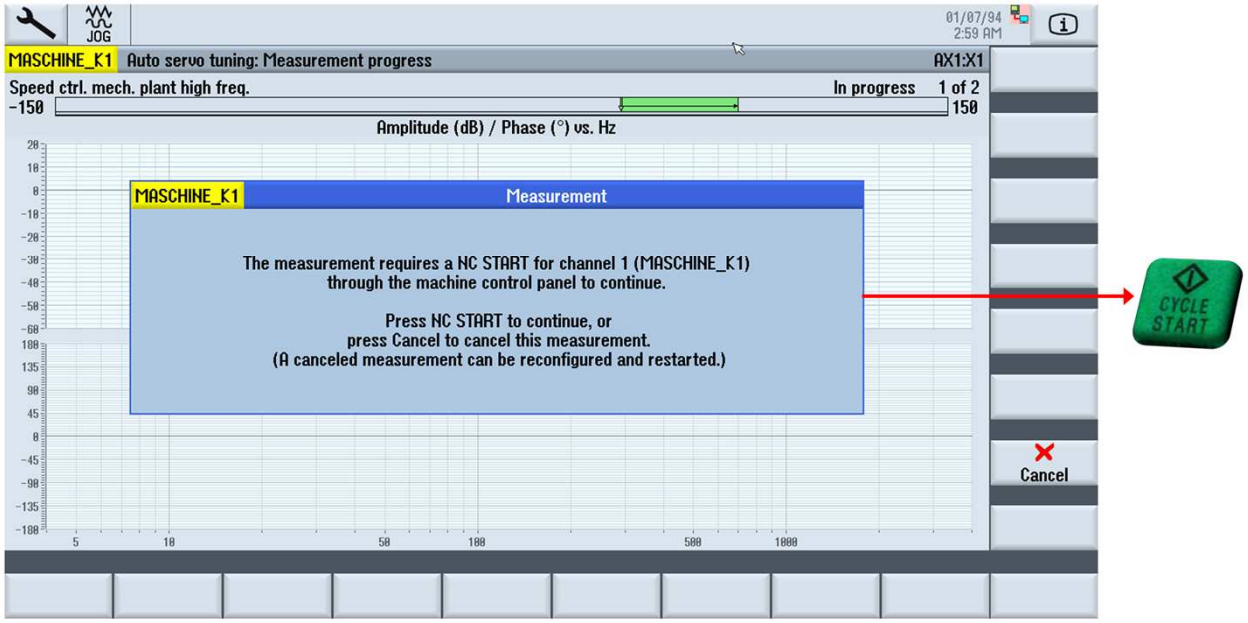
SIEMENS



Course NC-84SL-SIP Page 9.3 - 28 SITRAIN © Siemens AG 2017

Start of measurement Each measurement is initiated by pressing the **softkey: "Start meas."**
- The number of times it has to be pressed depends on the options!

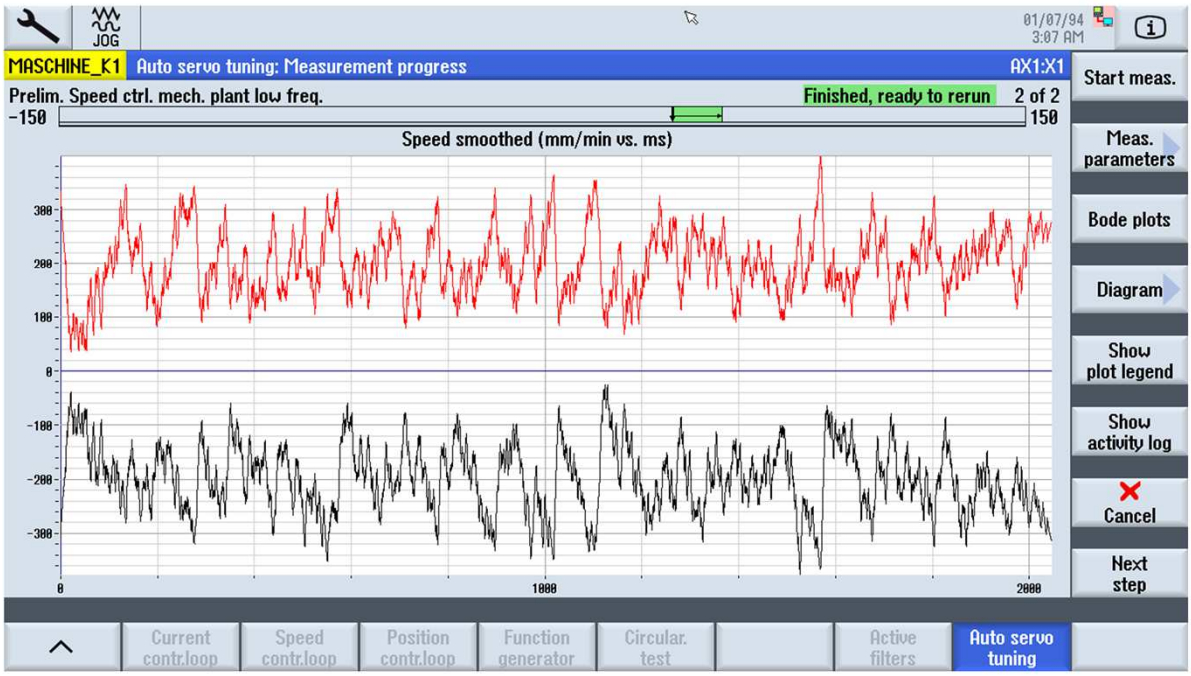
Tips on AST
Performing measurements



SITRAIN © Siemens AG 2017

Start of measurement To start measurement, press NC Start.
Pay attention to the safety notes!
The number of times the NC Start key has to be pressed depends on the options!

Tips on AST
Premeasurement



SITRAIN © Siemens AG 2017

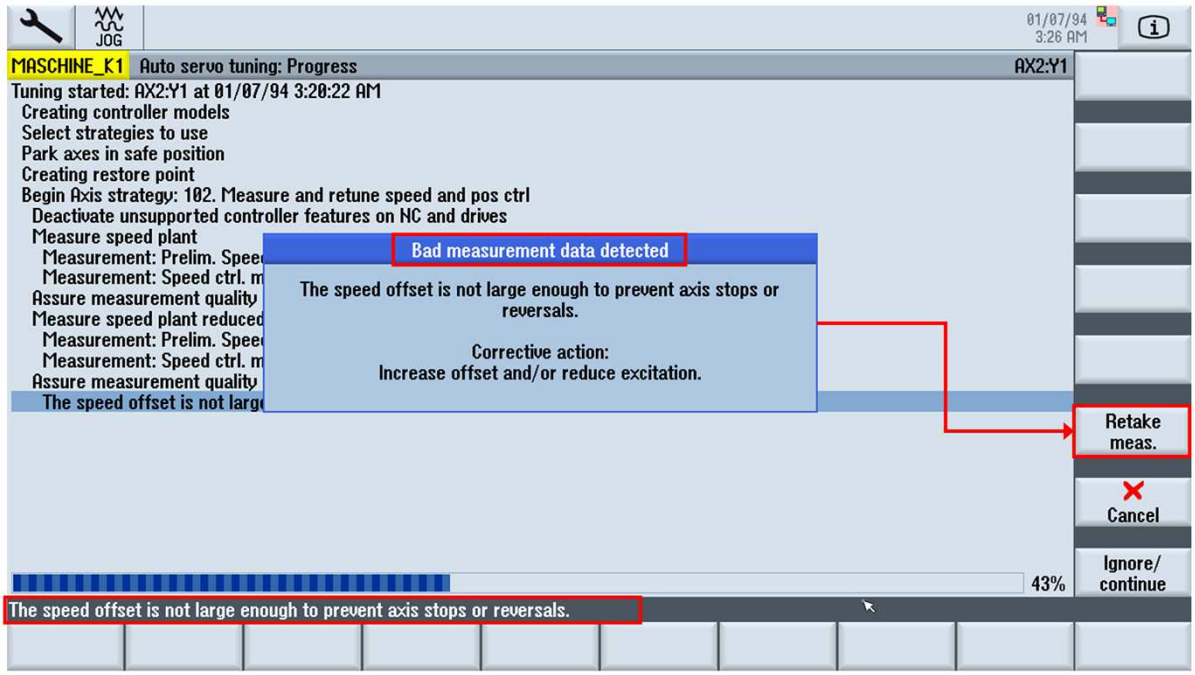
Premeasurement

Premeasurement is used to determine suitable values for amplitude and offset. The results apply:
- Signal too "noisy" -> increase amplitude
- Signal crosses zero line -> increase offset
  (axis remains at standstill during measurement)

Traversing distance

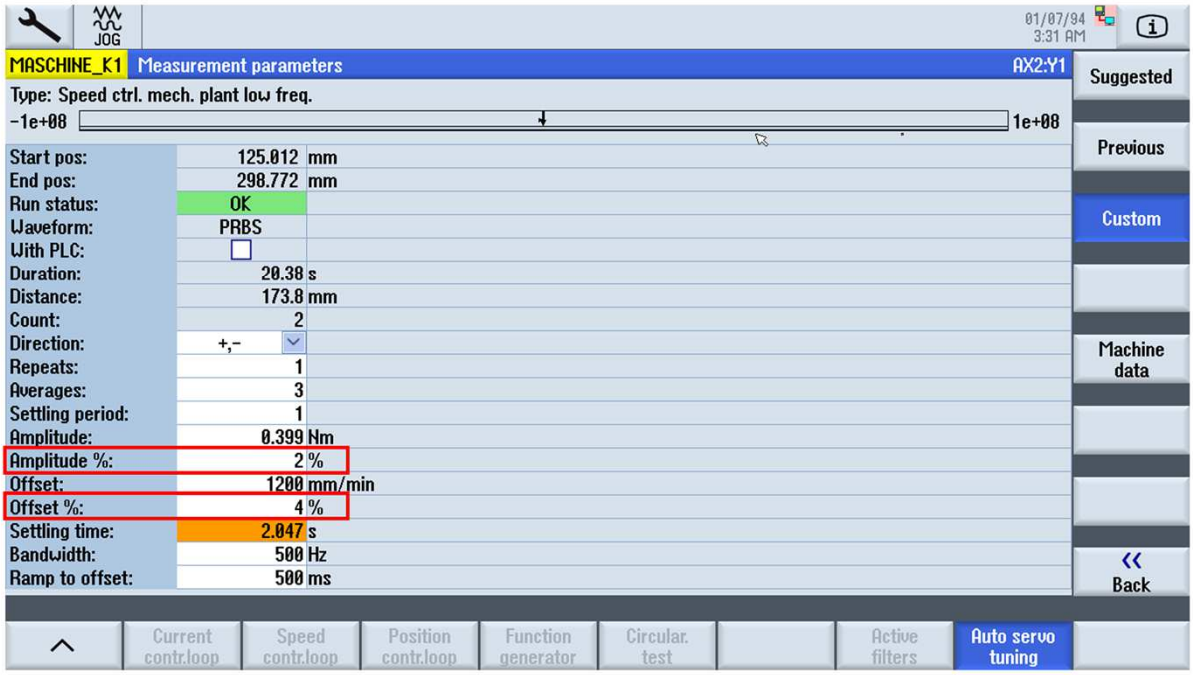
It must be considered, however:
the required traversing distance increases with the offset!

Tips on AST
Invalid measurement data



Invalid measurements Sometimes the above message appears toward the end of the measurement series.
In this case, press the softkey: "Repeat measurement"!

Tips on AST
Invalid measurement data

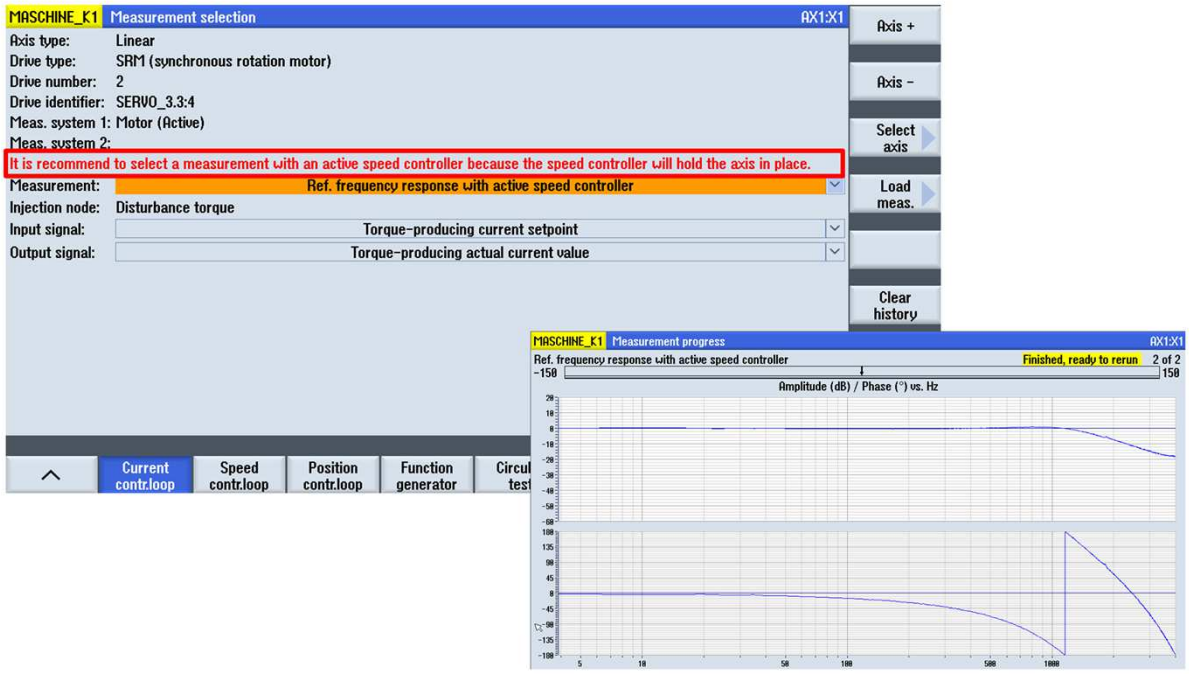


SITRAIN © Siemens AG 2017

Offset = 2 *amplitude

In the image that then follows select the amplitude and offset.
Value according to the rule of thumb offset = 2 * amplitude,
e.g.: amplitude = 2 %
offset = 4 %
Then repeat the last message with
softkey: << Back -> Start meas.

Tips on AST
Measurement check for current controller



SITRAIN © Siemens AG 2017

Measurement checks: Current controller

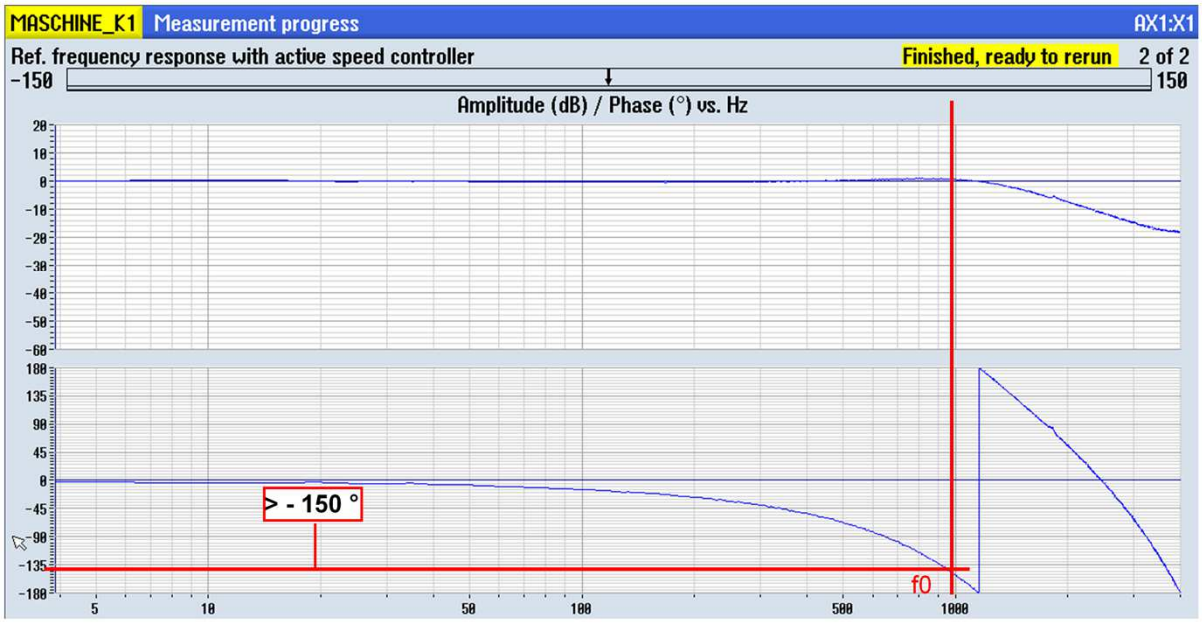
After optimization has been completed, measurement checks can be performed. You can measure the current controller with **softkey: Current controller**.

Reference frequency response with active speed controller

This type of measurement is recommended strongly, because it prevents big axis movements (vertical axes!)

Tips on AST
Measurement check for current controller

SIEMENS

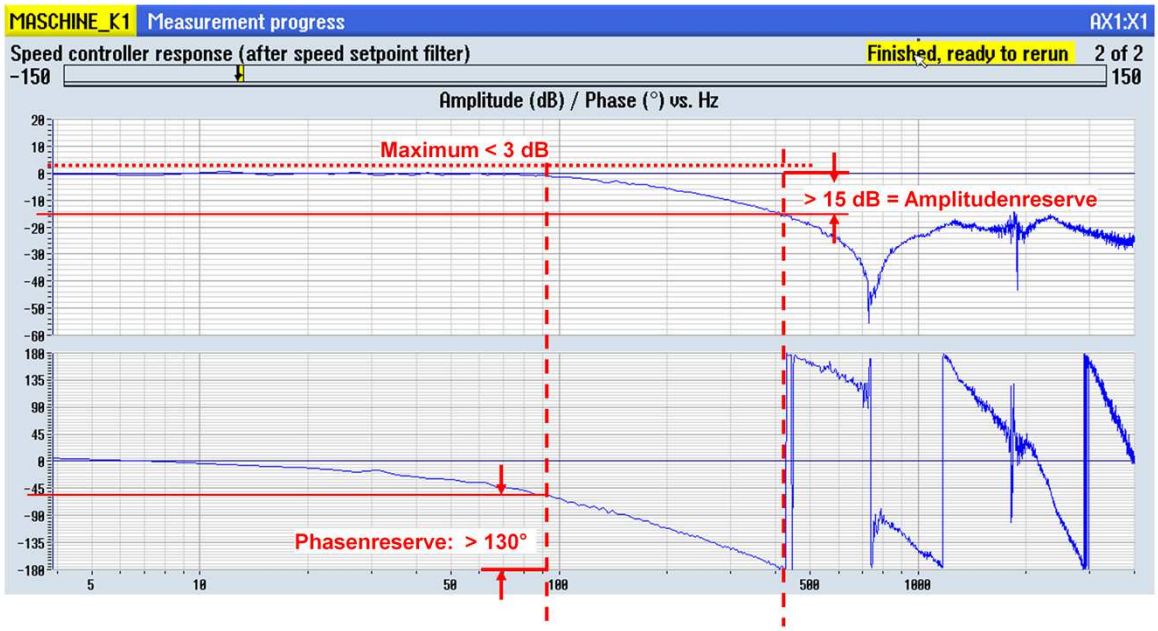


SITRAIN © Siemens AG 2017

Stability: Phase margin >= 30 °

It is a common stability criterion for the frequency response:
The phase margin is at least 30 °.
I.e.: With frequency f0, the phase is at least 30 ° from the critical case (phase = 180 °).
The frequency f0 is defined as the frequency
of the last passage through 0 dB of the amplitude (f>f0: amplitude < 0 dB)

Tips on AST
Measurement checks for speed controller

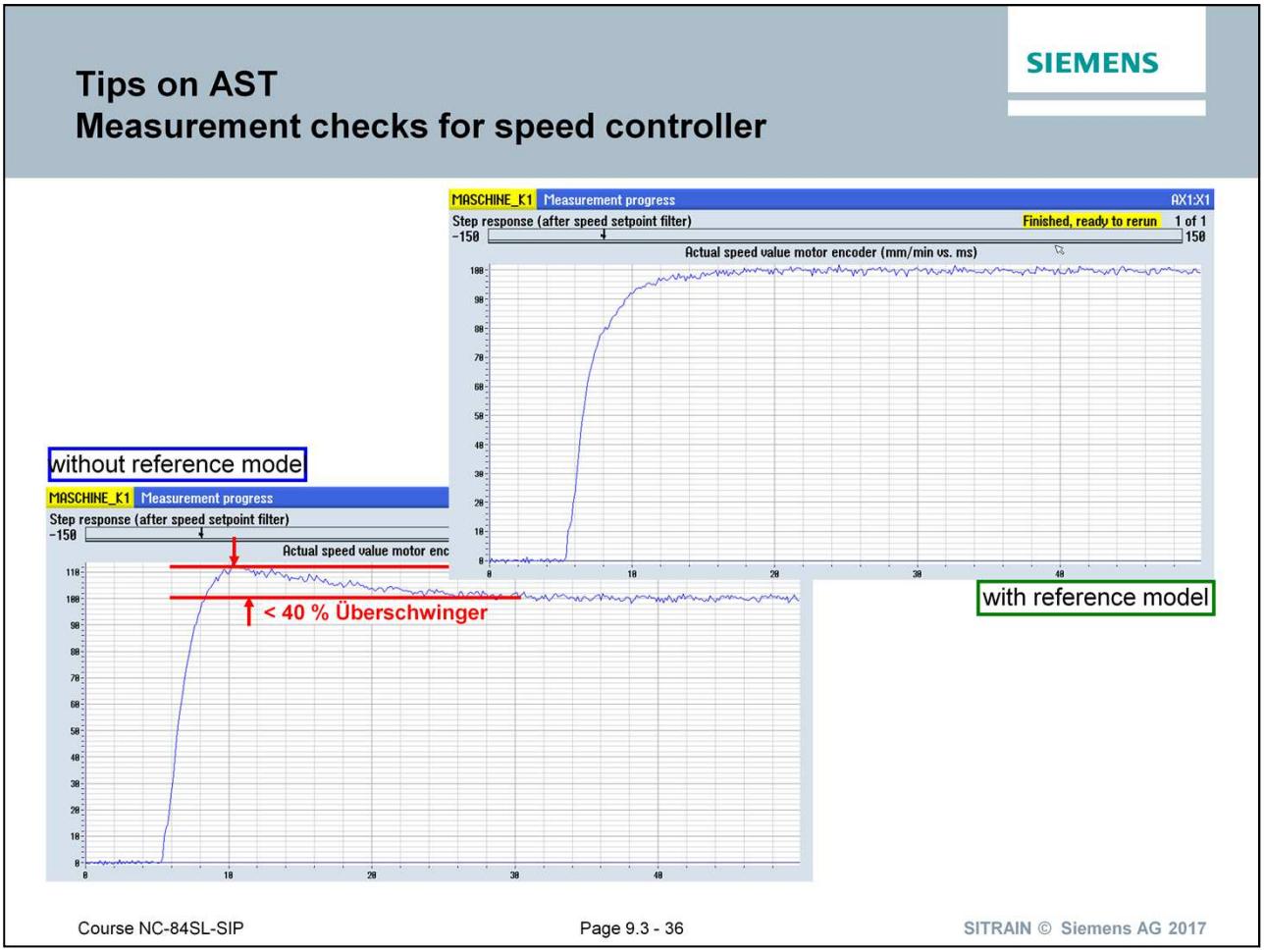


 SITRAIN © Siemens AG 2017

Measurement checks: Speed controller

You can measure the speed controller with **softkey: Speed controller**.
A corresponding frequency response measurement is shown in the above image.
The variables maximum, amplitude margin, and phase margin can be read here:

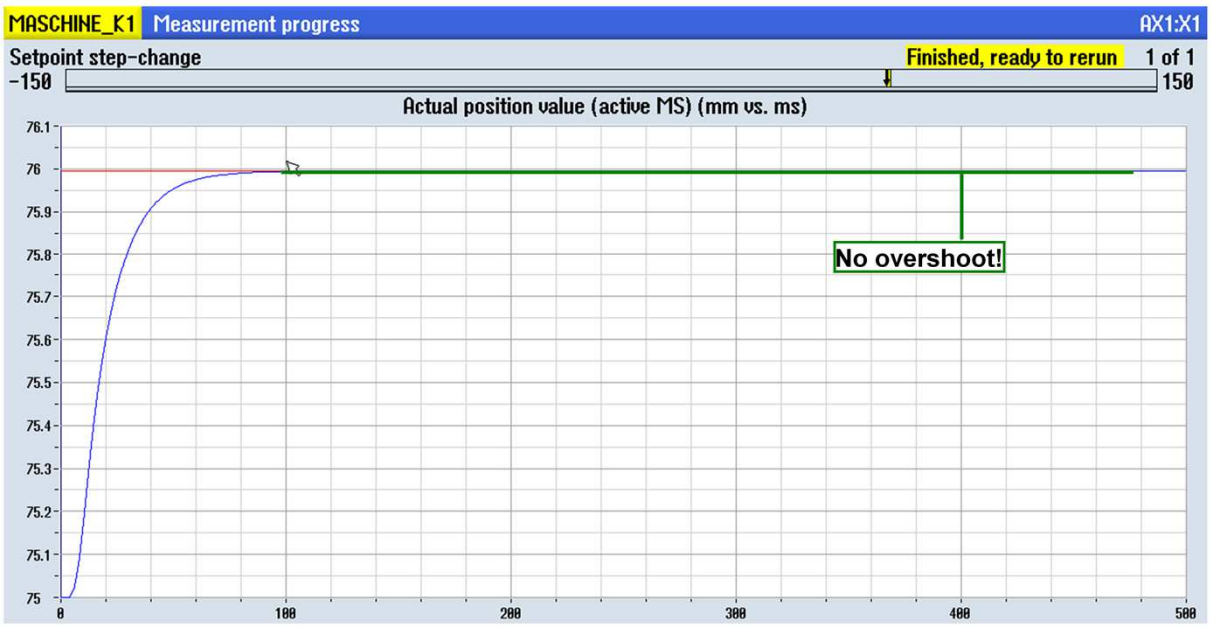
- **Maximum** That is the maximum value of the amplitude
  The following is usually required: Maximum < 3 dB.
  Because 3 dB = 10 ^ (3/20) ~ 1.4 that corresponds to an overshoot of the amplitude by 40 %.
- **Amplitude margin** That is the (negative) amplitude at the frequency, where the phase is 180 ° for the first time.
- **Phase margin** That is the distance of the phase from 180 ° at the last (highest frequency) passage through 0 dB of the amplitude.



Measurement checks: Speed controller

Here, we see measurement of the speed controller by means of a step change in the setpoint, with and without the reference model (1400 bit 3 = 1).
It can clearly be seen that using the reference model reduces the overshoot!

Tips on AST
Measurement check for position controller



SITRAIN © Siemens AG 2017

Measurement checks: Position controller

Here, you can see a measurement of the position controller by means of a step change in the setpoint.
Here, the principle is (p-controller!): no overshoot!

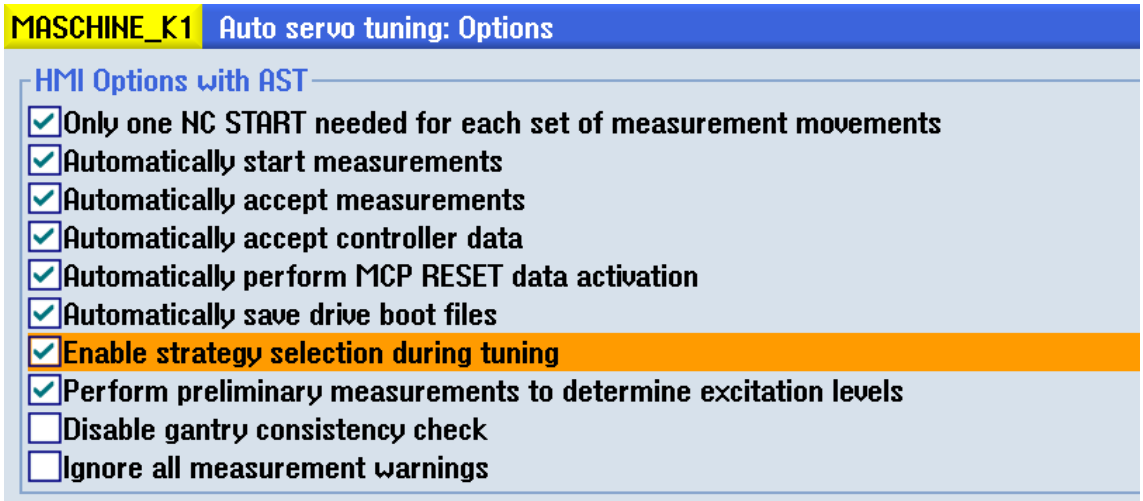
Exercise: Auto servo tuning

Before optimization: Dynamic stiffness control MD32640 should be set = 1 and a second measuring circuit should be enabled (if there is one). When a spindle is optimized automatically, there is a possibility that the PLC enable signal will be missing. The machine manufacturer should provide an option to enable the PLC interface signal for optimizing the servo. For example, a special key combination or query of a special flag in the PLC status.
For feedforward control, MD 33000 = 3 should be set.

After auto servo tuning:

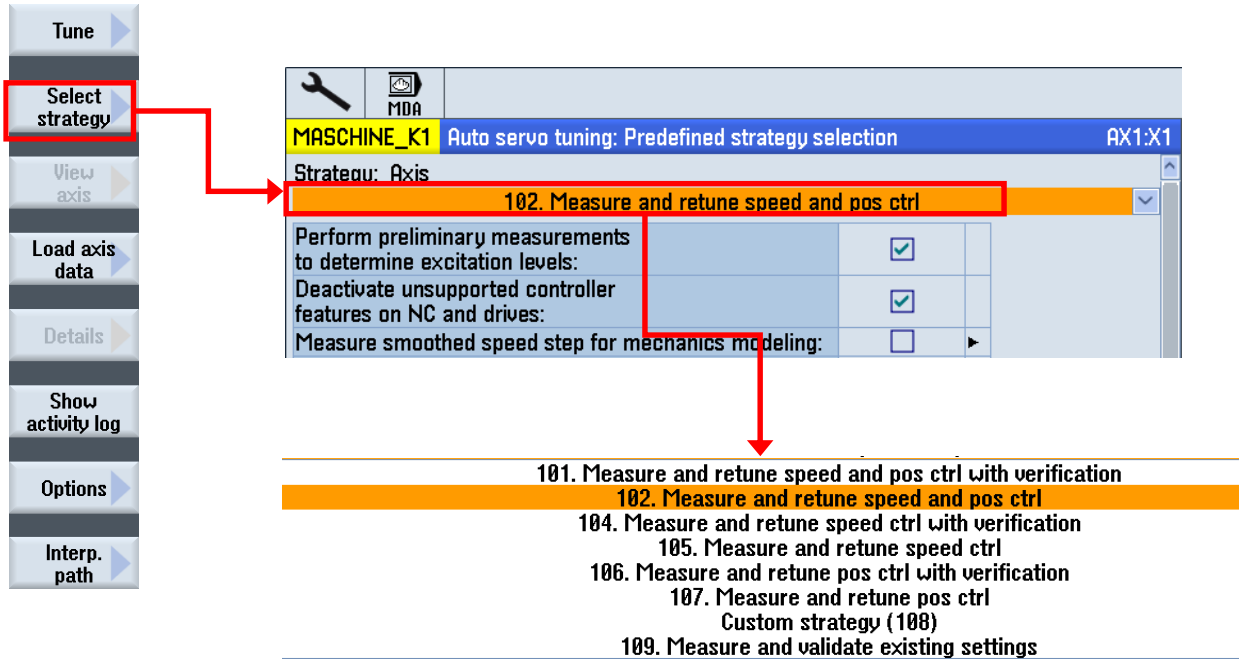
- Adaptation of the dynamic response of the axes (interpolation).
- Adaptation of the reference frequency response model with the lowest dynamic response
(lowest frequency p1433, highest damping p1434)
- Adaptation of the servo gain factor to the axis with the lowest dynamic response
(lowest value of MD 32200)
- Adaptation of the equivalent speed controller cycle time (highest value of MD 32810)
- Activation of FFW_MODE = 3 in all interpolating axes
(including spindle)

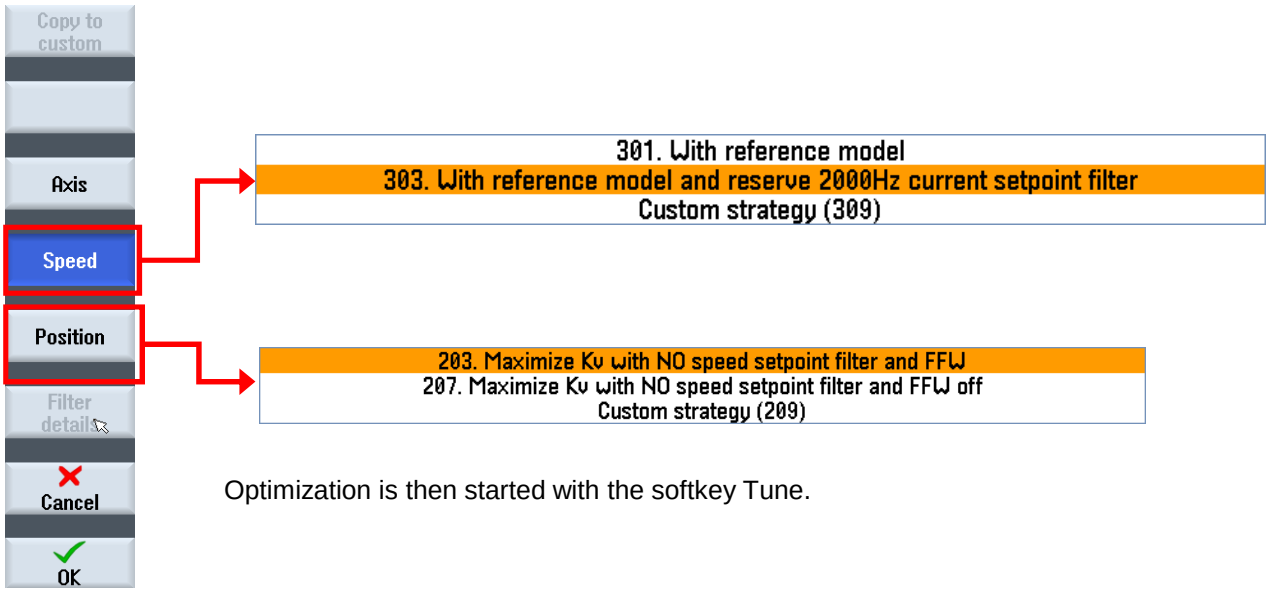
To start the auto tuning function, press the **"Optimiz./Test"** softkey in the **"Setup"** menu.
In the screen that is now displayed, select the axis to be optimized and press the **"Options"** softkey to check the basic settings.



Select the options so that optimization is completed with minimum user intervention. Check the option bits as shown above and then press the **"OK"** softkey to apply the settings.

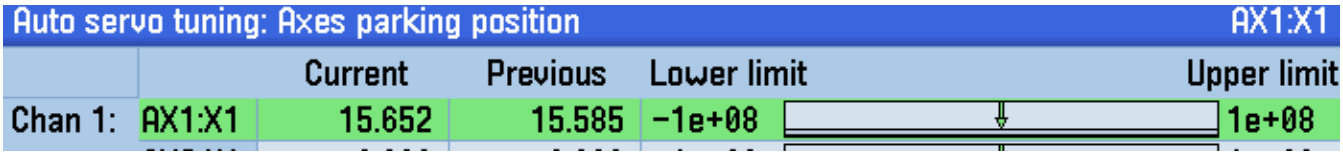
Check the strategies in the basic settings as shown here and then press **"OK"** to apply the settings.



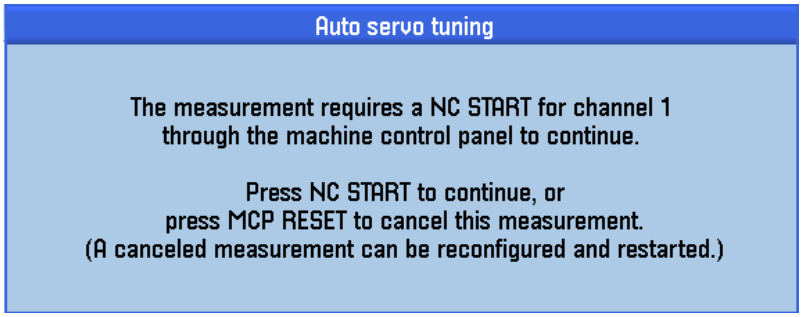


Optimization is then started with the softkey Tune.

Make sure that the axis to be optimized is in a safe position. If it is not, move the axis to the center of its traversing range in "Jog". Press the **"OK"** softkey when the axis is in a safe position.

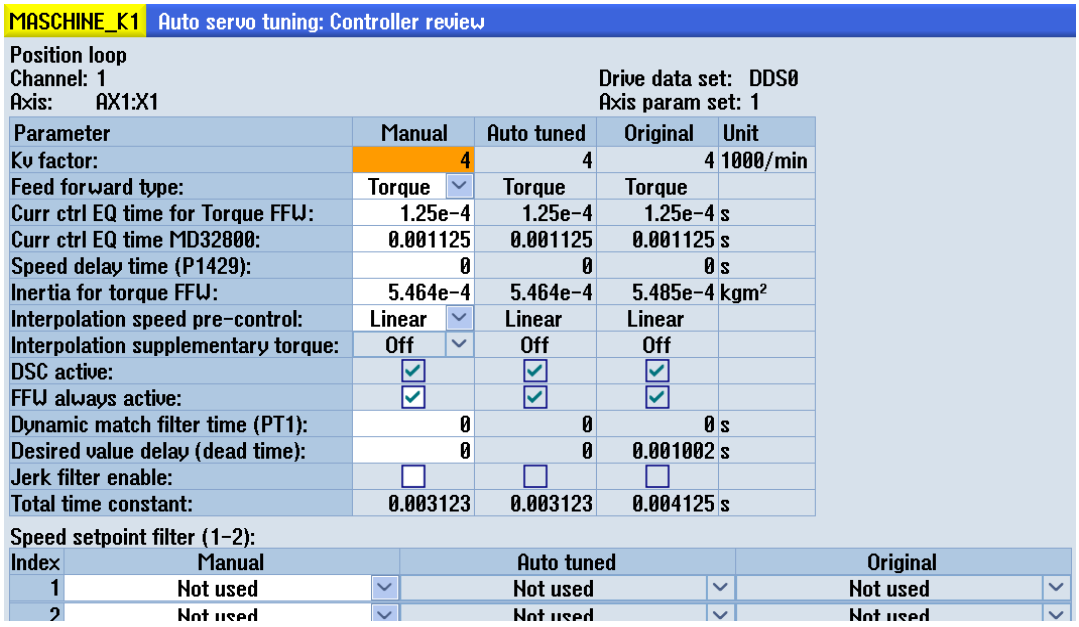


Start the measurement:

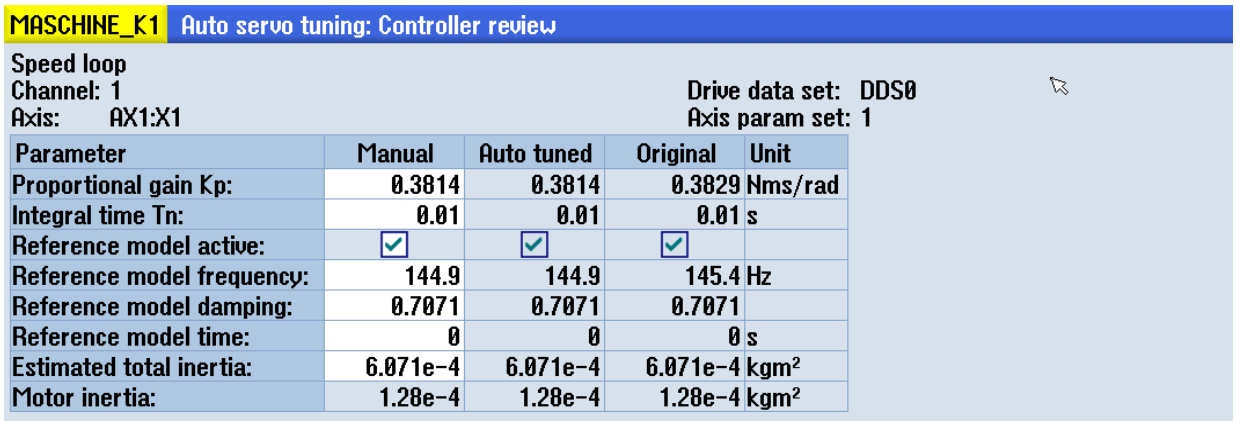


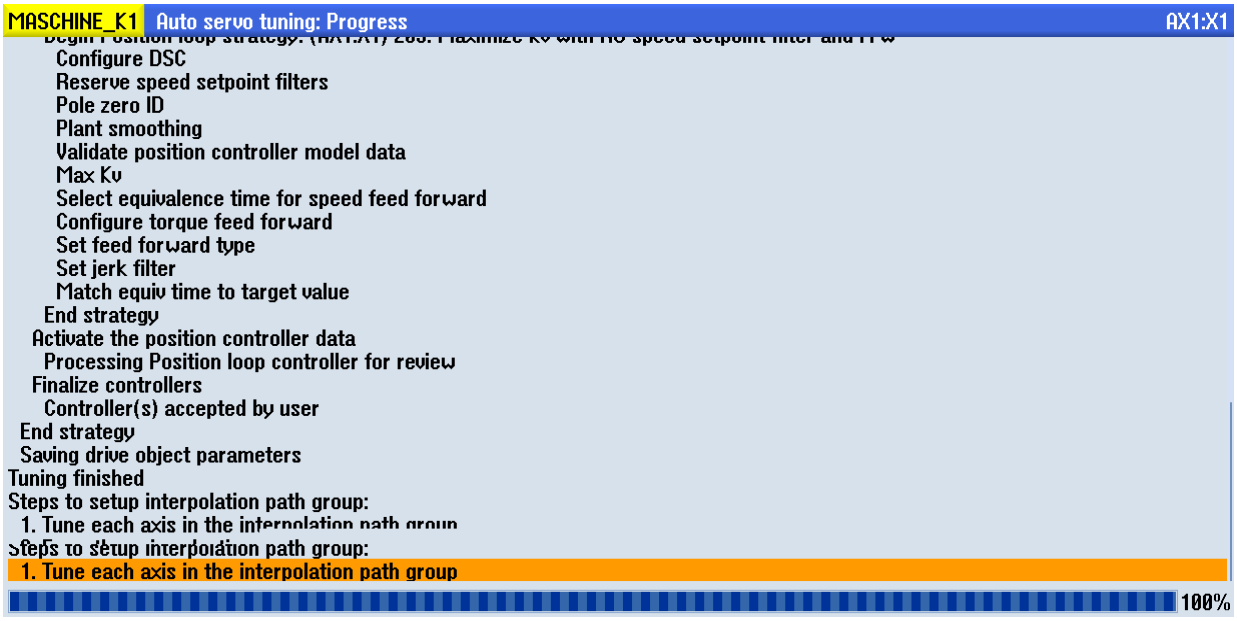
During auto tuning, the current status of the optimization procedure is displayed in a progress log.

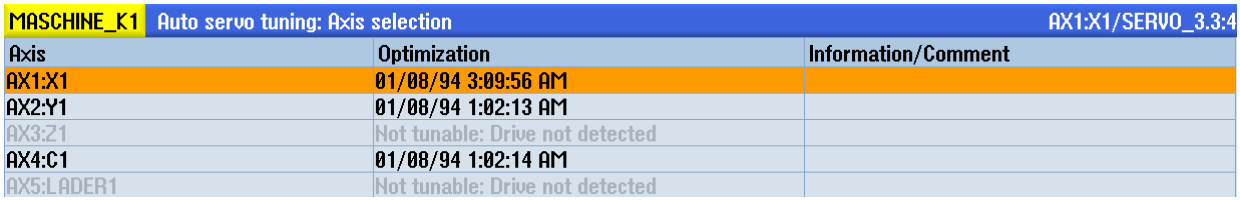
At the end of the auto tuning procedure, the optimized values are displayed for checking. In this case, the values of the position controller are displayed.



The values of the speed controller also have to be checked. To use these values, press the "Accept" softkey.
The values are now saved.
Press "OK" to complete the optimization procedure. The axis is optimized.







Once all of the axes in the system have been optimized in the same way, the parameters of the interpolating axes should be checked. Use the values YOU have calculated!
Example: 3 axes plus main spindle, 1 gear stage VMC machine.

Kv factor after auto tuning

X axis MD32200[0] $MA_POSCTRL_GAIN = 15.27
Y axis MD32200[0] $MA_POSCTRL_GAIN = 12.23
Z axis MD32200[0] $MA_POSCTRL_GAIN = 13.63
Spindle MD32200[1] $MA_POSCTRL_GAIN = 5.73

Values entered after checking
X axis MD32200[0] $MA_POSCTRL_GAIN = 12
X axis MD32200[1] $MA_POSCTRL_GAIN = 5
Y axis MD32200[0] $MA_POSCTRL_GAIN = 12
Y axis MD32200[1] $MA_POSCTRL_GAIN = 5
Z axis MD32200[0] $MA_POSCTRL_GAIN = 12
Z axis MD32200[1] $MA_POSCTRL_GAIN = 5
Spindle MD32200[0] $MA_POSCTRL_GAIN = 5
Spindle MD32200[1] $MA_POSCTRL_GAIN = 5

Equivalent time constant of speed controller for feedforward control after auto tuning

X axis MD32810[0] $MA_EQUIV_SPEEDCTRL_TIME = 0.003
Y axis MD32810[0] $MA_EQUIV_SPEEDCTRL_TIME = 0.0046
Z axis MD32810[0] $MA_EQUIV_SPEEDCTRL_TIME = 0.0042
Spindle MD32810[1] $MA_EQUIV_SPEEDCTRL_TIME = 0.0053

Values entered after checking
X axis MD32810[0] $MA_EQUIV_SPEEDCTRL_TIME = 0.0046
X axis MD32810[1] $MA_EQUIV_SPEEDCTRL_TIME = 0.0053
Y axis MD32810[0] $MA_EQUIV_SPEEDCTRL_TIME = 0.0046
Y axis MD32810[1] $MA_EQUIV_SPEEDCTRL_TIME = 0.0053
Z axis MD32810[0] $MA_EQUIV_SPEEDCTRL_TIME = 0.0046
Z axis MD32810[1] $MA_EQUIV_SPEEDCTRL_TIME = 0.0053
Spindle MD32810[0] $MA_EQUIV_SPEEDCTRL_TIME = 0.0053
Spindle MD32810[1] $MA_EQUIV_SPEEDCTRL_TIME = 0.0053

Speed controller reference model of natural frequency after auto tuning

X axis p1433[0] Speed controller reference model natural freq = 126.6
Y axis p1433[0] Speed controller reference model natural freq = 118.3
Z axis p1433[0] Speed controller reference model natural freq = 122.8
Spindle p1433[0] Speed controller reference model natural freq = 111.6

Values entered after checking
X axis p1433[0] Speed controller reference model natural freq = 110
Y axis p1433[0] Speed controller reference model natural freq = 110
Z axis p1433[0] Speed controller reference model natural freq = 110
Spindle p1433[0] Speed controller reference model natural freq = 110

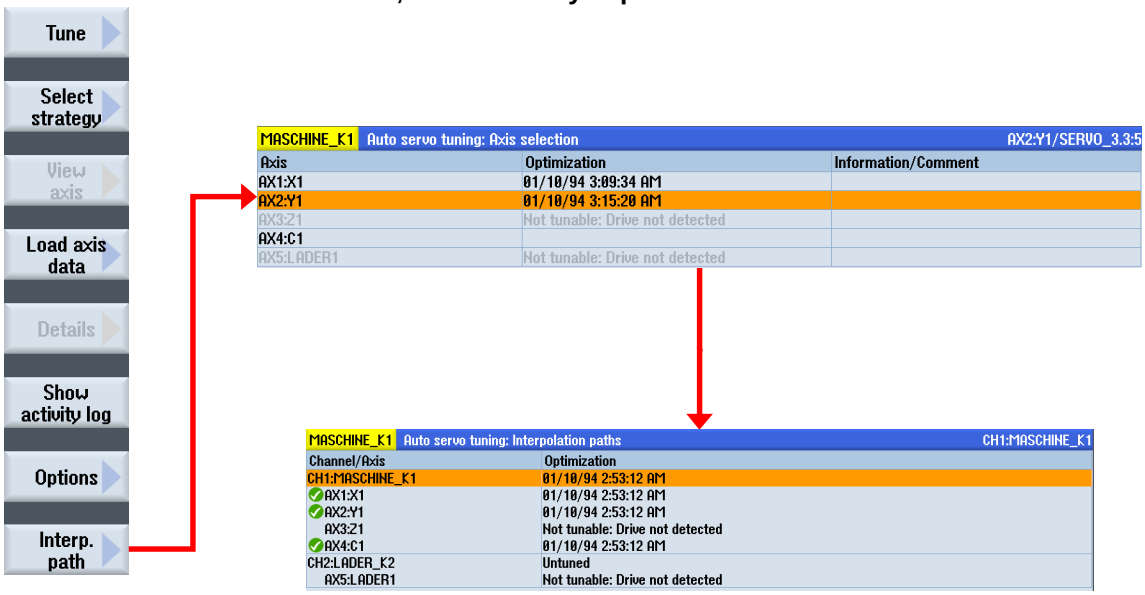
Speed controller reference model of damping after auto tuning

X axis p1434[0] Speed controller reference model damping = 0.707
Y axis p1434[0] Speed controller reference model damping = 0.707
Z axis p1434[0] Speed controller reference model damping = 0.707
Spindle p1434[0] Speed controller reference model damping = 0.707
If values differ, the largest value should be entered in all axes.

NOTE Since the Z axis and the main spindle are usually in the second parameter set [1] during thread cutting, the second parameter set of the feed axis should be adjusted to the main spindle as shown above. This also means that the dynamic response of the axes in an interpolating axis group in the first parameter set [0] is not limited so that it can adjust to the main spindle. Therefore, we will use the axis values with the slowest dynamic response in the parameter set [0] and then adjust the second parameter set [1] so that they can work with the main spindle.

Additional exercise: Interpolation path

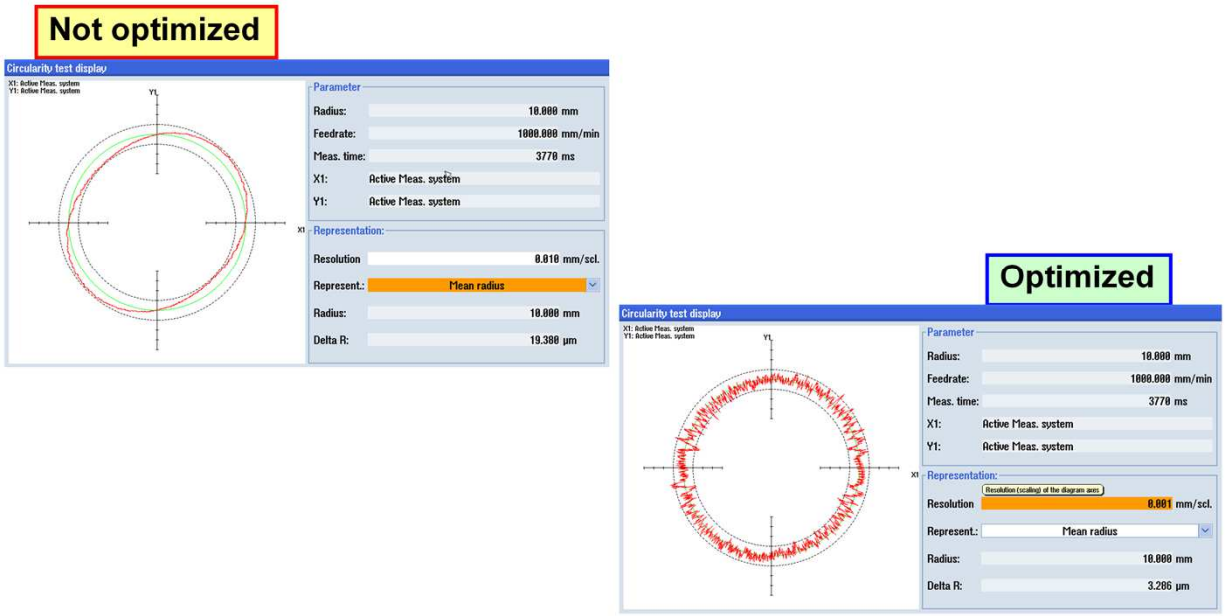
As an additional exercise following optimization of X, Y, you can trigger optimization of the interpolation path for X and Y.
-> Softkey: Path interpolation -> Select channel 1 -> softkey: Edit & tune
-> Select axes: X, Y -> softkey: Optimize



Tip

If an error occurs at the start of this optimization routine, try the following:
1. Softkey: Display controller
2. Softkey: Accept
for all axes to be interpolated and
restart optimization of the interpolation path!

Exercise: Circularity test with AST



Course NC-84SL-SIP Page 9.3 - 44 SITRAIN © Siemens AG 2017

Objective To use the circularity test to assess optimization .

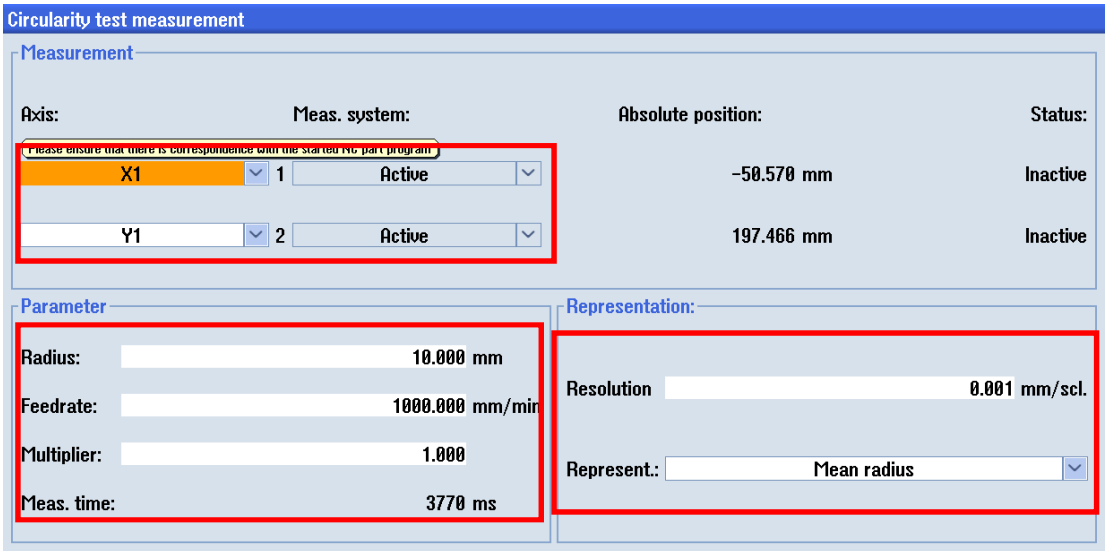
Steps

1 Parameterize test
2. Generate circle program
3. Start the program and start the circularity test
4. Display
5. Change parameters like servo gain
   -> Check how this affects the test

Sequence

1. **Select OA "Setup" -> softkey "Optimiz./Test"** and **softkey "Circular. test"**.
2. **Set the test parameters**

   Change the parameters (see image).



2. Generate circle program

With softkey "**Generate circ. prog",** a circle program is generated, in accordance with the entered measurement parameters, with a radius and feedrate parameterized and selected.

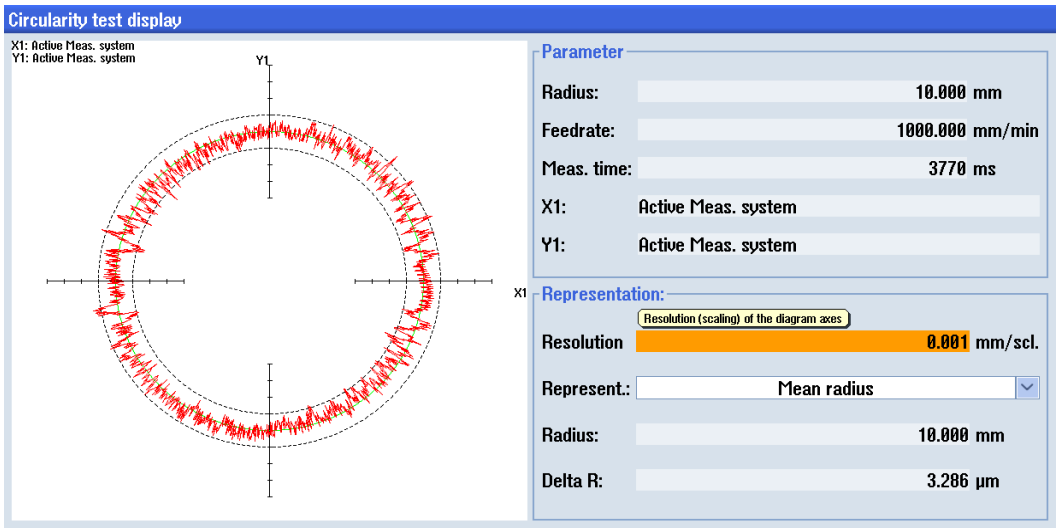
3. Start the circle program

Switch to automatic mode and press the NC Start button.

4. Display

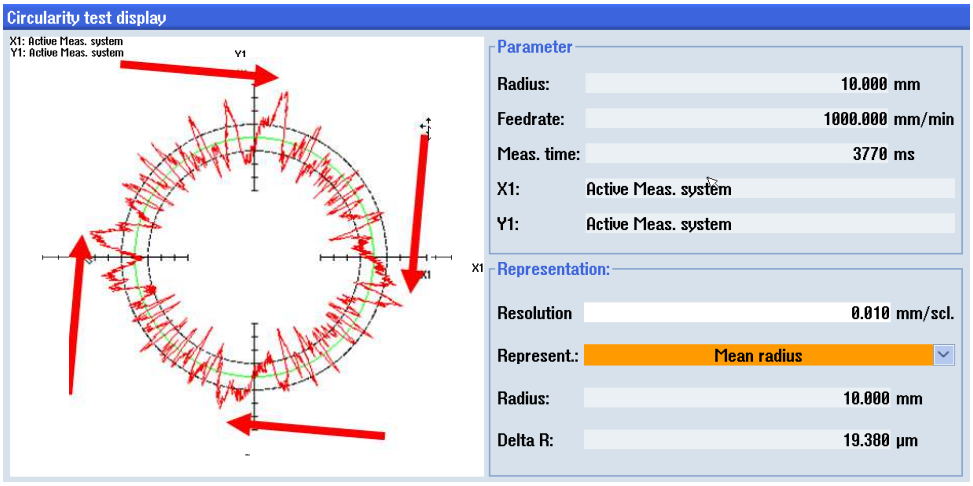
On completion of the circularity test, press the **"Graphic" softkey** to view the measurement.

Example: Graphic following optimization of the interpolation path (AST):



By displaying the medium radius, conclusions can be drawn about optimization of the speed controller. For axes that have not been optimized, deviations from the target circle are more significant, particularly at quadrant transitions:

Example: Circularity test prior to optimization:



5. Impact of parameters such as servo gain, etc.

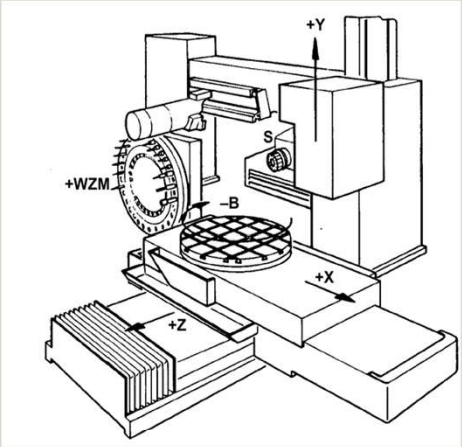
Investigate the impact of parameters such as servo gain (Kv)(position controller) = MD 32200
or proportional gain (Kp)(speed controller) = p1460 on the circularity test.

Set, for example:

	X	Y
a) Kv = MD 32200	4	2
b) Kp = p1460	0.2	0.5

and repeat the circularity test in each case.

SIEMENS



Chapter 10

Extended Startup

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

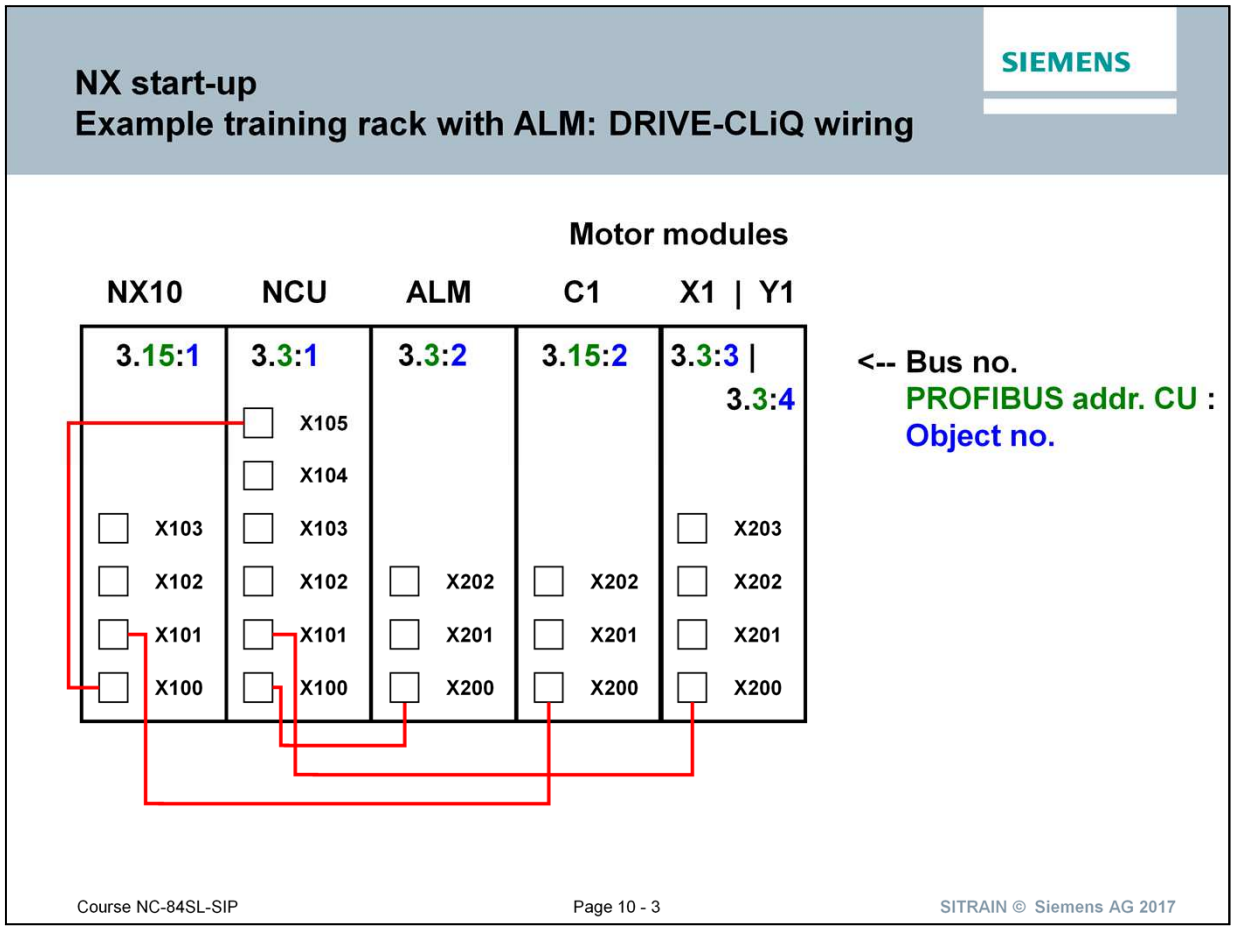
Learning objectives 2
NX start-up 3
SINAMICS start-up: Device configuration NX 7
NX start-up: 9
Overview: Topology changes for acknowledgment 10
Adding components 11
Adding objects 14
Adding an NX 15
Deleting an object 16
Run-up with partial topology 19
Module exchange (Setup Archive) 21
Confirm topology change 22
Swapping components 23
Backing up/Loading drive data 24
Global User Data (GUD) 27
Basic settings 32
G groups 33
G code machine data 34
PLC interface G functions 35
G codes 36
Exercise: NX start-up HW Config 37
Exercise: NX start-up by adding components 39
Exercise: Starting up a second encoder 45
Appendix 51
Program lists: Activation; editor; selection 65
Program lists: Control from the PLC 66

Learning objectives

- You know how to start up NX modules
- You know how to make changes to topologies

SITRAIN © Siemens AG 2017

Notes



Introduction

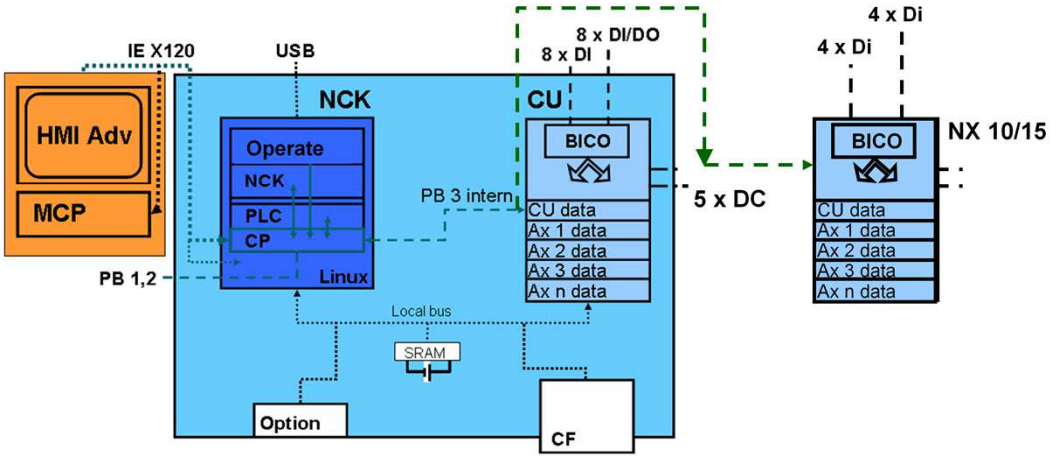
Machines with more than 6 axes require additional NX controller modules.
The following variants are available:

- NX10 for 3 additional axes
- NX15 for 6 additional axes

Training rack

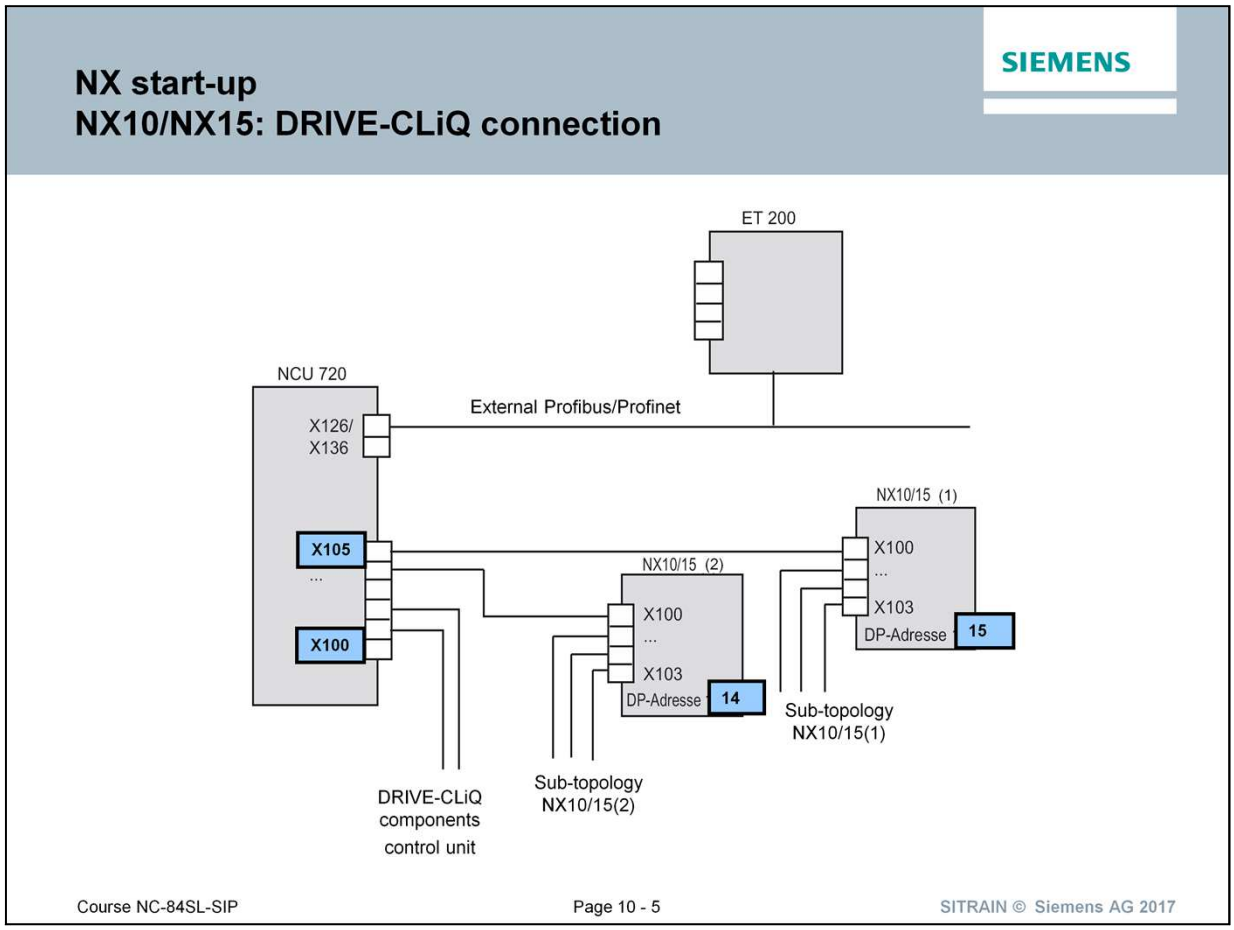
An NX10 module is available on the training rack.
It can be used to practice start-up.

NX start-up
NX10/NX15: Internal PROFIBUS via DRIVE-CLiQ



SITRAIN © Siemens AG 2017

Notes



Rules for DRIVE-CLiQ wiring

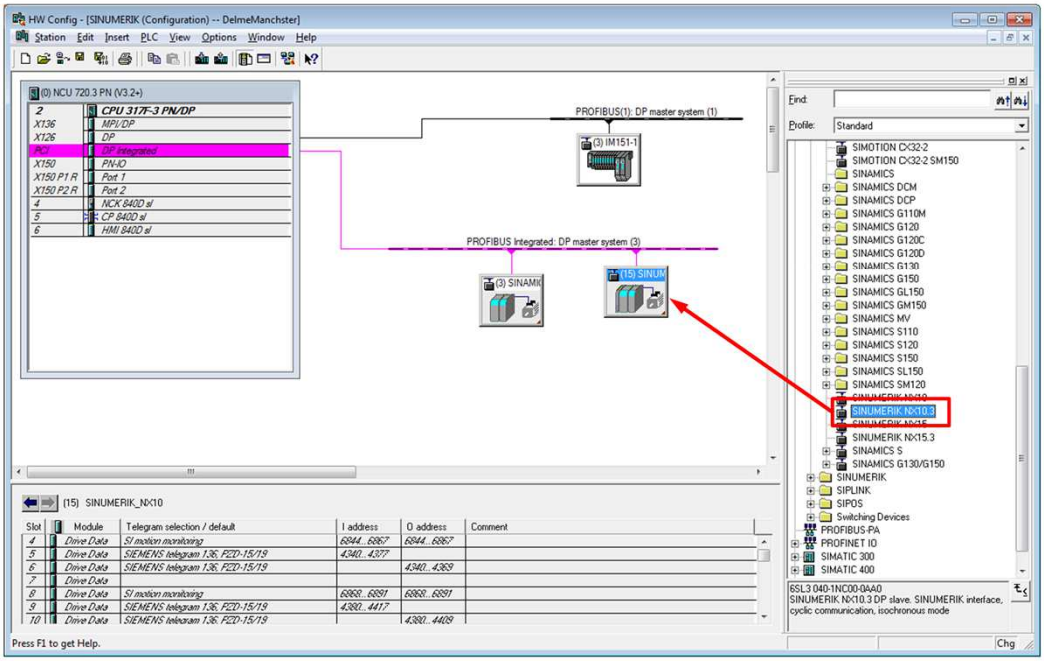
NX10/15 components can be connected to the control unit via DRIVE-CLiQ.

The following rules apply to wiring of the NX10/15:

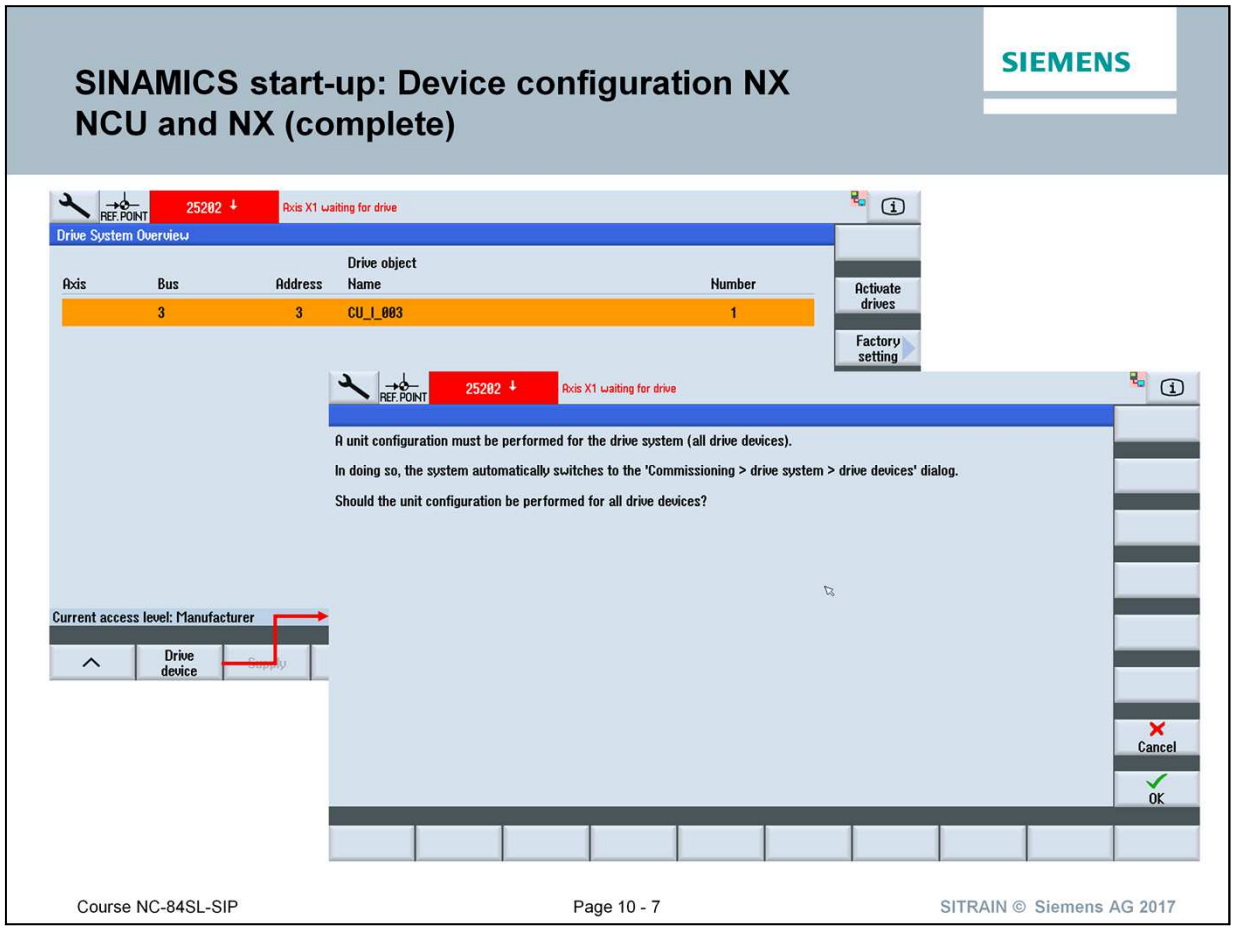
- Only one **star topology** is permitted between the NX10/15 and the control unit. This means that only one NX10/15 can be operated per DRIVE-CLiQ port on a control unit.
- DRIVE-CLiQ ports not assigned to NX10/15 can be wired to other DRIVE-CLiQ components.
- Once an NX10/15 has been connected and configured, you cannot simply insert it into a different DRIVE-CLiQ port, as the addresses of the integrated drives are set permanently from the point of view of the PLC. The following table illustrates this relationship:

DRIVE-CLiQ Port	PROFIBUS address of the CU
X105	15
X104	14
X103	13
X102	12
X101	11

The PROFIBUS addresses of the NX modules can only be modified at the time they are inserted. This determines the slot assignment on the NCU DRIVE-CLiQ port.



From the catalog PROFIBUS DP, SINAMICS section, the corresponding NX module is appended to the integrated PROFIBUS.

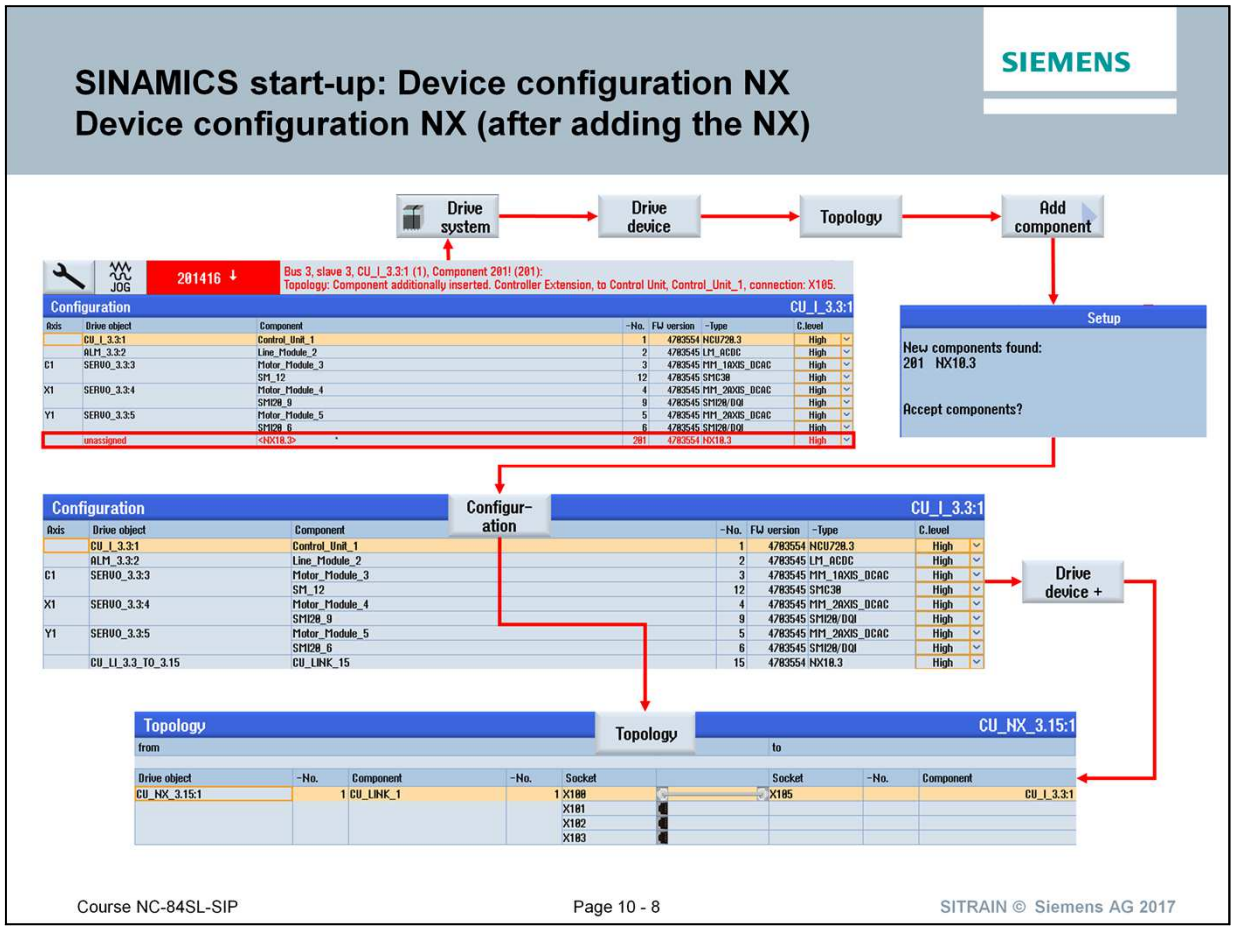


Commissioning axes Several strategies can be applied to commission a control system with one or several NX modules:

- **Simultaneously** From the very beginning, all NX are imported into HW Config, and then device configuration is performed in the factory setting state.

- **Sequential** To start, only the CU of the NCU is commissioned together with its axes, and then the NX modules are subsequently added:
  CU and its drives are commissioned
  HW Config is expanded to include the NX modules
  The NX modules and their drives are connected
  Components are added using SK: „Add component“

  The second way is subsequently described.



Requirements

You have done the device configuration of the NCU.

You have also added an NX to the NCU in the HW Config.
The NX module has been connected to the NCU accordingly via the DRIVE-CLiQ cable. The NX is connected to other Motor Modules.

- **Adding components**

  Next, you must add the NX module with the 'Add component' softkey in the 'Topology' screen.

- **Device configuration**

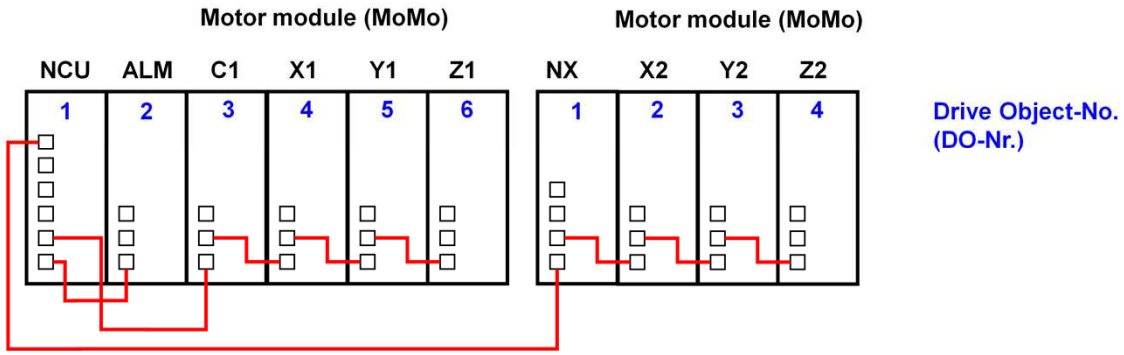
  Device configuration (which is offered automatically) can then be carried out.

- **Manual device configuration**

  Alternatively, you can carry out device configuration at a later time as follows:

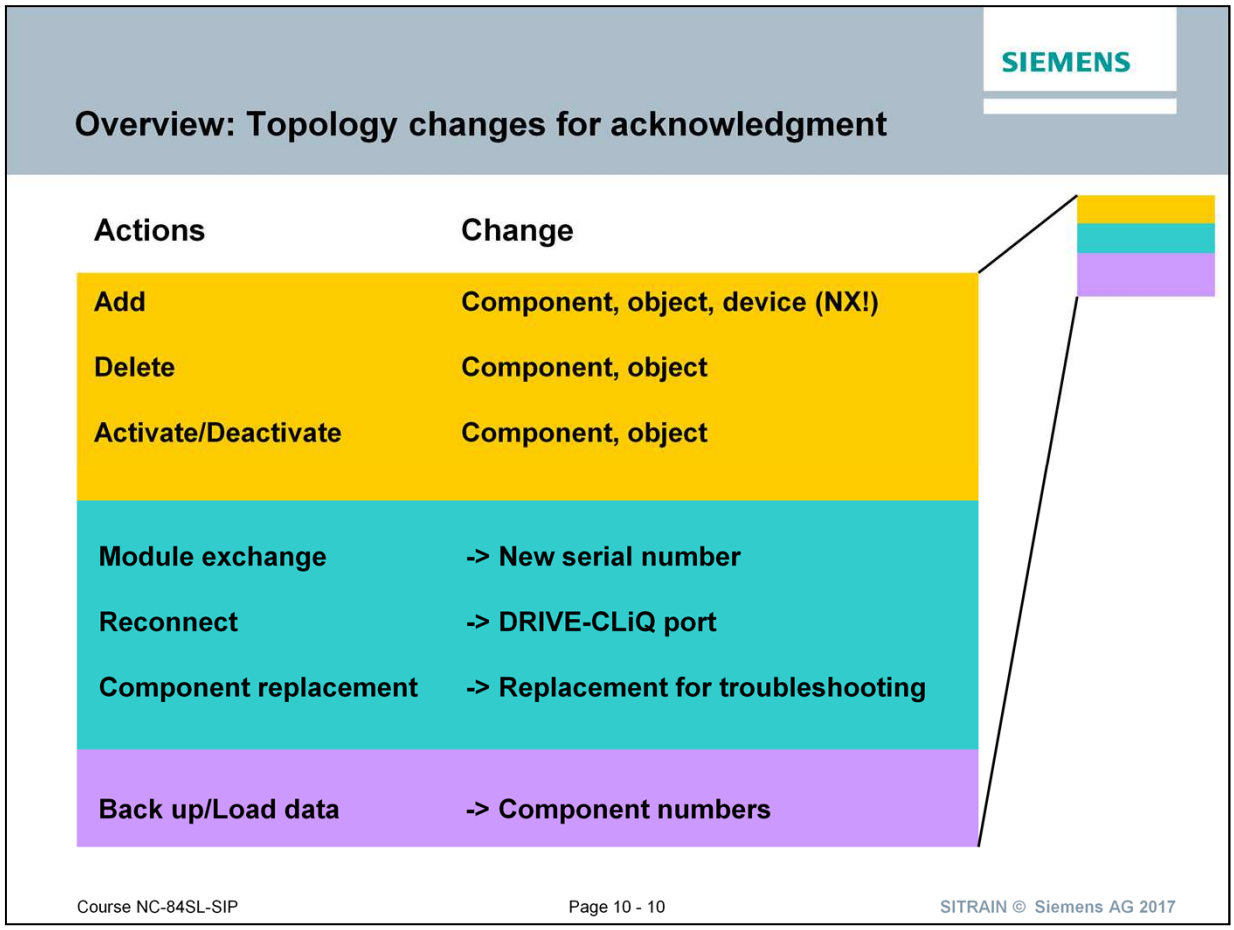
  1. Select the drive device of the NX:
     - Either in the 'Drive system' menu with the cursor keys and by pressing the “Drive device” softkey
     - Or in the 'Drive devices' menu with the softkeys "Drive device + / Drive device -"
  2. The HMI detects that device configuration has not yet been carried out and prompts you to do this.
  3. Confirm all prompts up to saving and power on at the end.

NX start-up:
Example communication – DRIVE-CLiQ wiring

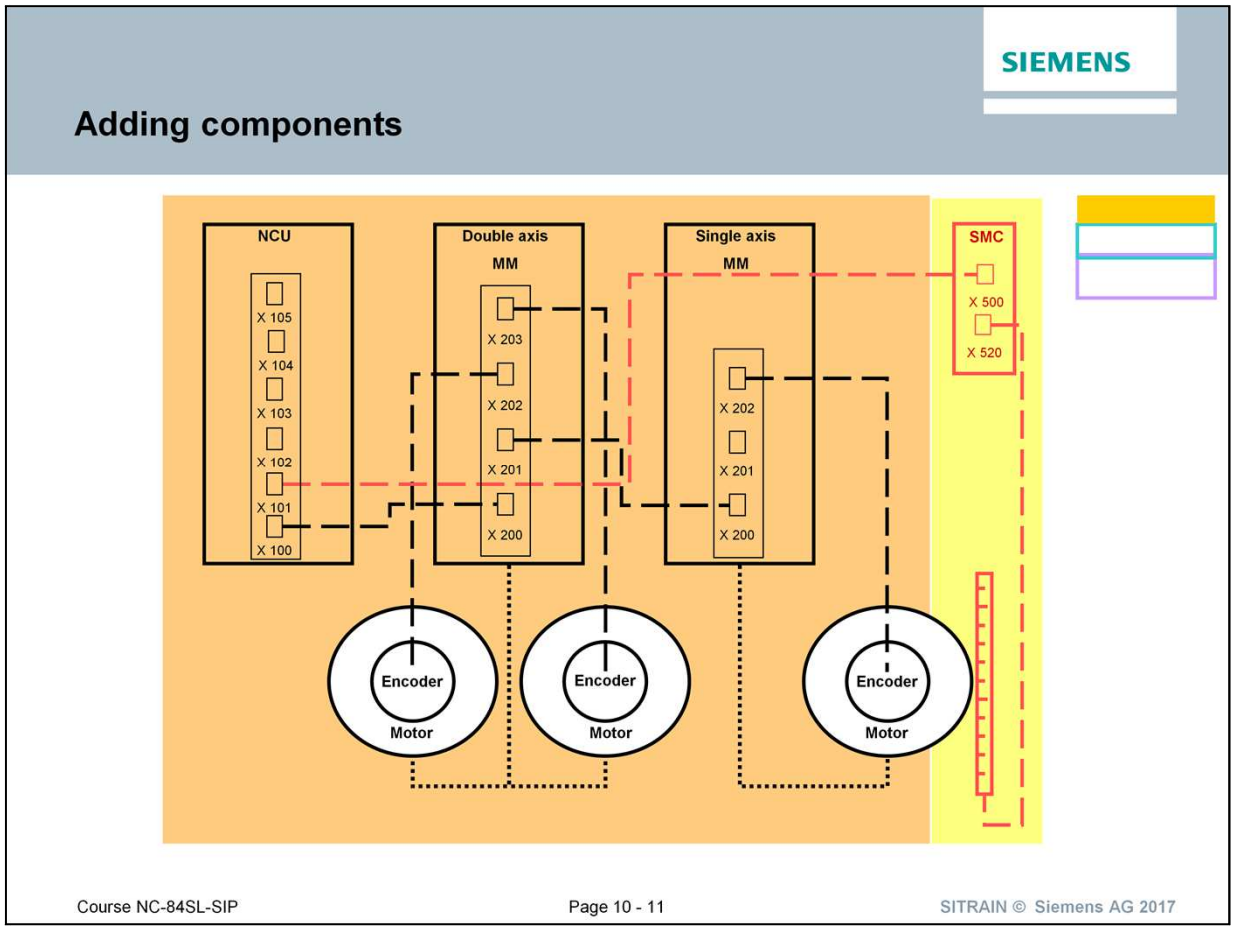


SITRAIN © Siemens AG 2017

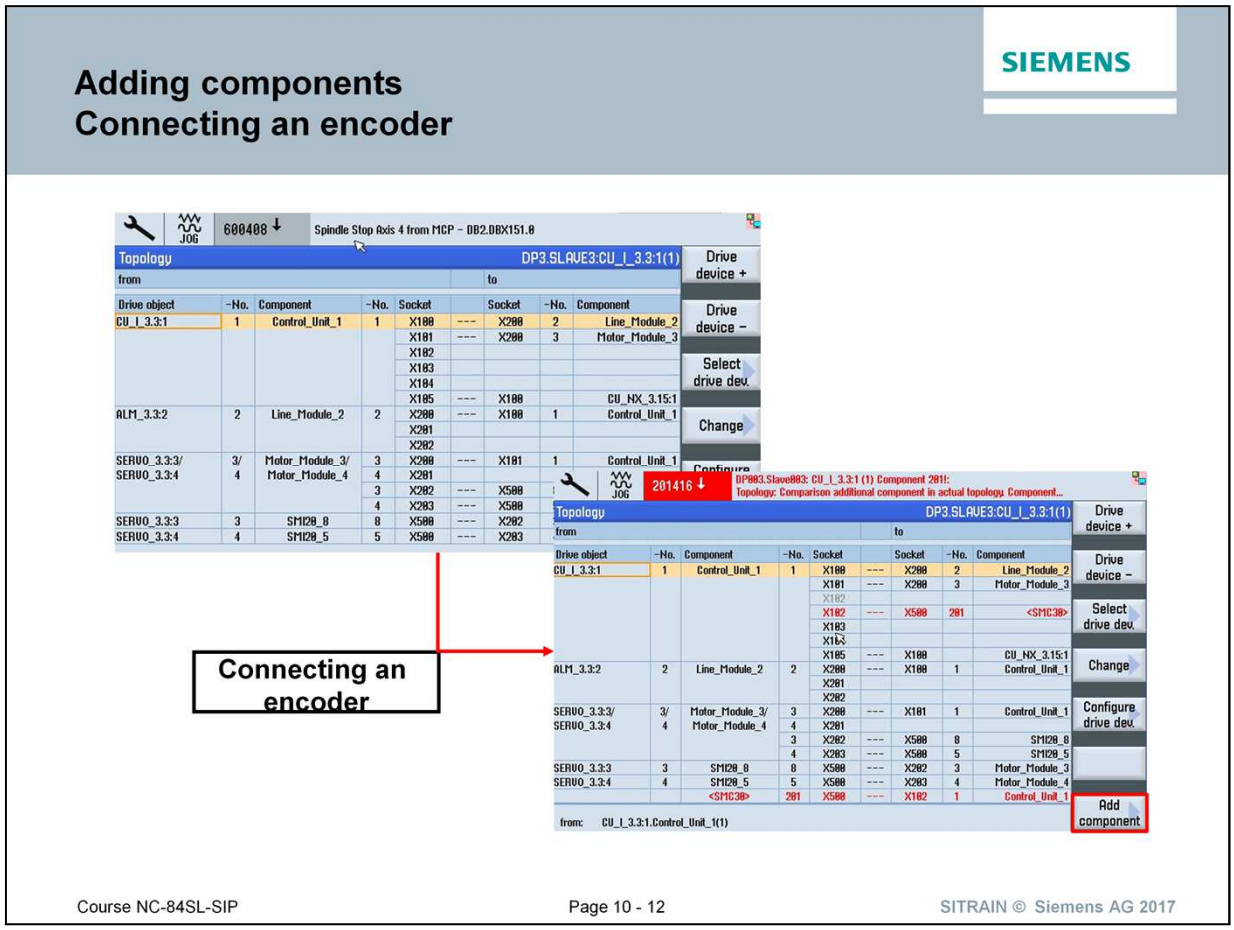
Notes


SIEMENS
Overview: Topology changes for acknowledgment
Actions
Change
Add
Component, object, device (NX!)
Delete
Component, object
Activate/Deactivate
Component, object
Module exchange
-> New serial number
Reconnect
-> DRIVE-CLiQ port
Component replacement
-> Replacement for troubleshooting
Back up/Load data
-> Component numbers
Course NC-84SL-SIP
Page 10 - 10
SITRAIN © Siemens AG 2017

Notes



Example A SMC is mounted subsequently with a second encoder.



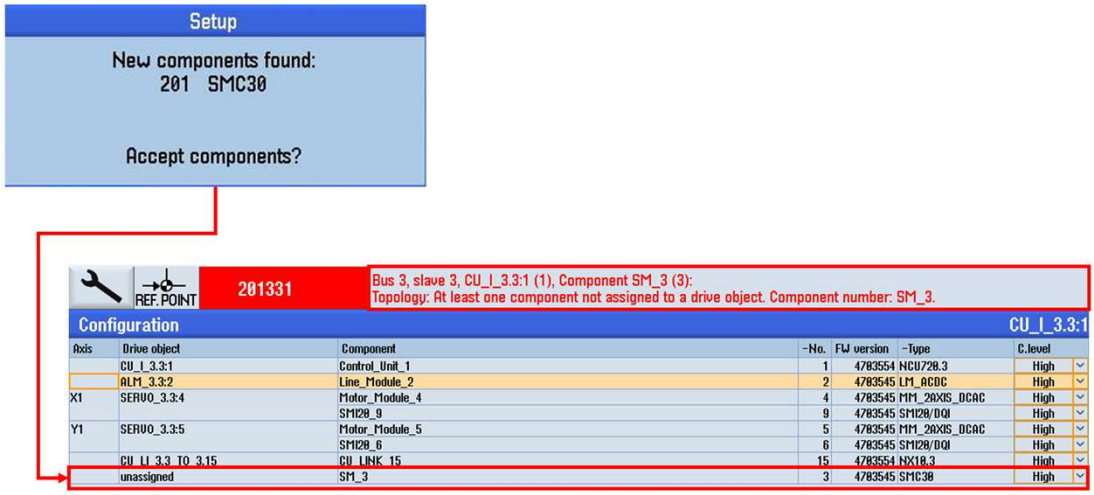
Adding components

Connection of the new components

-> Alarm 201416 appears.

-> SK Add components

Adding components
Confirming



 SITRAIN © Siemens AG 2017

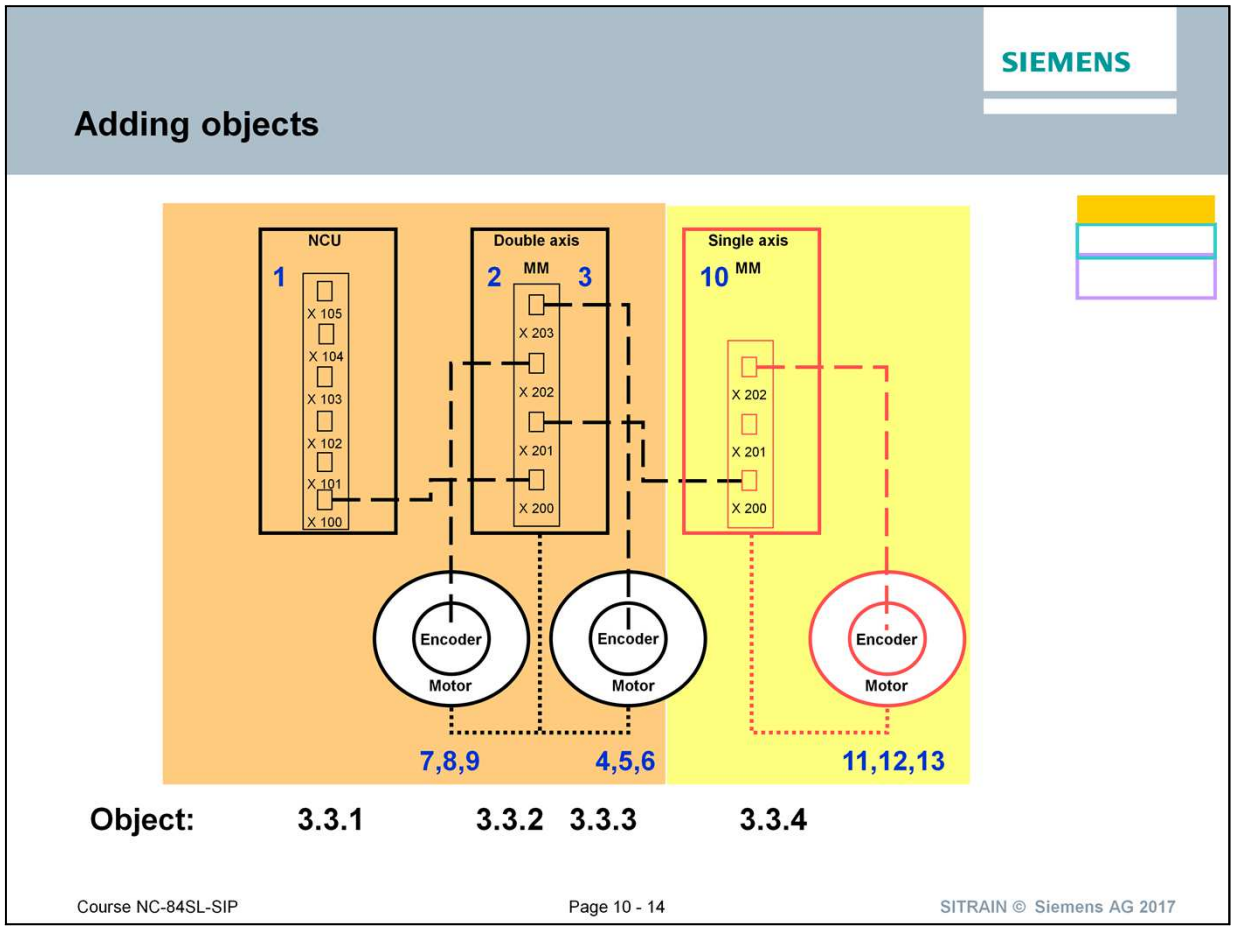
Fault 201331

The additional component is now in the target topology; however, it has still not been assigned to any object. This assignment to a specific object is described in one of the exercises.

Assign axis

The “Assign axis“ function is used to assign an object to an axis. This function has already been described in the Chapter, Basic setup. This axis assignment must always be made after setting up all of the encoders, as explained here, as when the axis is assigned. The drive also communicates to the axis that it has two encoders, (MD30200); it also communicates the encoder pulse number, encoder type etc.
This is described in an exercise.

.

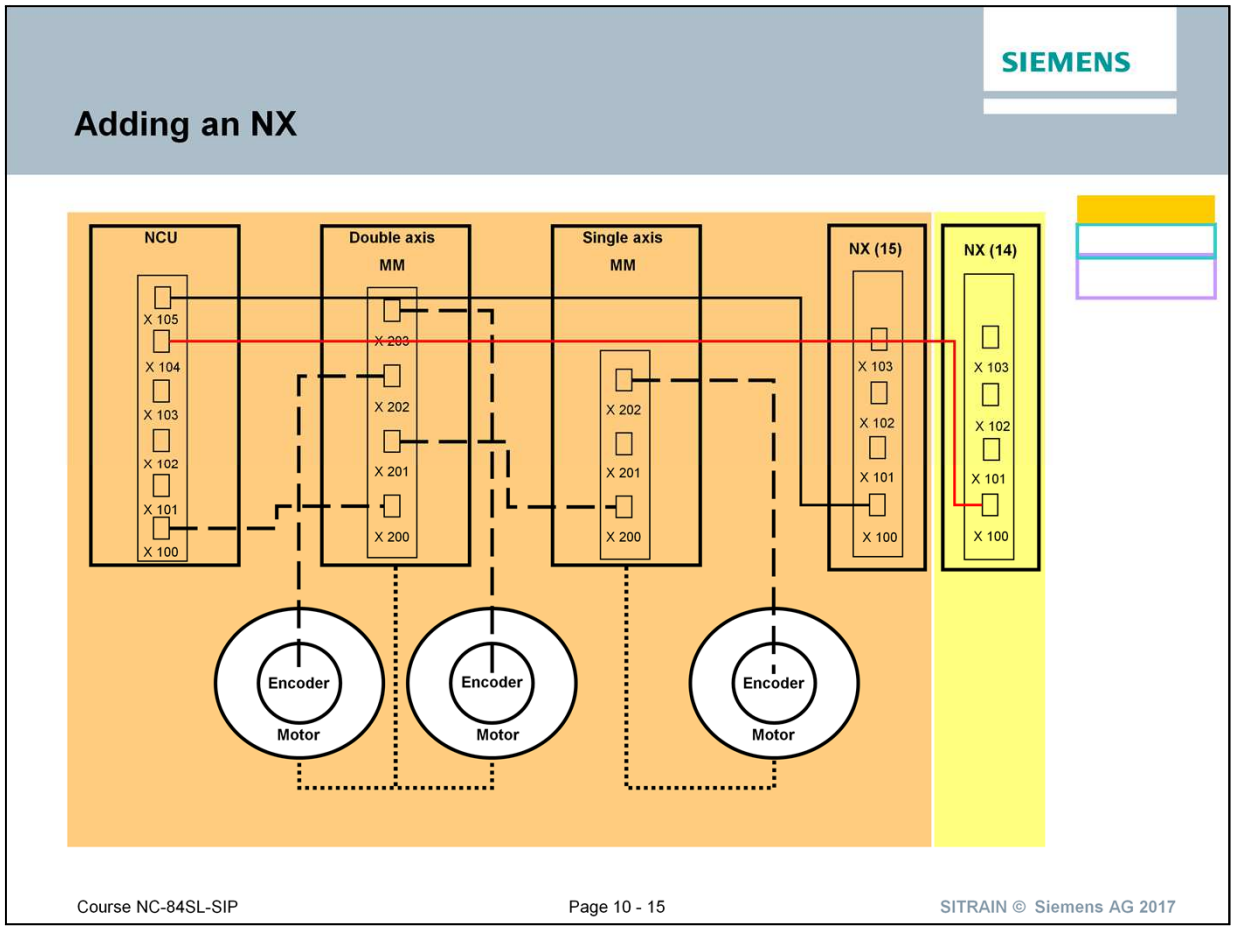


Example

Subsequently, a Motor Module (including motor, Sensor Module, and encoder) is connected to the drive line-up that has already been configured.

Procedure

Objects are also included in the target topology using the “Add component“ function. As soon as the control identifies a Motor Module as new component, then a new drive object is automatically created. The numbering of existing objects and their associated components is not modified when doing this. Once the object has been created, the commissioning engineer specifies components that have not been automatically identified, and assigns them to an axis, as was already described in the Chapter, Basic setup.

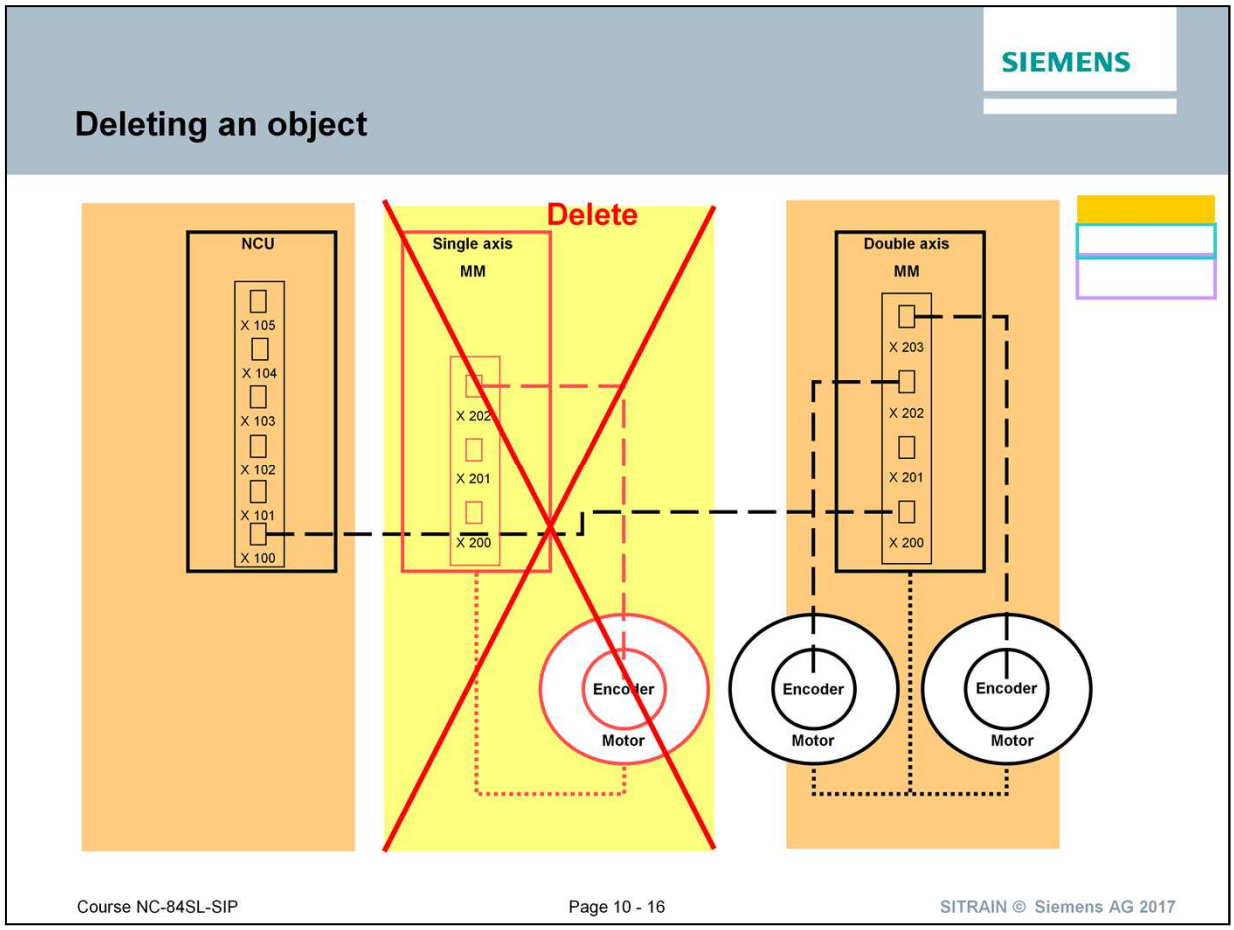


Example Subsequently, an NX module (with Motor Module(s)) is connected to the NCU.

NX in HW Config After the new NX was inserted, as described in the Step 7 Hardware configuration, it is added just like a component.

Adding a component

In the next step, the newly created drive device must then be set up. This is to again create a target topology from the drives that are connected to the NX module.

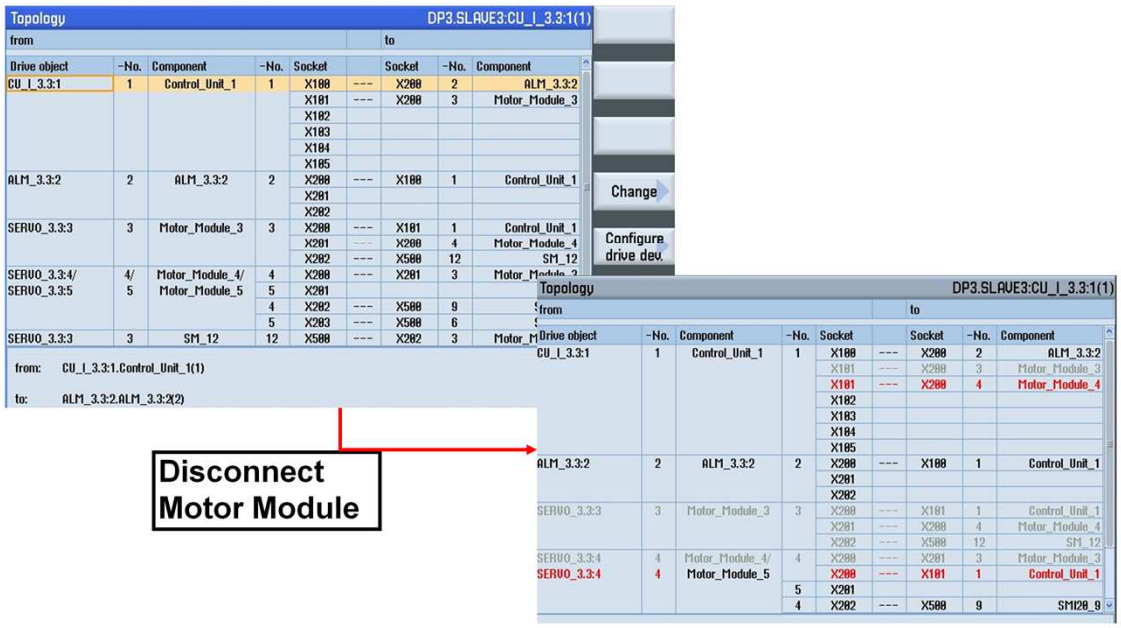


Deleting an object

An object is removed from the drive line-up by bridging the DRIVE-CLiQ connection.

The rest of the machine is to remain operational.

Deleting an object
Disconnecting a Motor Module

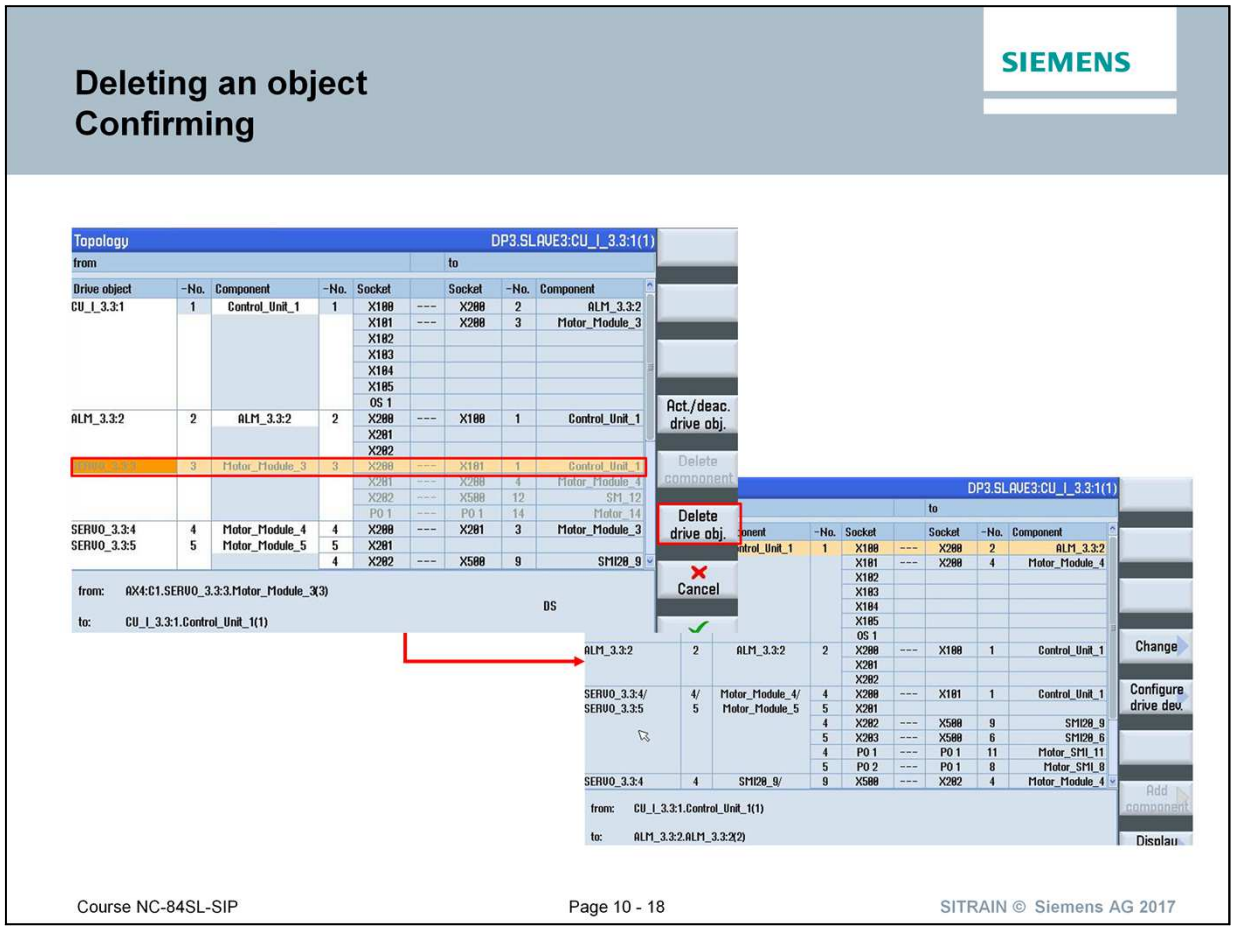


Course NC-84SL-SIP Page 10 - 17 SITRAIN © Siemens AG 2017

Disconnecting a Motor Module

The object is disconnected with the DRIVE-CLiQ plug.

-> Alarms such as 207800, 201481, etc. are displayed.



Confirming

Press the softkeys:

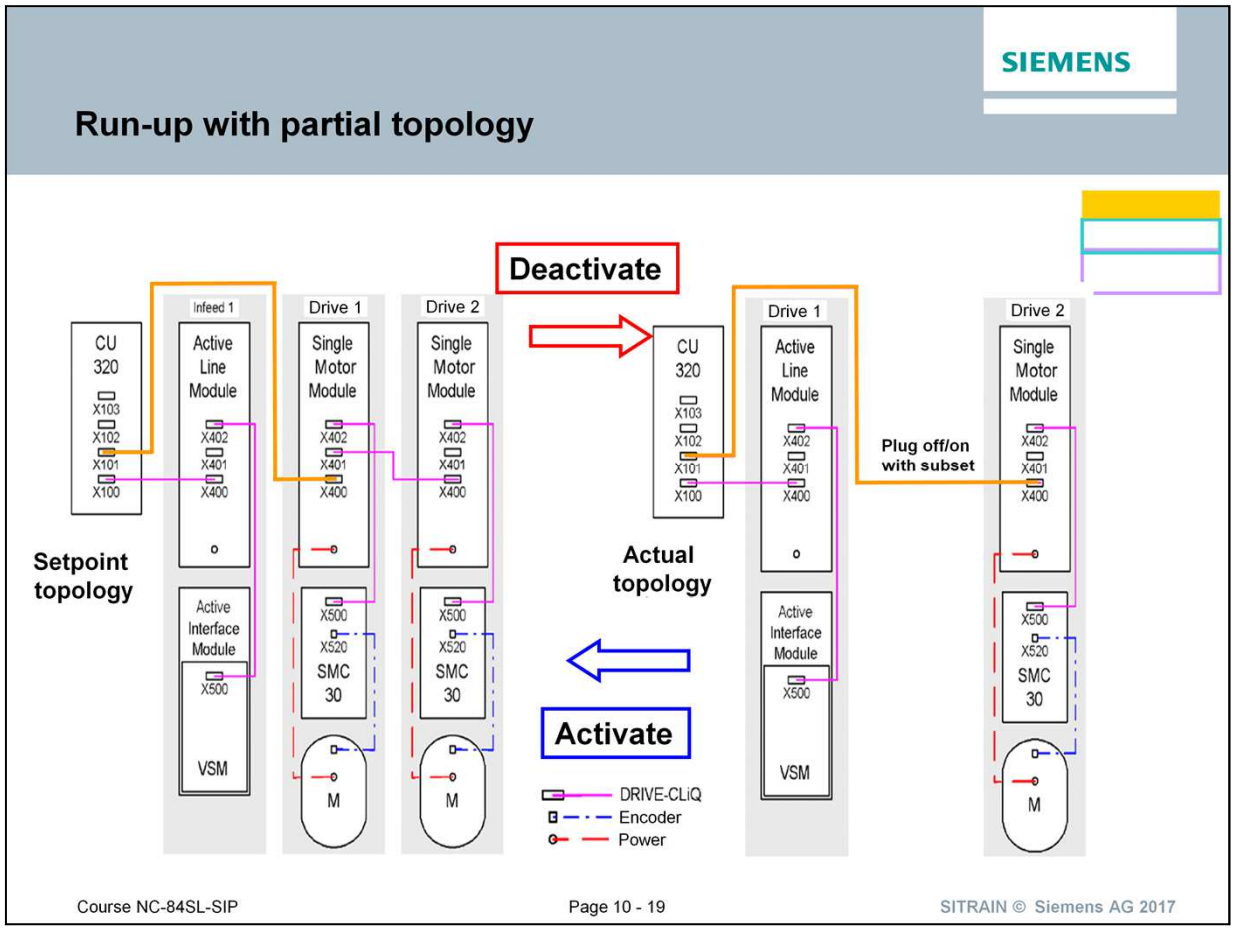
-> SK: Topology -> SK: Change

Click "OK" and then make a selection (**left!**) and press

-> **SK: Delete drive object**

to delete the object.

Confirm with "OK". (MD 30130 and 30220 are set to 0.)

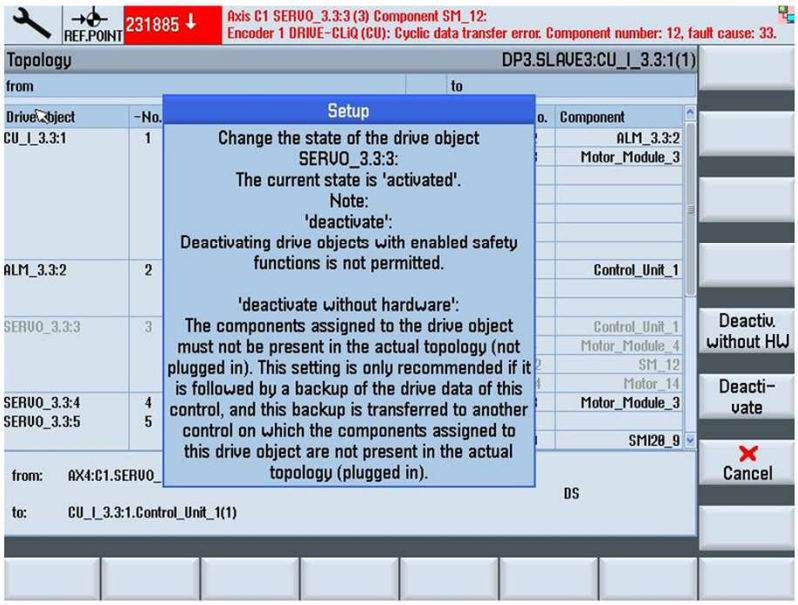


Overview

- **Deactivate** A drive object can be deactivated with
  SK: Deactivate
  The object can be removed from the drive configuration at the same time.
  Differing from deletion, the drive data are retained.

- **Activate** An object can be reactivated with
  SK: Activate.

Run-up with partial topology
Deactivating



SITRAIN © Siemens AG 2017

Removing a Motor Module

With the voltage disconnected, remove the Motor Module from the DRIVE-CLiQ connection.

After power ON, the drive is marked as "not ready for operation".

Corresponding alarms like 201315"...Drive object not ready for operation" are displayed.

Deactivating a drive object

This function is accessed via

-> SK: Topology -> Change -> Act./deact. drive obj.

There are two ways of deactivating a drive object:

- **Deactivate**

-> SK: Deactivate

This is the standard scenario, e.g. when removing faulty components from the DRIVE-CLiQ segment.

- **Deactivate without hardware**

-> SK: Deactiv. without HW

You should only select this option if the object concerned does not appear in the actual topology.

This is only recommended in the specific case that a drive data backup is being created so that it can be written to another machine **without** the corresponding hardware.

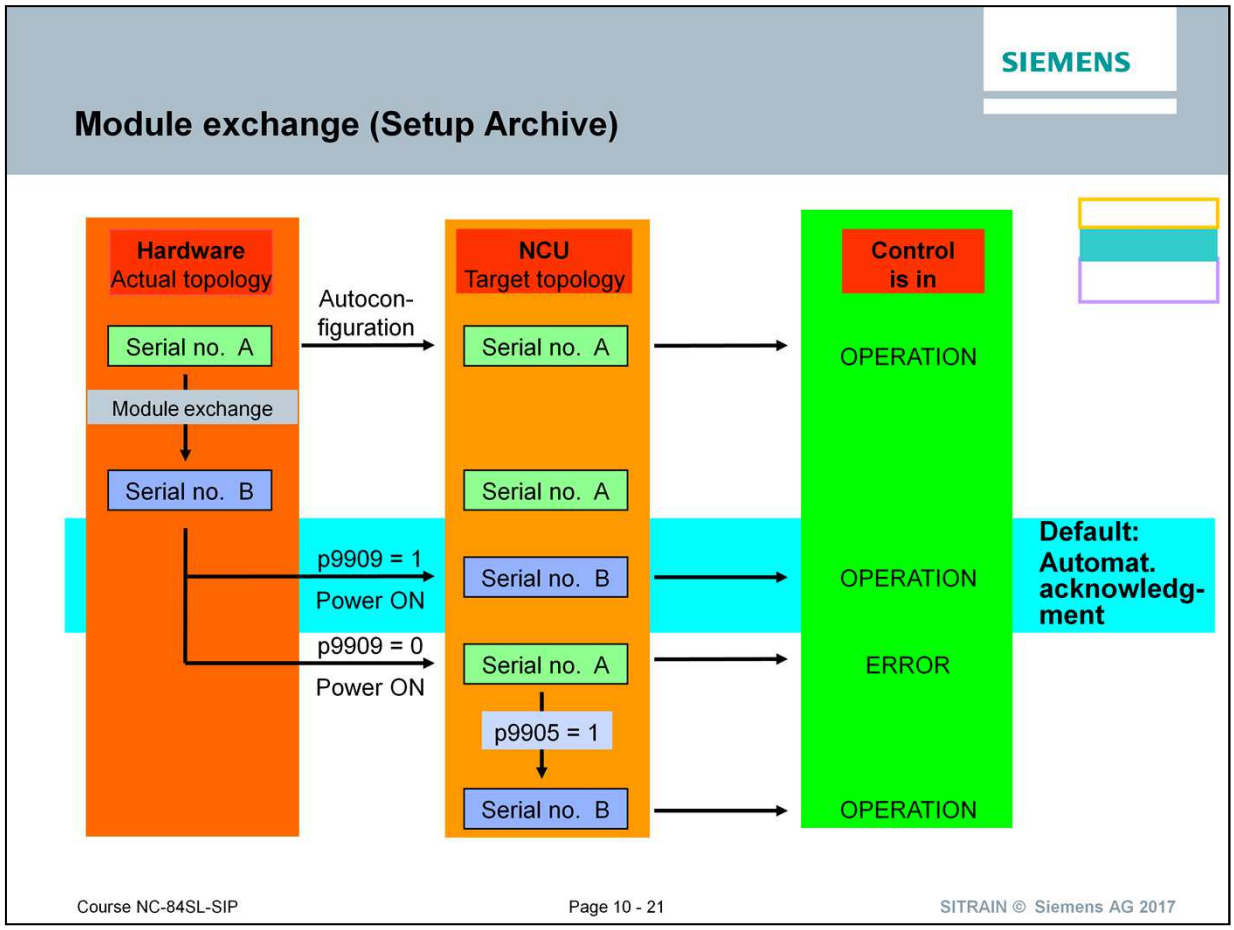
-> No alarm

In this case, the result (after Power On) is:

- No alarm despite topology error being displayed
- Other axes can be traversed

NOTICE
Safety

You are not permitted to deactivate drive objects with activated safety functions.



Module exchange

When a module is exchanged, the serial number of a component changes. How does this affect topology matching (acknowledgment)?

Acknowledgment module exchange

When a module is exchanged, the new serial number can be acknowledged in the following two ways:

- **Automatically**

  - p9909 = 1 (default): Automatic acknowledgment:
    The new serial number is accepted automatically.

- **Manually**

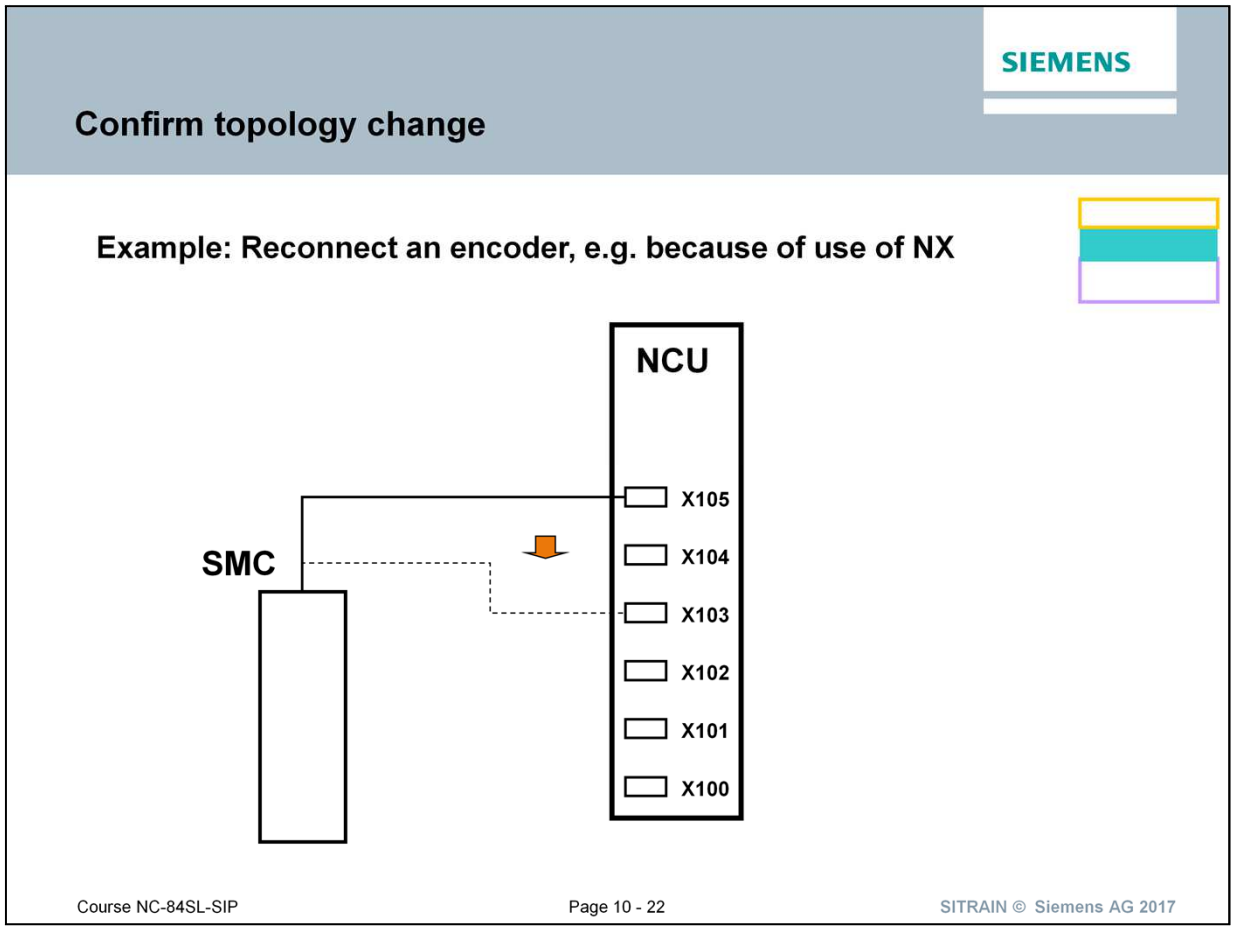
  - p9909 = 0 Manual acknowledgment
    The new serial number must be accepted manually by the user (p9905->1):

Manual acknowledgment

If p9909 = 0 is set (deviating from the default value), every module exchange must be acknowledged by setting p9905 = 1:

Sequence:

p9 = 1
p9905 = 1 Accept serial number
p9905 -> 0 (automatic): Acknowledgment OK
p9 = 0



Acknowledge differences p9904

If only one fault occurred during the comparison of the actual topology and the target topology that can be acknowledged, this parameter can be used to start a new comparison with acknowledgment of the fault in the target topology.

Acknowledgeable

Errors that can be acknowledged are:

- Topology comparison, component shifted
- Topology comparison shows one component that has a different serial number
- Topology comparison shows one component that is connected differently

Sequence

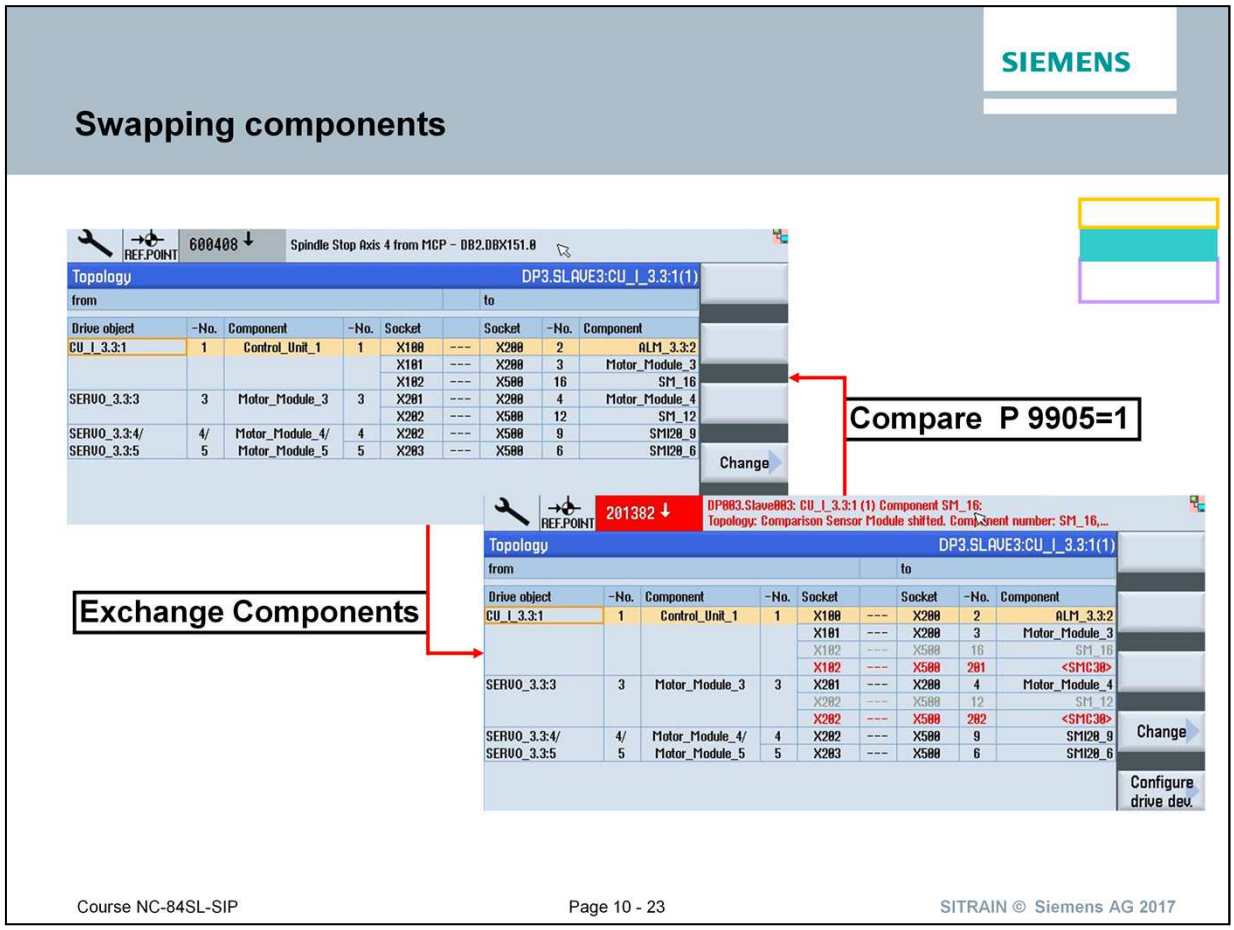
p9 = 1 Enable parameter change

p9904 = 1 -> The operation is started.

p9904 = 0 After starting -> The operation has been completed successfully.

p9904 > 1 After starting -> The operation has not been completed successfully.

p9 = 0 Normal operation



Swapping components

Swap within a drive device (e.g. for troubleshooting).
The setting in p9906 is relevant in this case.

Comparison quality p9906

p9906 can be set to the following values for topology comparison quality:

0 = High:	Compares the entire electronic rating plate
1 = Medium:	Compares the component type and the order number
2 = Low:	Compares the component type
3 = Minimum:	Compares the component class (e.g. Sensor Module or Motor Module)

Acknowledgment

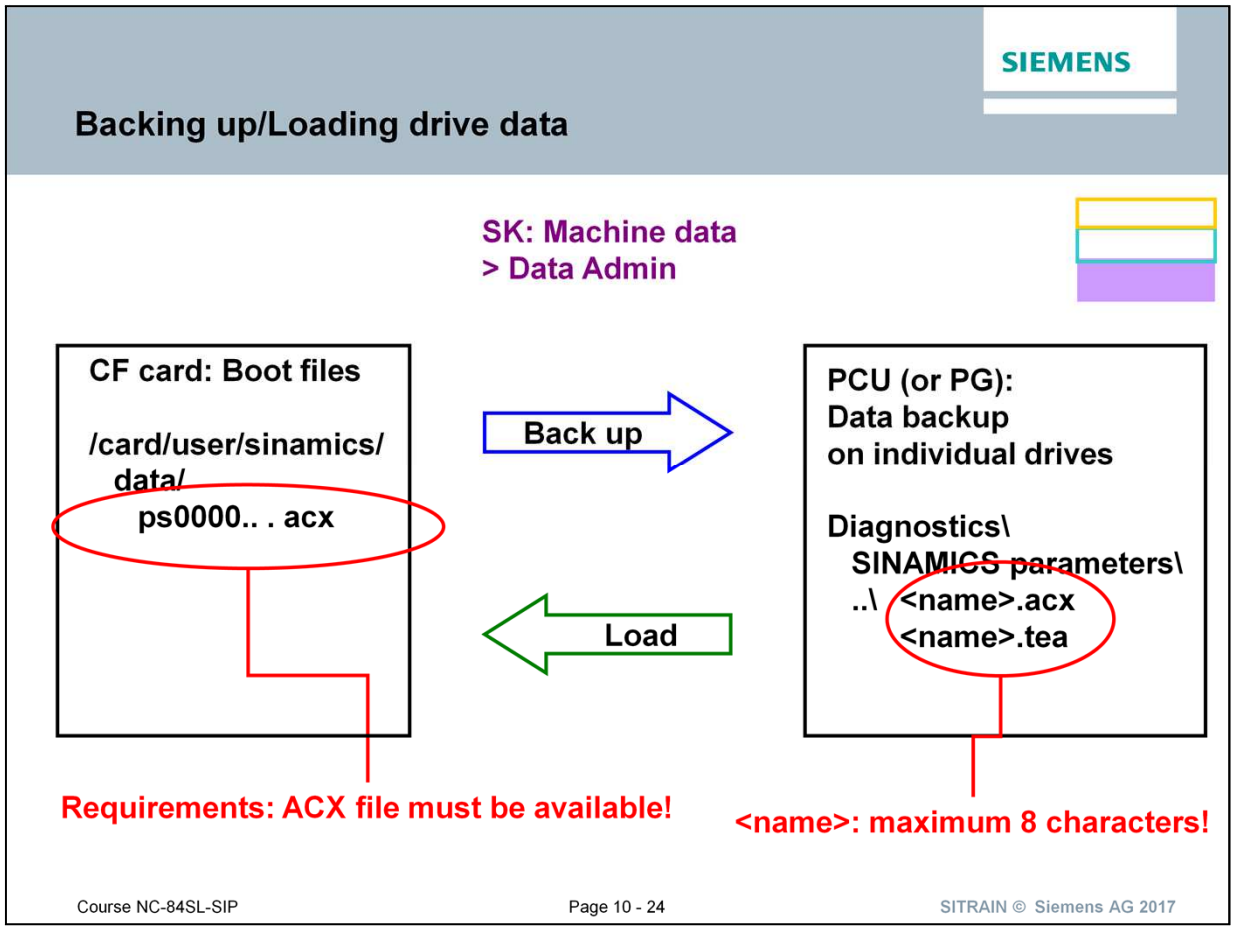
The exchange is acknowledged as follows:

CU/NX:	p9 = 1	Enable parameter changes
	p9906 = 2	Topology comparison quality low
	p9905 = 1	Transfer serial number and version to setpoint topology
	p9905 -> 0	(automatic): Acknowledges the transfer
	p9906 = 0	Topology comparison quality high
	p9=0	Normal operation

Do not forget to save!

Example:

Diagnostics whether an SMC module or the encoder is faulty. The DRIVE CLiQ connector on two neighboring SMCs and the encoder connector are swapped and an acknowledgment is generated with p9905.



Function

Selecting

OA: Setup -> SK: Machine data -> SK Data Admin

takes you to the menu where you can back up and load data.

In the case of drive data, the following requirements must be fulfilled:

Requirements:

- **Boot file available**

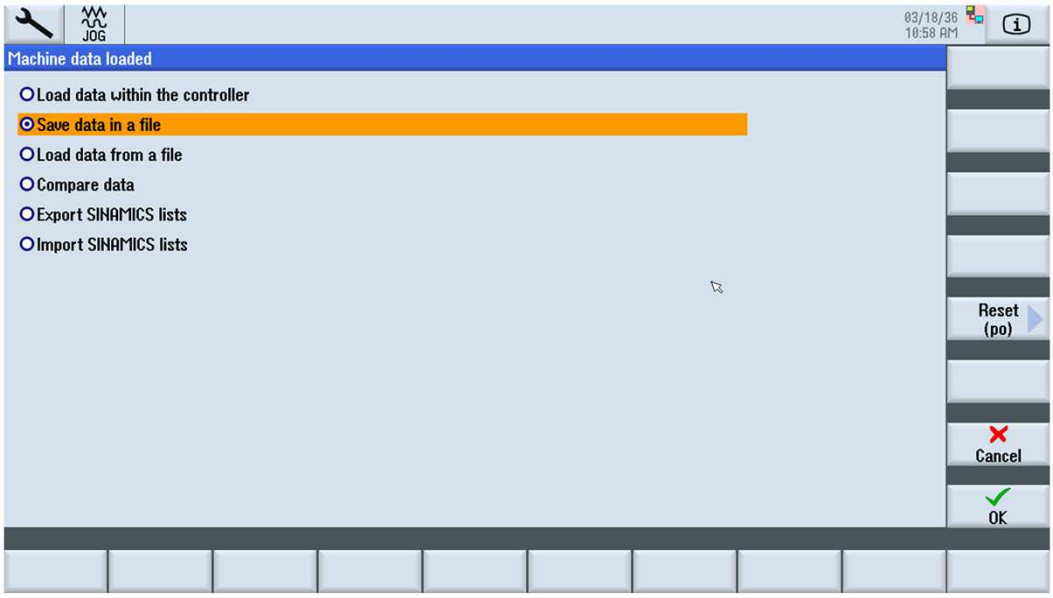
  Drive data can only be backed up or loaded if a corresponding ACX file (boot file) is available on the CF card.

  This means that: Automatic device configuration must have been carried out!

- **Max. 8 characters**

  The file name is restricted to a maximum of 8 characters because of the ACX file.

Backing up/Loading drive data
Backing up drive data



 SITRAIN © Siemens AG 2017

Backing up

Using **Save data in a file** you branch into the menu to back up data.

Procedure

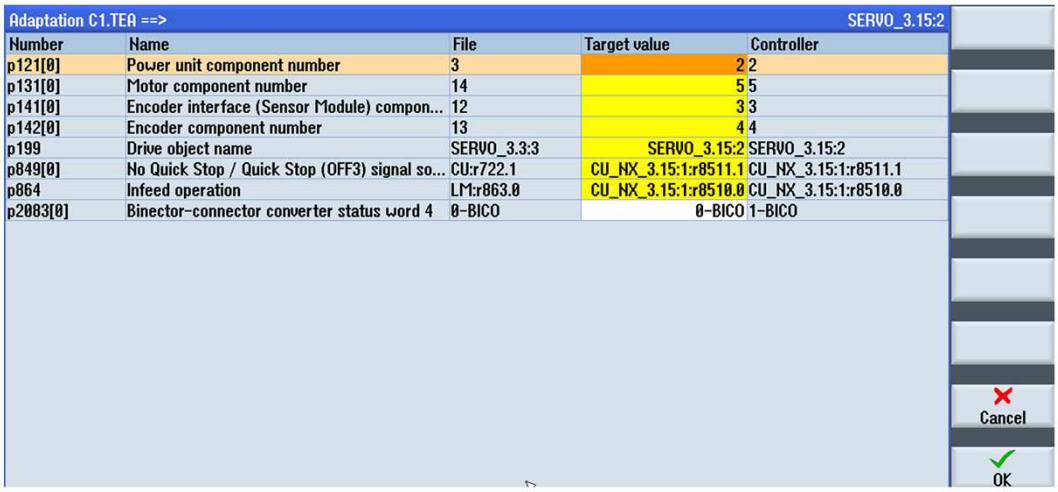
1. Select the required drive in SINAMICS parameters - Drive parameters
2. As target directory, either the recommended one should be kept, or if necessary, data can be saved to an USB_storage medium.
3. Assign filename xx....., and then the SK “OK“

Result

File xx……..TEA is created (ASCII file with drive data)
File xx.....ACX is created (file with parameter structure of the backed-up data)
File xx.....LOG is created (LOG file, no entry in the file)
Files are stored in

HMI data/Backup data/IB data/SINAMICS parameters/Drive parameters.

Backing up/Loading drive data
Loading/Adapting drive data



 SITRAIN © Siemens AG 2017

Load

Complete drive data files from “third-party” drives can be loaded.
It goes without saying that this only makes sense for identical power units, motors and encoders.

Load drive data

To load a drive data file after selecting the source directory, select the TEA file. In the next step, you are prompted to specify the drive in which the file should be loaded.

Setup archive

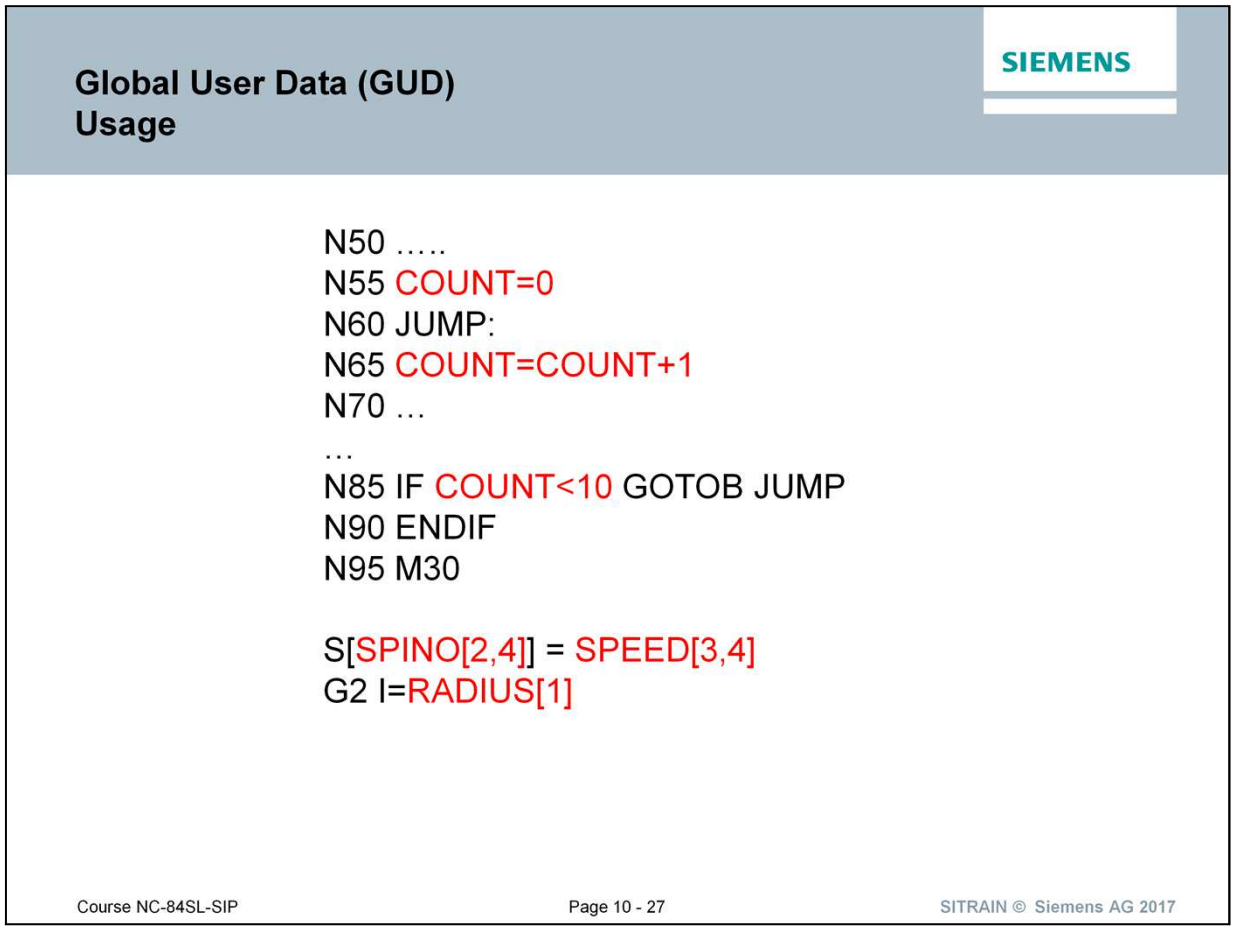
After acknowledging, you are told that you should not carry out the load operation without a setup archive being available. Of course you must take this comment seriously.

Adapting data

If data are different in the source file and target drive data - for example, component numbers - then these must be edited before loading using the **SK: ‘Adapt…’**

Generally, values are taken from the drive, and not from the file.

.



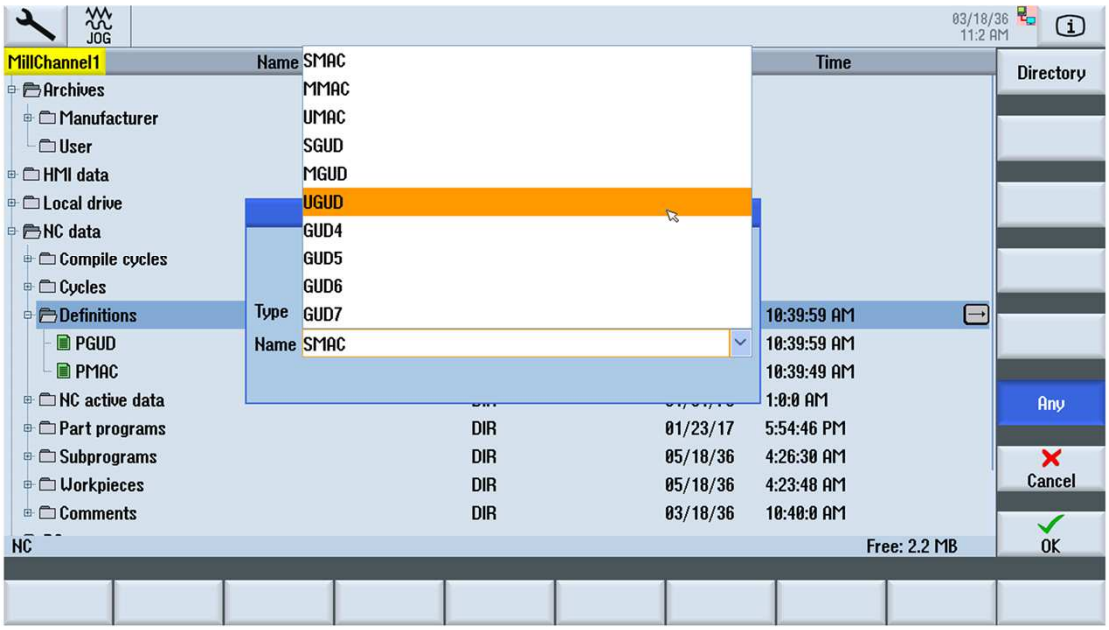
Usage

GUD are variables with a similar usage like R parameters.
They have to be defined in specific definition files.

Of course, like R parameters they live in buffered memory,
that‘s why their values are remembered even with a Power Off / Power On of the machine.

Global User Data (GUD)
Definition

SIEMENS



 SITRAIN © Siemens AG 2017

Definition

For defining GUD You have to find the correct path first:

> **OA: ‚Setup' > SK: ‚System data' >** directory: **NC data / Definitions**

Here, You can create new GUD definition files: MGUD, UGUD, …
(and macro files MMAC, UMAC, …)

The following file names are reserved:

File name	Meaning
MGUD.DEF	Definitions for global data (machine manufacturer)
UGUD.DEF	Definitions for global data (machine user)
GUD4-GUD9.DEF	Further user definition files

Activation

After creating a new or editing an already present definition file,
You can activate it.

This happens when You close the file,
then the control asks „Do You want to activate the definitions of …?"

and You press the **> SK: Activate**

Global User Data (GUD) Definition Syntax

SIEMENS

```
MillChannel1 NC/DEF.DIR/UGUD.DEF
DEF NCK INT LLI 1 ULI 12 Count=1¶
DEF CHAN REAL RADIUS[5]¶
DEF NCK INT SPINO[4, 5]¶
DEF CHAN REAL SPEED[4, 5]¶
M30¶
```

Value	Symbol	Meaning
Range	NCK	NC global user variable
	CHAN	Channel global user variable
Type	INT	Ganzzahliger Wert mit Vorzeichen
	REAL	Real number (LONG REAL according to IEEE)
	BOOL	Logical value TRUE (1) / FALSE (0)
	CHAR	ASCII code 0 … 255
	STRING<MaxLength>	Character string of size MaxLength
	AXIS	Axis / Spindle
	FRAME	Frame settings (static coordinate transformation)
Limits	LLI <value>	Lower limit
	ULI <value>	Upper limit
Name	Name of the variable	
[size_1,… size_3]:	Field sizes 1- up to max. 3- dimensional	Size_j = Number of components for each dimension
Init_Value	Initial value	Initial value

Course NC-84SL-SIP Page 10 - 29 SITRAIN © Siemens AG 2017

Definition of variables

- **GUD** DEF <Range> <Type> [<Limits>] <Name> [<Size_1> [,Size_2 [, <Size_3>]]] [=<Init_value>]

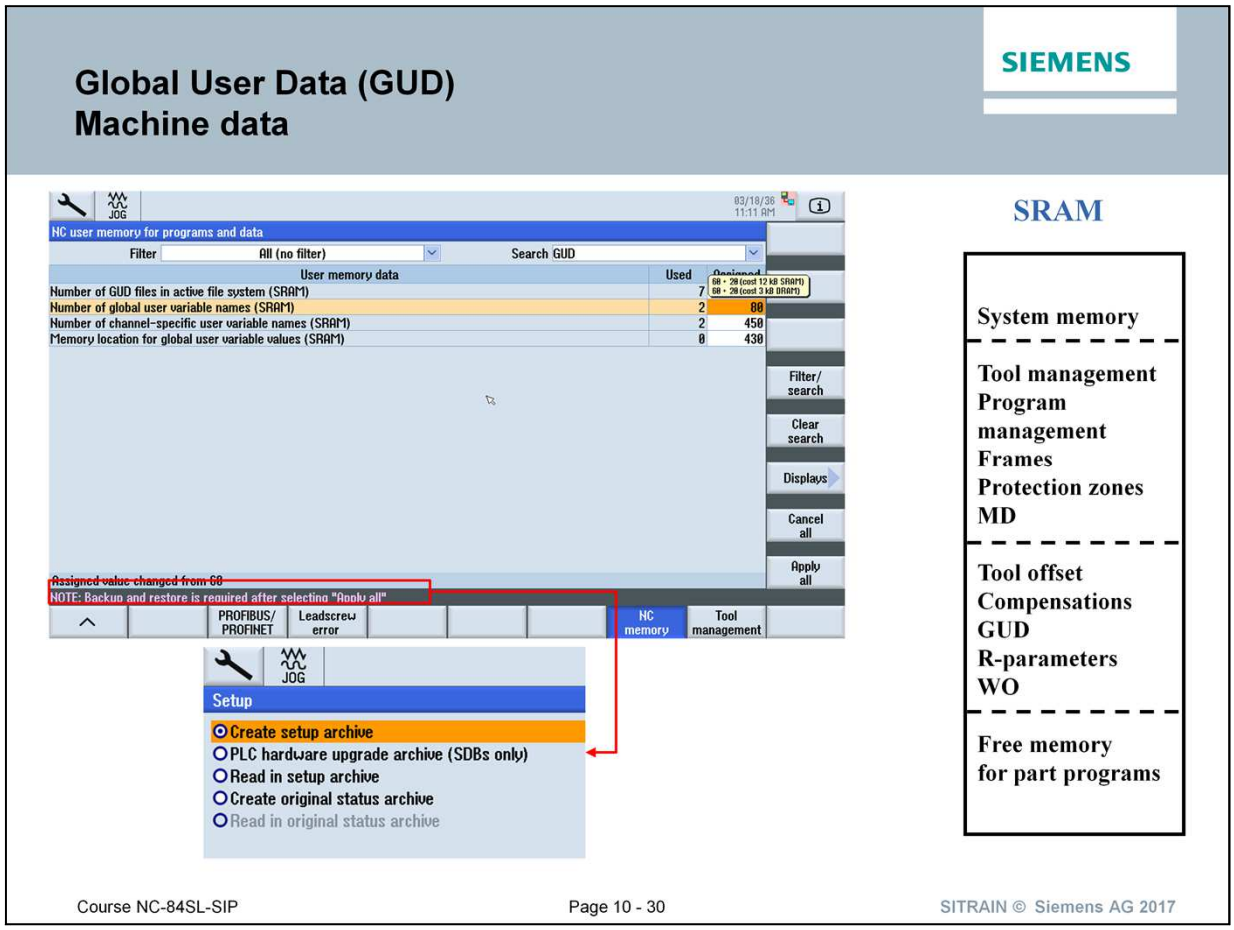
  where

  - Range = range of validity (NCK / CHAN)
  - Type = type of variable
  - Limits = specification of lower / upper limits
  - Size = Number of field components for each dimension (max 3 dimensions)
  - Init_value = initial value

  Specifications in square brackets […] are optional

Remark Much the same way one can define local user variables (LUD) directly in a NC program. The difference is just that the Range specification is omitted, so

- **LUD** DEF <type> <limits> <name>[<Size_1>, <Size_3>, =<Init_value>]



Machine data

For GUD memory has to be reserved in SRAM of the NCK.

This is done by supplying the machine data

MD18118
MD18120
MD18130
MD18150

SRAM memory

The machine data can be entered in the menu

> OA: ‚Setup' > SK: ‚NC' > SK: ‚NC memory'

Setting the filter „Global user data (GUD)" you can find the MD easier.

A change of these machine data has to be confirmed by

> SK: ‚Apply all'

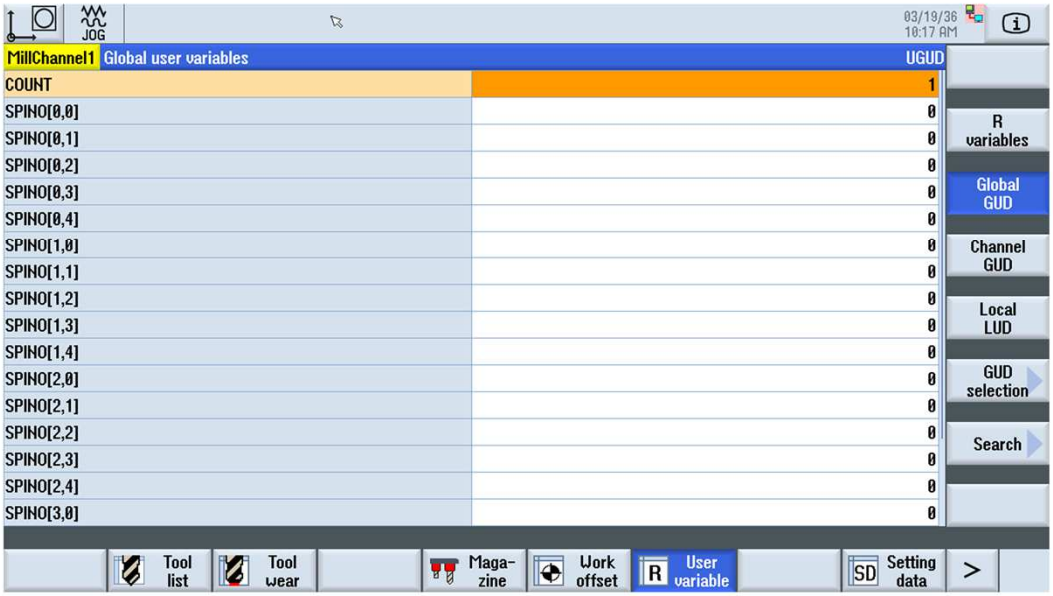
Then, in order not to lose any part programs and other data like actual values of R parameters, the SINUMERIK Operate selects the screen ‚Setup archive'

There, You have to create a setup archive (NC data) and read in this archive afterwards.

(Of course, this is the usual procedure when changing machine data which affect memory configuration in SRAM)

Global User Data (GUD) Display

SIEMENS



 SITRAIN © Siemens AG 2017

Display

The GUD variables can be viewed and changed in the menu

> OA: ‚Parameter' > SK: ‚User variable'

and then either:

> SK: ‚Global GUD' for global variables (DEF NCK …)

or

> SK: ‚Channel GUD' for channel specific variables (DEF CHAN …)

Basic settings

An initial state is the state of an NC function (e.g. G codes, tool length compensation, transformation, coupled motion, etc.) that is adopted when a channel is in a certain state.

Channel states for which initial states can be parameterized are:

- Control run-up (NC reset)
- Reset (channel or mode group reset)
- Part program start
- Part program end

The initial state of an NC function is retained until it is explicitly changed by means of operation or programming actions.

SITRAIN © Siemens AG 2017

Notes

G groups

Group	Explanation
1	Modal motion commands (e.g. G0, G1, G2, G3)
2	Non-modal motions, dwell time (e.g. G4, G74, G75)
3	Programmable offsets, working area limitations, and pole programming (e.g. TRANS, ROT, G25, G110)
6	Plane selection (G17, G18, G19)
7	Tool radius compensation (e.g. G40, G42)
8	Settable work offset (e.g. G54, G57, G500)
9	Offset suppression (e.g. SUPA, G53)
10	Exact stop - continuous-path mode (e.g. G60, G641)
13	Workpiece dimensioning inches/metric (e.g. G70, G700)
14	Workpiece dimensioning, absolute dimension/incremental dimension (G90/G91)
15	Feedrate type (e.g. G93, G961, G972)
16	Feedrate override on inside and outside curvature (e.g. CFC)
21	Acceleration profile (e.g. SOFT, DRIVE)
22	Tool offset types (e.g. CUT2D, CUT2DF)
29	Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF)
30	Compressor ON/OFF (e.g. COMPOF)

SITRAIN © Siemens AG 2017

The G functions are divided into function groups. It is only possible to write one G function in a group in a block: For example, you cannot have both G1 and G2 in a block.
A G function can be either modal (until it is canceled by another function of the same group) or only effective for the block in which it is programmed (non-modal).

Note You will find a list of G function groups in the appendix to this chapter.

G code machine data

MD 20150 [0..69] $MC_GCODE_RESET_VALUES
- Default setting of the G groups

MD 20152 [0..69] $MC_GCODE_RESET_MODE
- Reset behavior of G groups

MD 20110 $MC_RESET_MODE_MASK =4041h(milling) / 4071h(turning)
- Bit 0 = 1 G code acc. to MD 20150
  Bit 6 = 1 Tool length compensation retained beyond reset
  Bit 14 = 1 Basic frame retained beyond reset
  Bit 16 = 0 Initial state for master spindle acc. to $MC_SPIND_DEF_MASTER_SPIND

MD 20112 $MC_START_MODE_MASK
- Definition of control properties on part program start

SITRAIN © Siemens AG 2017

The relevant initial states are parameterized via the following machine data:

MD20110 $MC_RESET_MODE_MASK ("Definition of the initial control state in case of reset")

MD20112 $MC_START_MODE_MASK ("Definition of the initial control state at NC start")
Bit = 1 -> Default (MD 20150)

MD20150 $MC_GCODE_RESET_VALUES ("Delete setting of G codes"). Evaluation only with MD20110 bit 0=1.

MD20152 $MC_GCODE_RESET_MODE ("G code initial state in case of reset") This MD is only evaluated in MD20110 if bit 0 is set. This MD is used to define, for each entry in MD20150 $MC_GCODE_RESET_VALUES (i.e. for each G group), whether a setting according to MD20150 $MC_GCODE_RESET_VALUES is undertaken again for a reset/part program end (MD = 0) or whether the current setting is retained (MD = 1).

The control-system response after:	can be set with:
Power-up (POWER ON) *)	MD20110 $MC_RESET_MODE_MASK (Definition of control default settings after reset/parts program end) MD20144 $MC_TRAFO_MODE_MASK (Selection of the kinematic transformation function) MD20150 $MC_GCODE_RESET_VALUES (initial setting of the G groups)
RESET/part program end	MD20110 $MC_RESET_MODE_MASK (Definition of control default settings after reset/parts program end) MD20150 $MC_GCODE_RESET_VALUES (initial setting of the G groups) MD20152 $MC_GCODE_RESET_MODE (Definition of control initial setting after reset)
Part program start	MD20112 $MC_START_MODE_MASK (definition of initial control system settings after part program start) MD20110 $MC_RESET_MODE_MASK (Definition of control default settings after reset/parts program end)

PLC interface G functions

DB21 - DB30	Signals from the channel (NCK → PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB208	Number of the active G function of G function group 1 (8 bit INT) /Z1-K1/							
DBB209	Number of the active G function of G function group 2 (8 bit INT) /Z1-K1/							
DBB210	Number of the active G function of G function group 3 (8 bit INT) /Z1-K1/							
DBB211	Number of the active G function of G function group 4 (8 bit INT) /Z1-K1/							
DBB212	Number of the active G function of G function group 5 (8 bit INT) /Z1-K1/							
DBB213	Number of the active G function of G function group 6 (8 bit INT) /Z1-K1/							
DBB214	Number of the active G function of G function group 7 (8 bit INT) /Z1-K1/							
DBB215	Number of the active G function of G function group 8 (8 bit INT) /Z1-K1/							
...	...							
DBB270	Number of the active G function of G function group n-1 (8 bit INT) /Z1-K1/							
DBB271	Number of the active G function of G function group n (8 bit INT) /Z1-K1/							

64 G groups

Course NC-84SL-SIP Page 10 - 35 SITRAIN © Siemens AG 2017

Note

The active G functions of the groups are updated for each programming of a G function or a mnemonic identifier (e.g. SPLINE).

G functions within a G group are output as a dual value, starting with 1.
A G function with the value 0 means that for this G group, no G function is active.

G codes
Initial states of selected G functions (standard)

SIEMENS

No. in the G function display is 1 higher

MD 20150 $MC_GCODE_RESET_VALUES[...]

Delete setting of the G groups (extract)

[0]	1 = G0,	2 = G1 (def)	
[5]	1 = G17 (def),	2 = G18,	3 = G19
[7]	1 = G500 (def),	2 = G54,	3 = G55, 4 = G56, 5 = G57
[9]	1 = G60 (def),	2 = G 64,	3 = G641
[11]	1 = G601 (def),	2 = G602,	3 = G603
[12]	1 = G70,	2 = G71 (def)	
[13]	1 = G90 (def),	2 = G91	
[14]	1 = G93,	2 = G94 (def),	3 = G95
[20]	1 = BRISK (def),	2 = SOFT	
[22]	1 = CDOF (def),	2 = CDON	
[23]	1 = FFWOF (def),	2 = FFWON	
[28]	1 = DIAMOF (def),	2 = DIAMON	

The activation time point of the G codes depends on MD 20110 and MD 20112.

SITRAIN © Siemens AG 2017

Notes

Exercise: NX start-up HW Config

Objective To practice PLC start-up using an NX10 module.

Steps

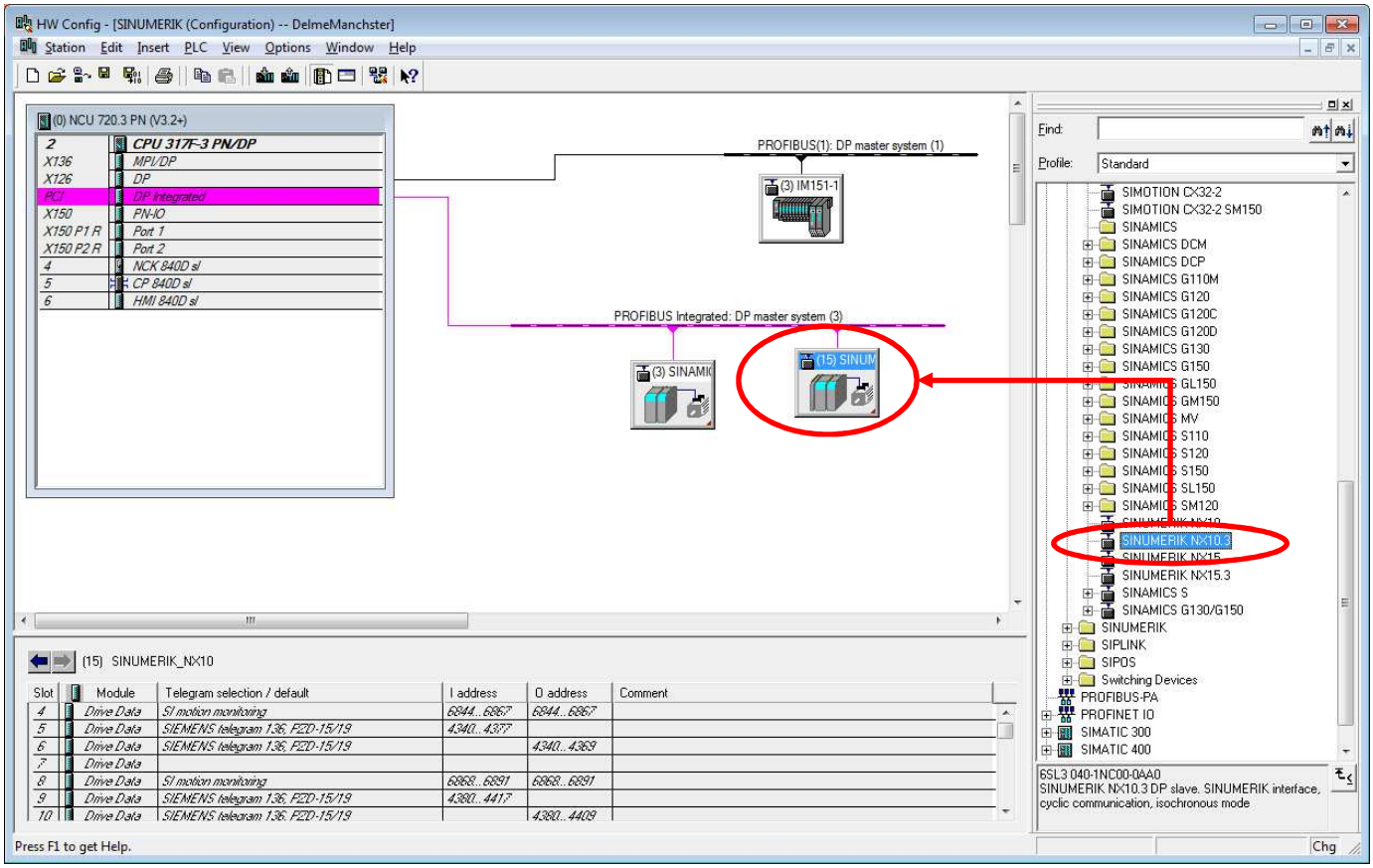
1. Drag NX10 from the catalog to the project.
2. Confirm the DP slave properties.
3. Confirm the DRIVE-CLiQ wiring information
4. Load HW Config

Sequence Since the NX is connected to the internal PROFIBUS, it must be added in the HW Config.

1. NX10.3 from catalog

The corresponding GSD file is stored under PROFIBUS-DP → SINAMICS

Left-click to select the "SINUMERIK NX ..." module and drag it to the "PROFIBUS Integrated DP master system" segment in the "Station design" station window.

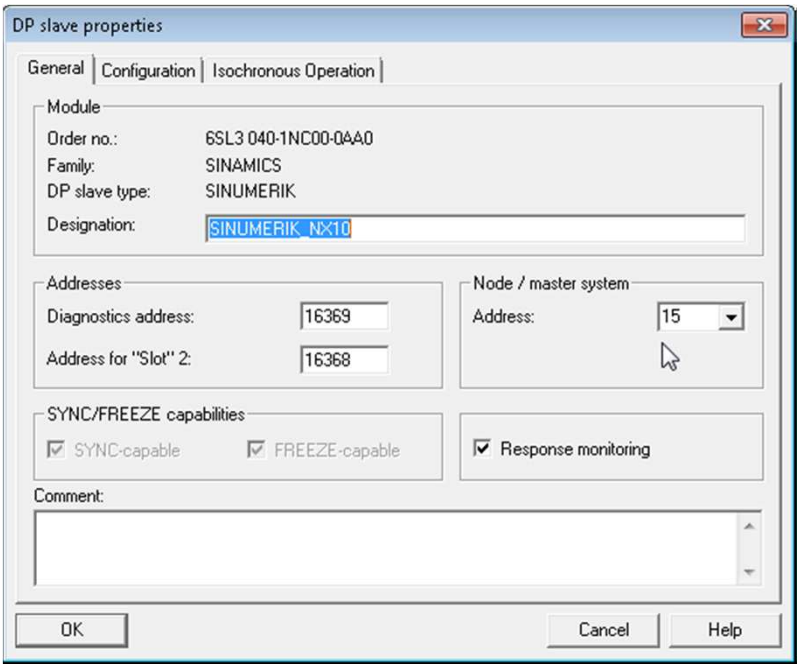


2. DP slave ...

The "DP slave properties" dialog box opens.

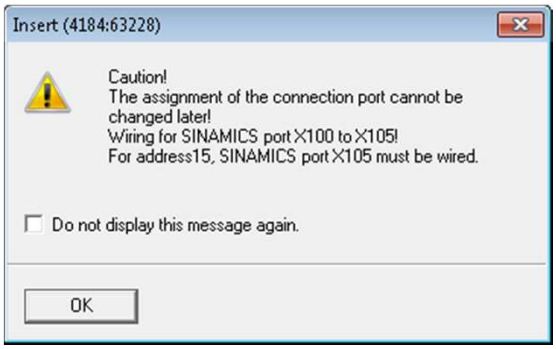
Enter the address for the integrated PROFIBUS in this dialog box. "15" is the default setting for the first NX.

We recommend using this address.



3. DRIVE-CLiQ ...

Click "OK" to confirm the note about the DRIVE-CLiQ wiring (SINAMICS port X105 on the NCU):



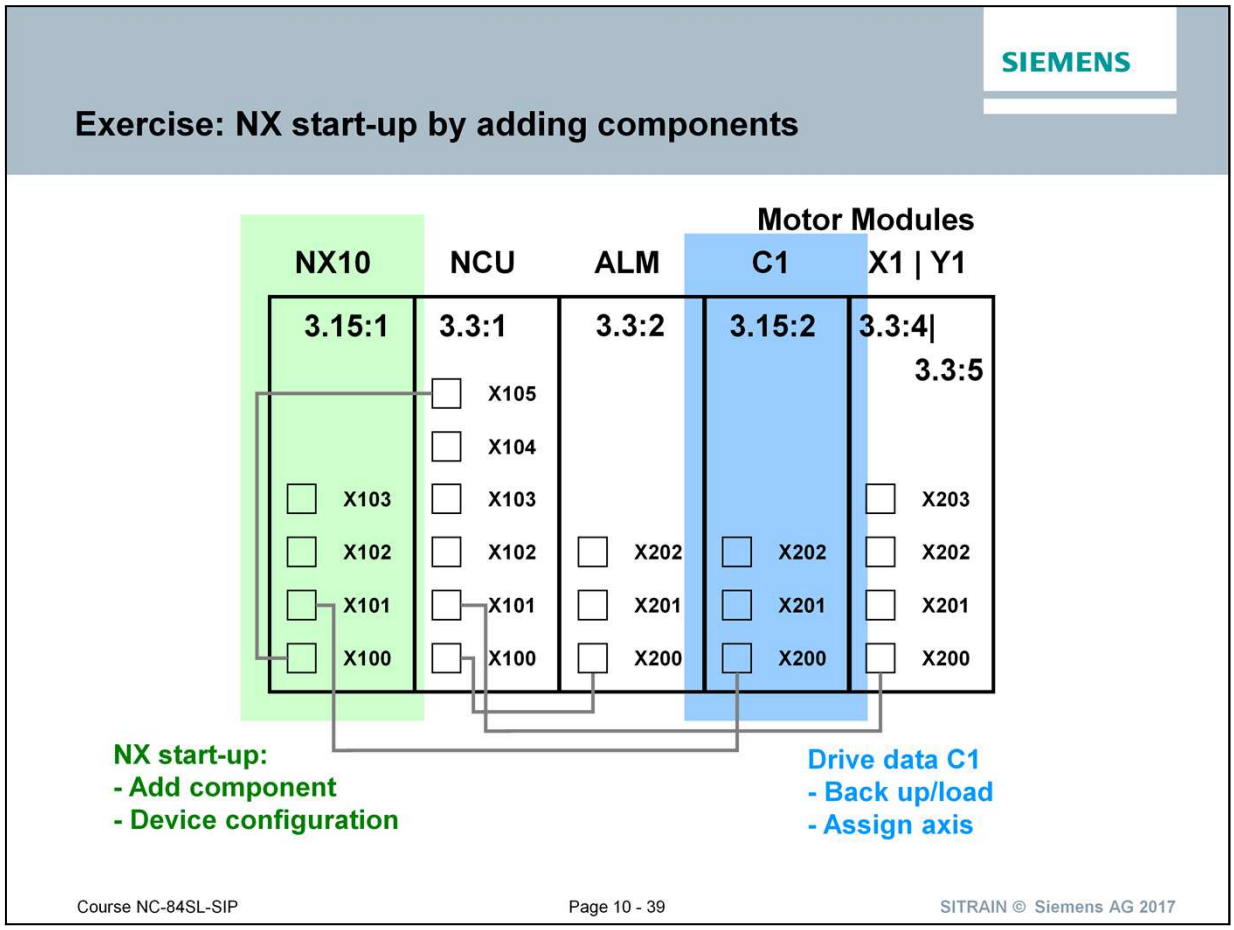
DRIVE-CLiQ cable

Remember that the DRIVE-CLiQ connection between the CU and the NX must be wired from the corresponding connector on the CU (**X105** for PROFIBUS address **15**) to **X100** on the NX. If there are more NX modules, the addresses and slots are specified accordingly in the Step 7 HW Config.

4. Load HW Config

Save and download the HW Config as generated.

5. SK: Reset (po)



Objective

This exercise provides an opportunity to practice starting up an NX with the "Add component" function.

An axis (C1) is appended to the NX module.

Its drive data (e.g. motor data, optimization, etc.) should be saved with the file functions and then loaded again once the topology has been changed.

Functions

The exercise practices the following functions:

- Back up/load drive data with changed component number
- Delete object
- Add component
- Device start-up NX
- Assign axis

Requirements

The PLC HW Config exercise must have been completed.

Steps

1. Back up drive data C1
2. Remove Single Motor Module
3. Acknowledgment: Delete object
4. Connect NX10 to Single Motor Module
5. Acknowledgment: Add component
6. Device configuration with NX and Servo_3.15:2
7. Load drive data C1
8. Assign axis C1 to Servo_3.15:2
9. Test: Traverse axes
10. Back up data

Alternative

Alternatively, a device configuration of the fully wired system can be carried out. However, to do this, you must write data to all drives and assign the axes.

1. Back up drive data C1

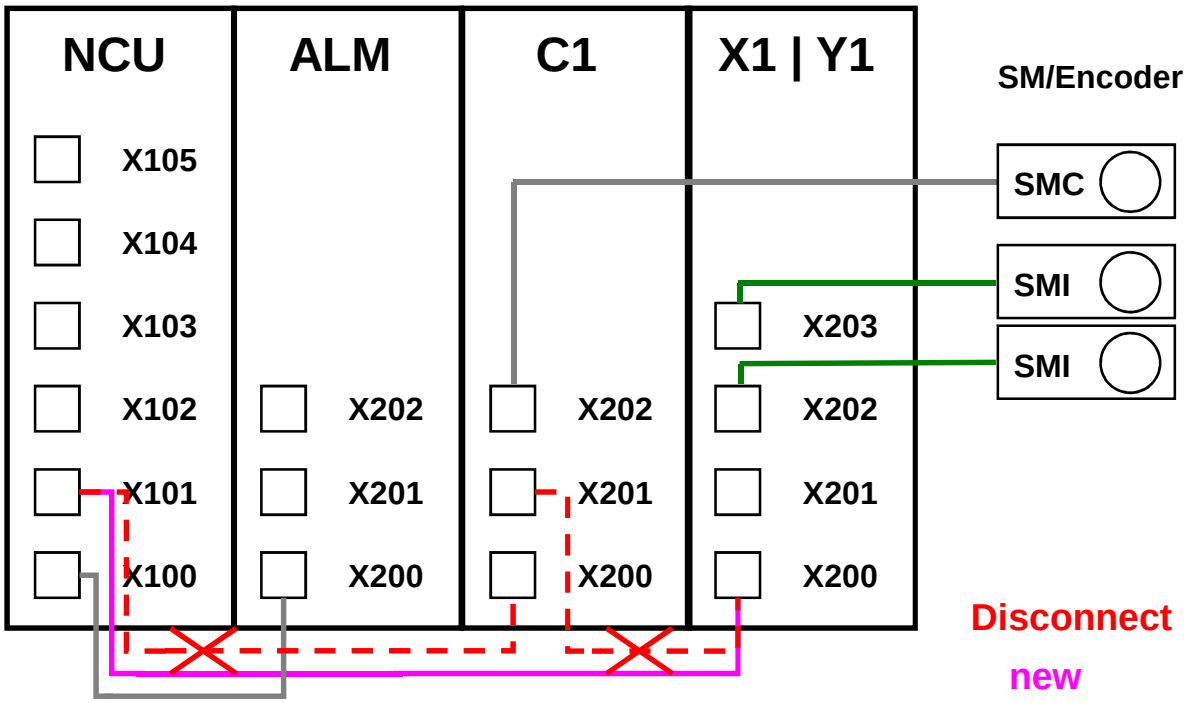
Back up the drive data of the C1 axis in menu:

> OA: 'Setup' > SK: 'Machine data' > SK: 'Data admin'

> Option button: **„Save data in a file"**

2. Remove Single Motor Module

Remove the Single Motor Module of the C1 axis from the DRIVE-CLiQ line-up:



3. Delete object

After SK: Reset (po), acknowledge this change with the "Delete object" function

> OA: 'Setup' > SK: 'Drive system'

> SK: 'Drive device' > SK: 'Topology' > SK: 'Change'

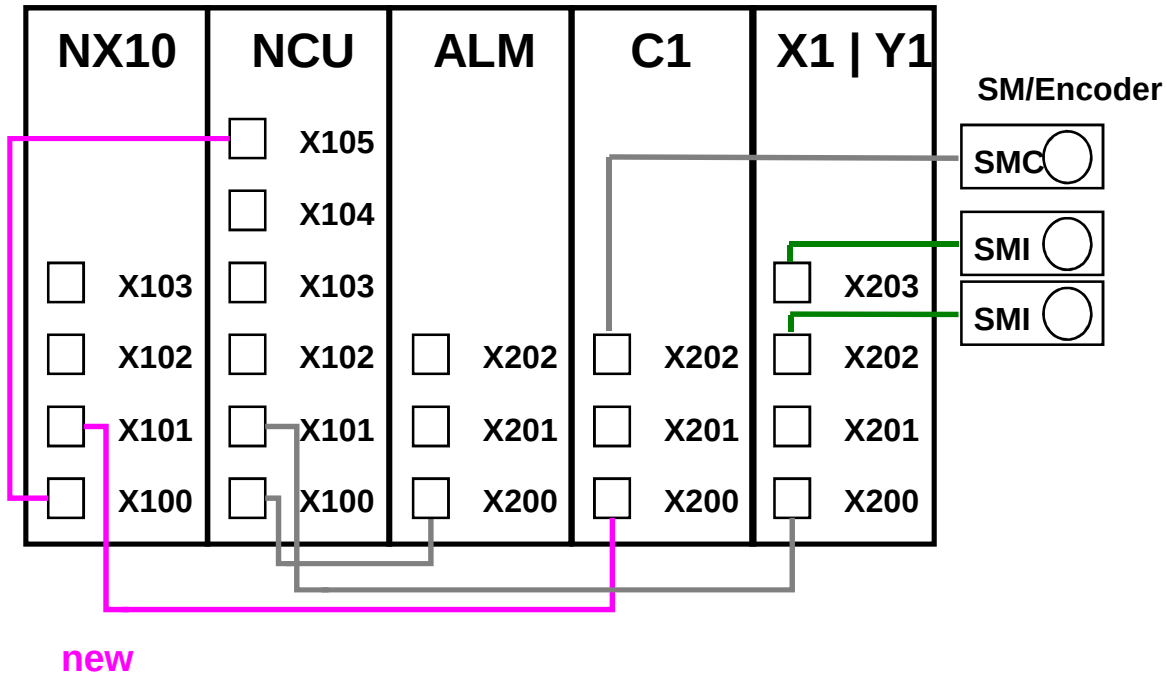
Select object **Servo_3.3:3.**

Acknowledge with **SK: 'Delete drive obj.'**

Click **SK: 'OK'** to confirm all dialog boxes.

4. Connect NX10

Connect the NX10 module with the Single Motor Module of the C1 axis to the NCU:



5. Add component

Alarm 201416 "Component also in actual topology" is displayed.

Acknowledge this change with the "Add component" function:

> OA: ‘Setup’ > SK: ‘Drive system’
> SK: ‘Drive device’ > SK: ‘Topology’ > SK: ‘Add component’

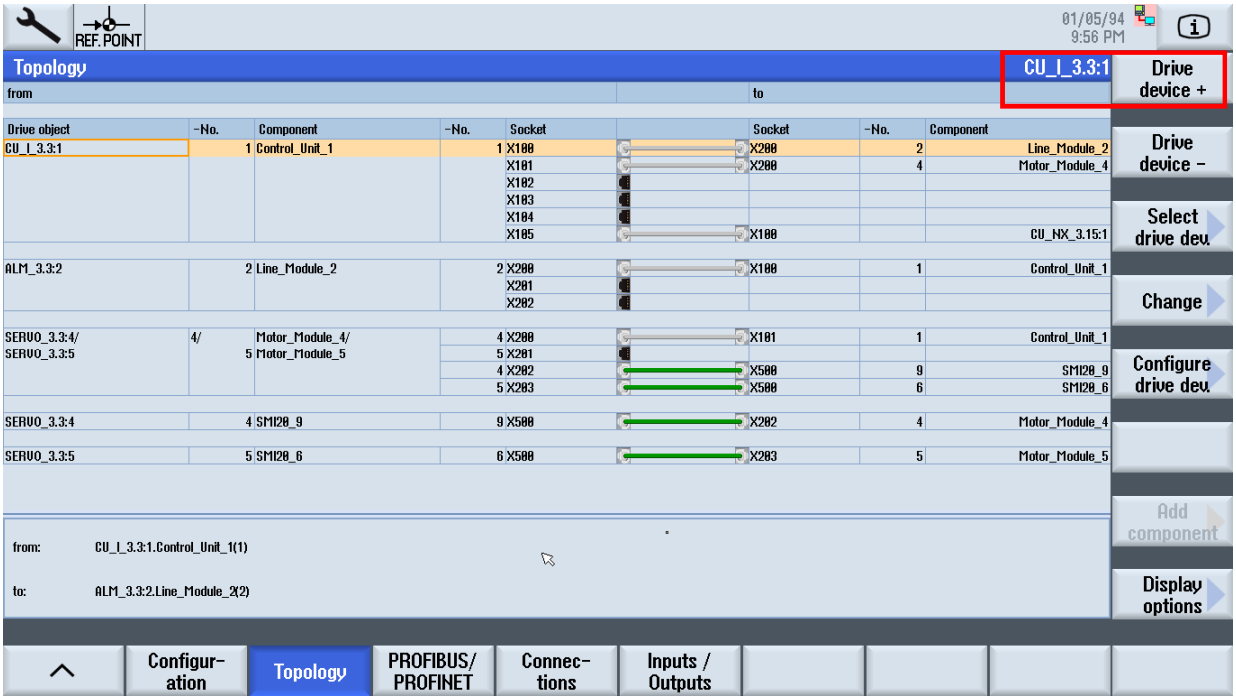
Next, the NX10 is located and added.
Do this by confirming the subsequent dialog box with "OK":

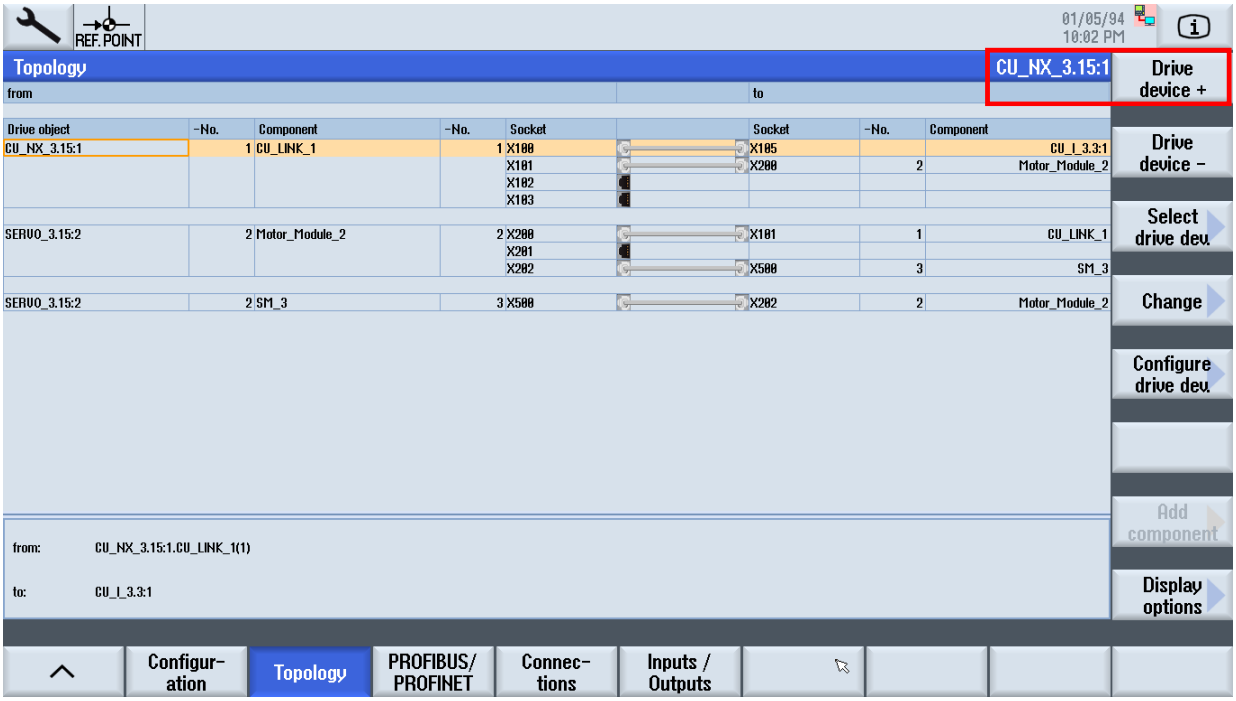
6. Device configuration NX

Next, the fact that the NX has not undergone device configuration, is detected and the user is prompted to do this;

Here too, you should acknowledge the series of dialog boxes up to power ON.

The result of steps 2 … 6 should look like this::



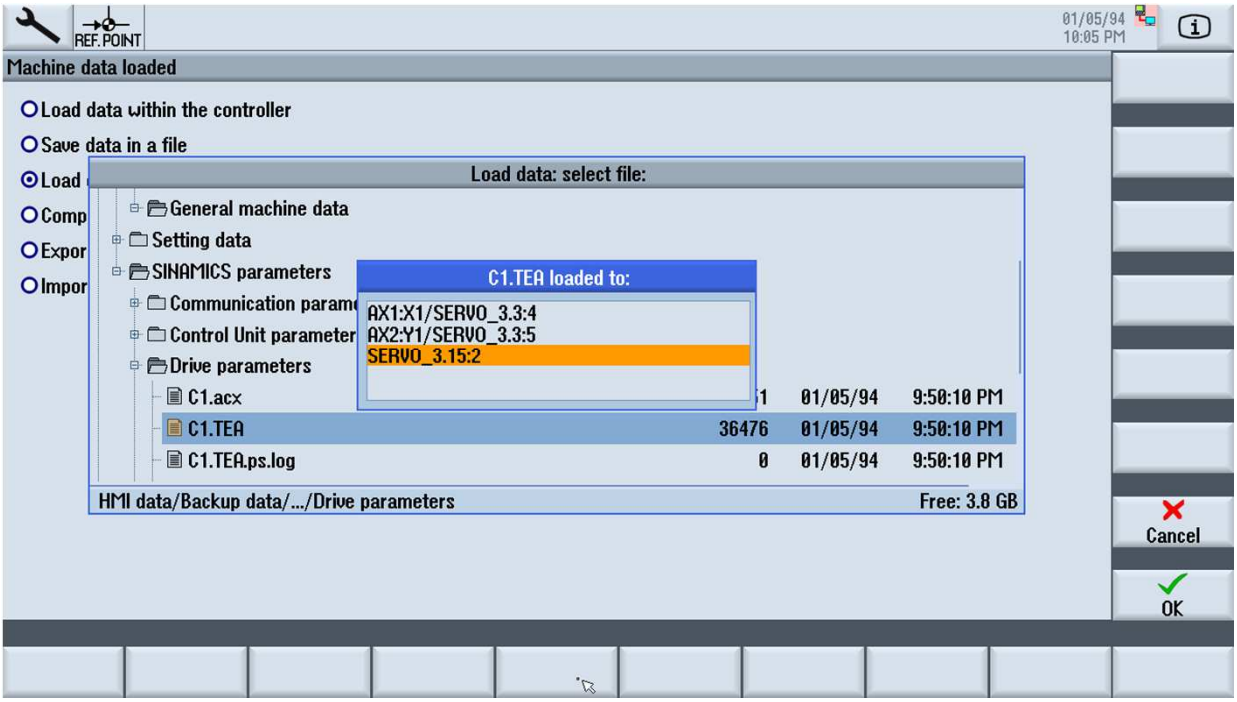


7. Load optimized drive data C1

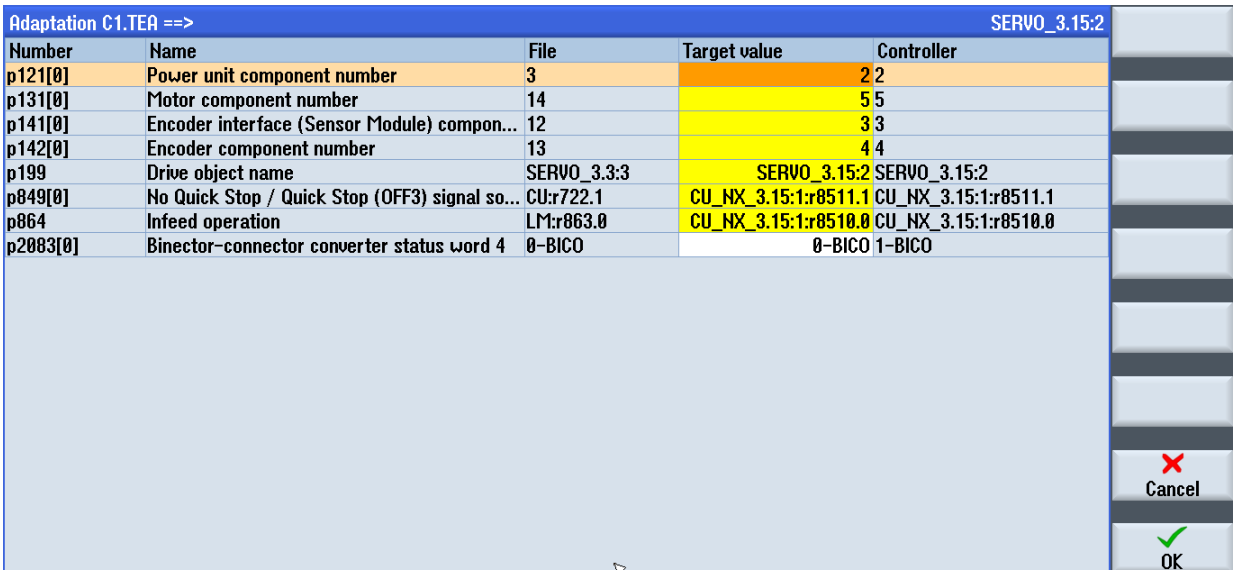
Load the backed-up optimized drive data of the C1 axis in the menu:

OA: ‘Setup’ > SK: ‘Machine data’ > SK: ‘Data admin’

Selecting **SINAMICS drive parameters,** choose the backup files C1 and load it in Servo_3.15:2



The control detects changes to the component numbers (for drive Servo_3.15:2)
Confirm the message that appears subsequently with **SK: ‘Adapt’**:
Confirm the proposed changes to the component numbers:



8. Assign axis

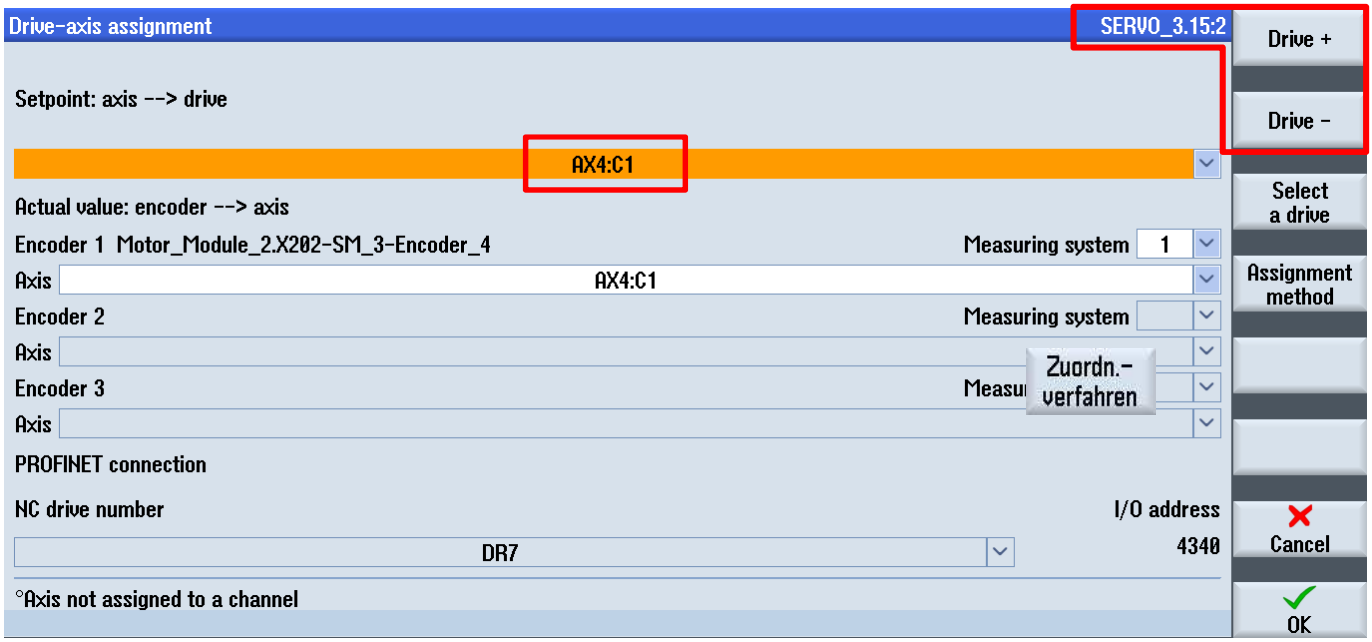
All that remains to be done is to connect the axis **C1** to the drive **Servo_3.15:2**.

Use the "Assign axis" function to do this:

> OA: 'Setup' > SK: 'Drive system'
> SK: 'Drives' > SK: 'Axis assignment'



Use **SK: 'Drive +' / 'Drive –'**
to select **Servo_3.15:2.**



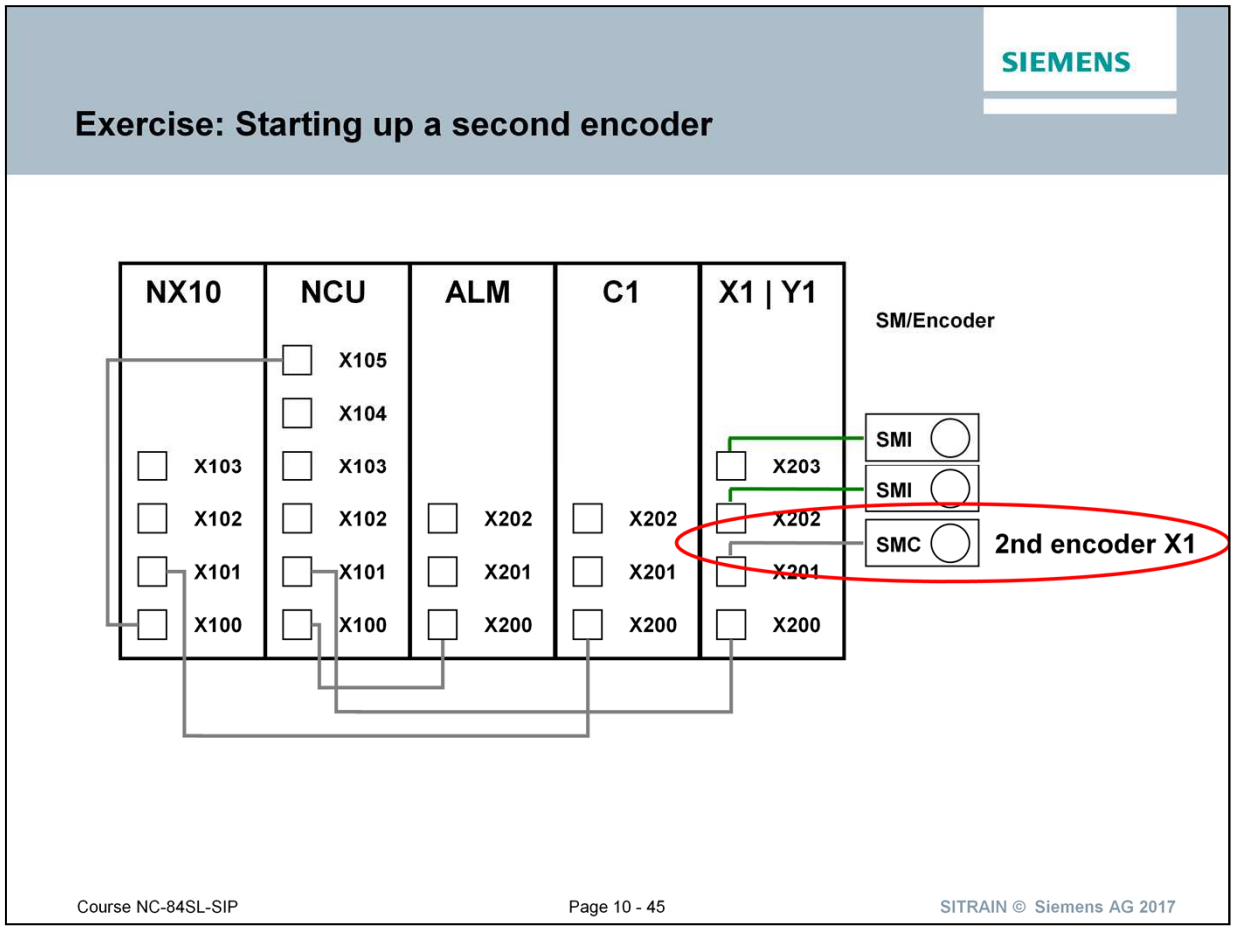
The selection is complete after selecting the C axis from the pulldown menu and confirming with **> SK: ‚OK'**

9. Test

Test the start-up after power ON by traversing all axes.

10. Back up data

Back up the drive data as a DP archive



Background

In machine tools it is often the case that a measuring system mounted directly on the axis to be traversed, and not a motor encoder, is responsible for position control of an axis. The advantage of this approach is that interplay (from gears, for example) no longer affects axis positioning.

Objective

This exercise will show how to integrate a second encoder system of this nature into the control and how to carry out start-up.

Functions

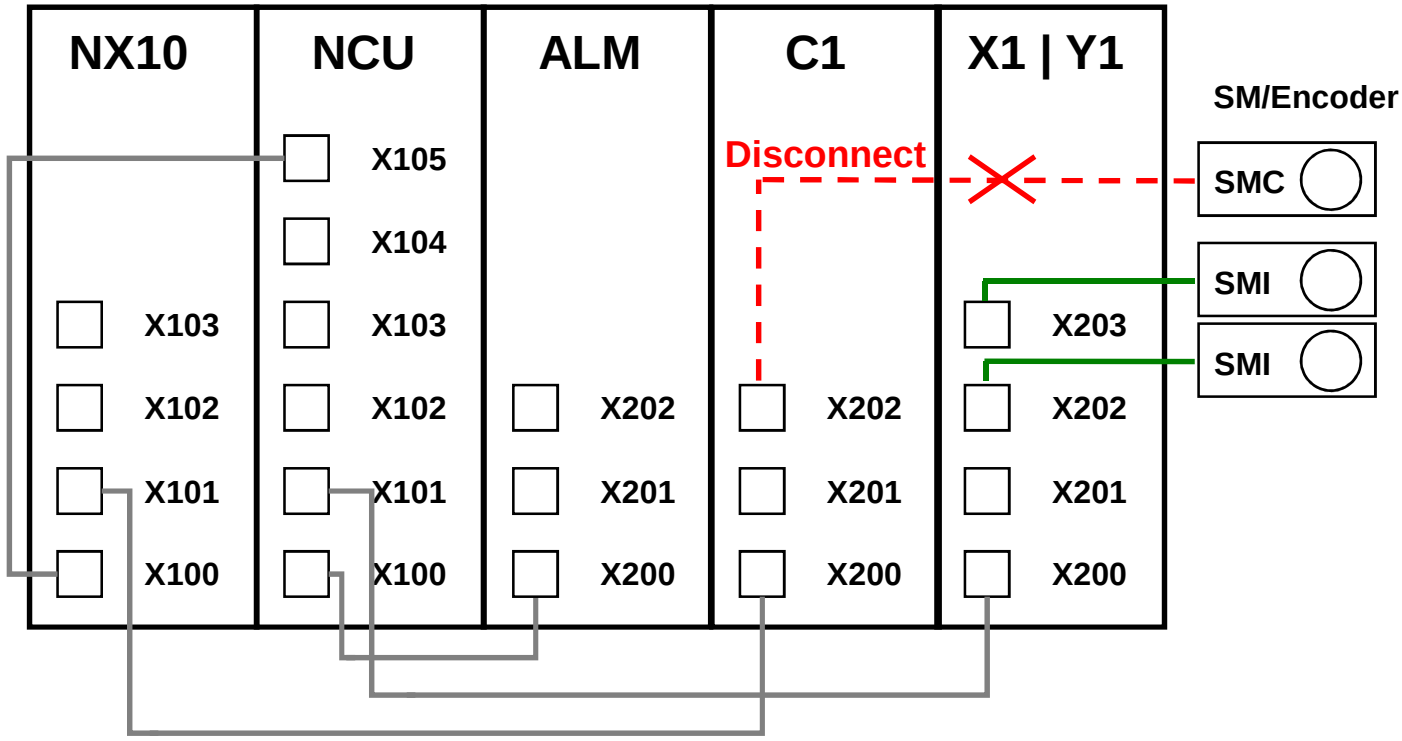
The exercise practices the following functions:

- Back up/load drive data with changed component number
- Delete object
- Add component
- Device start-up NX
- Assign axis

Steps

1. Remove SMC with encoder from axis C1
2. Acknowledgment: Delete SMC component
3. Connect SMC with encoder
4. Acknowledgment: Add component
5. Assign SMC and encoder to Servo_3.3:4 (X1) as second encoder
6. Assign axis: Assign encoder to axis X1 as second encoder
7. Optional in PLC: Program change-over encoder 1 <-> 2
8. Test: Monitor actual values

1. Remove SMC Remove the DRIVE-CLiQ connection to the SMC (and encoder) from axis C1:



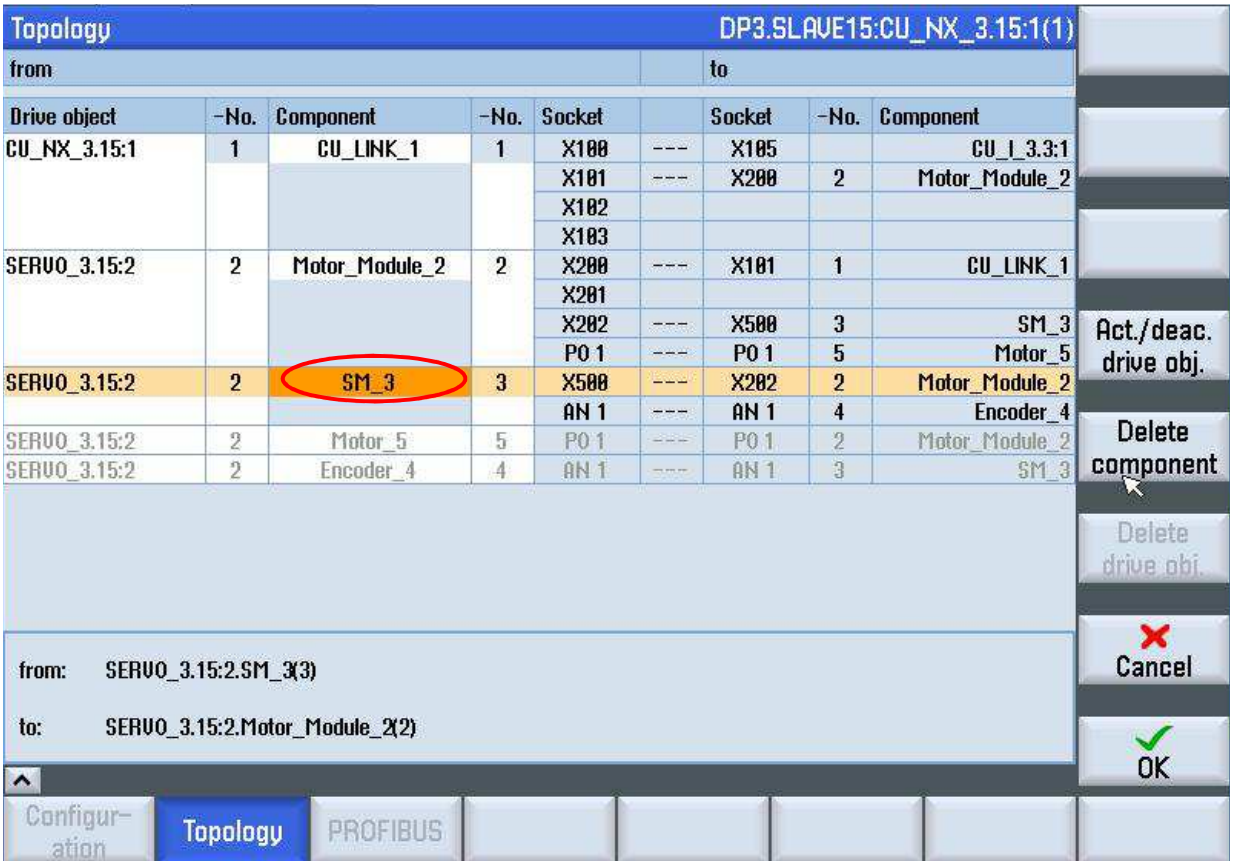
2. Delete component

Acknowledge this change with the "Delete component" function:

> OA: 'Setup' > SK: 'Drive system'
> SK: 'Drive device' > SK: 'Topology' > SK: 'Change'

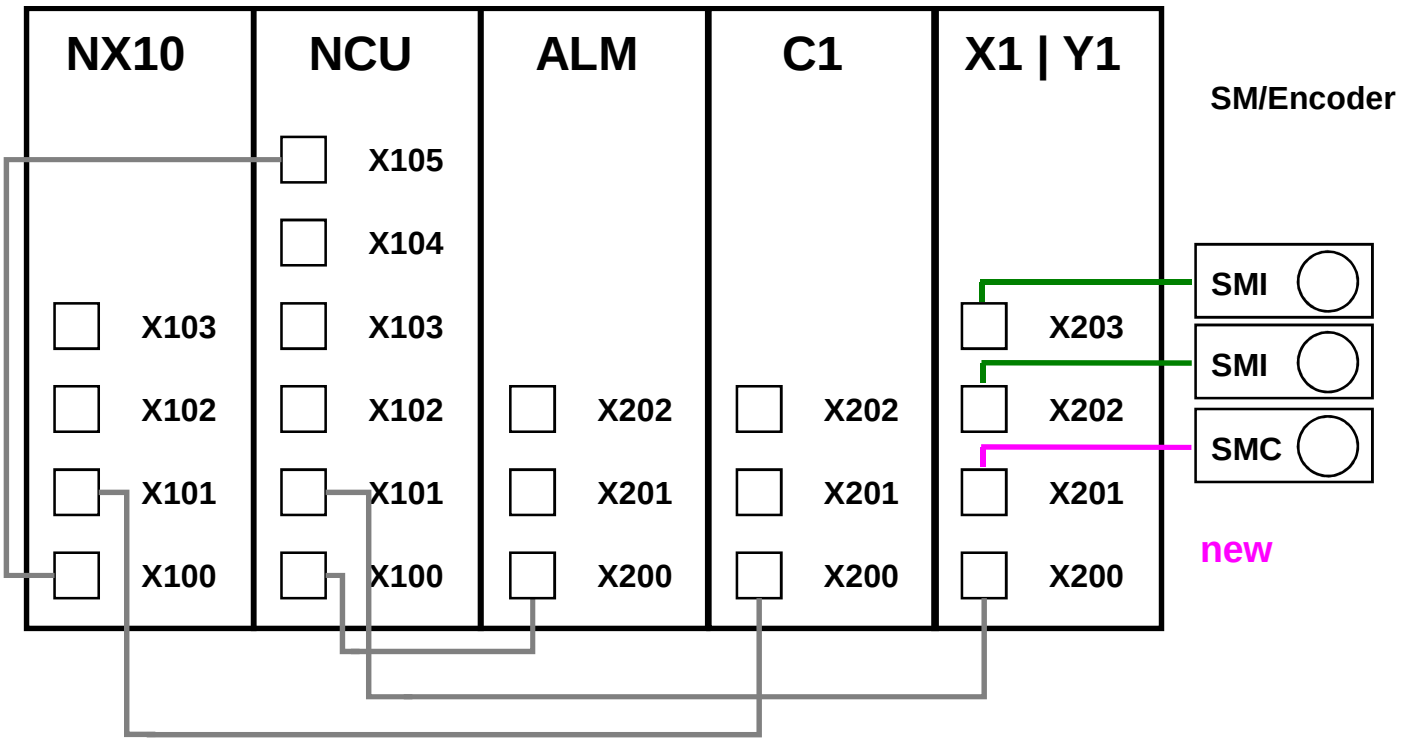
Select component **Servo_3.15:2 SM_3**

> SK: 'Delete component'



3. Connect SMC with encoder

Connect the Drive CLiQ cable of the SMC to the Double Motor Module port X201:



4. Add component

Alarm 201416 "… additional component in actual topology…" is displayed.

Acknowledge this topology change with the "Add component" function:

> OA: 'Setup' > SK: 'Drive system'
> SK: 'Drive device' > SK: 'Topology' > SK: 'Add component'

The result is fault 201331 " … component not assigned to a drive object."

5. Assign SMC and encoder to servo (X1)

Next, assign the SMC and the encoder to Servo_3.3:4 (X1) as a 2nd encoder.

To do this, select the "Change" function for Servo_3.3:4 (X1).

Proceed as follows:

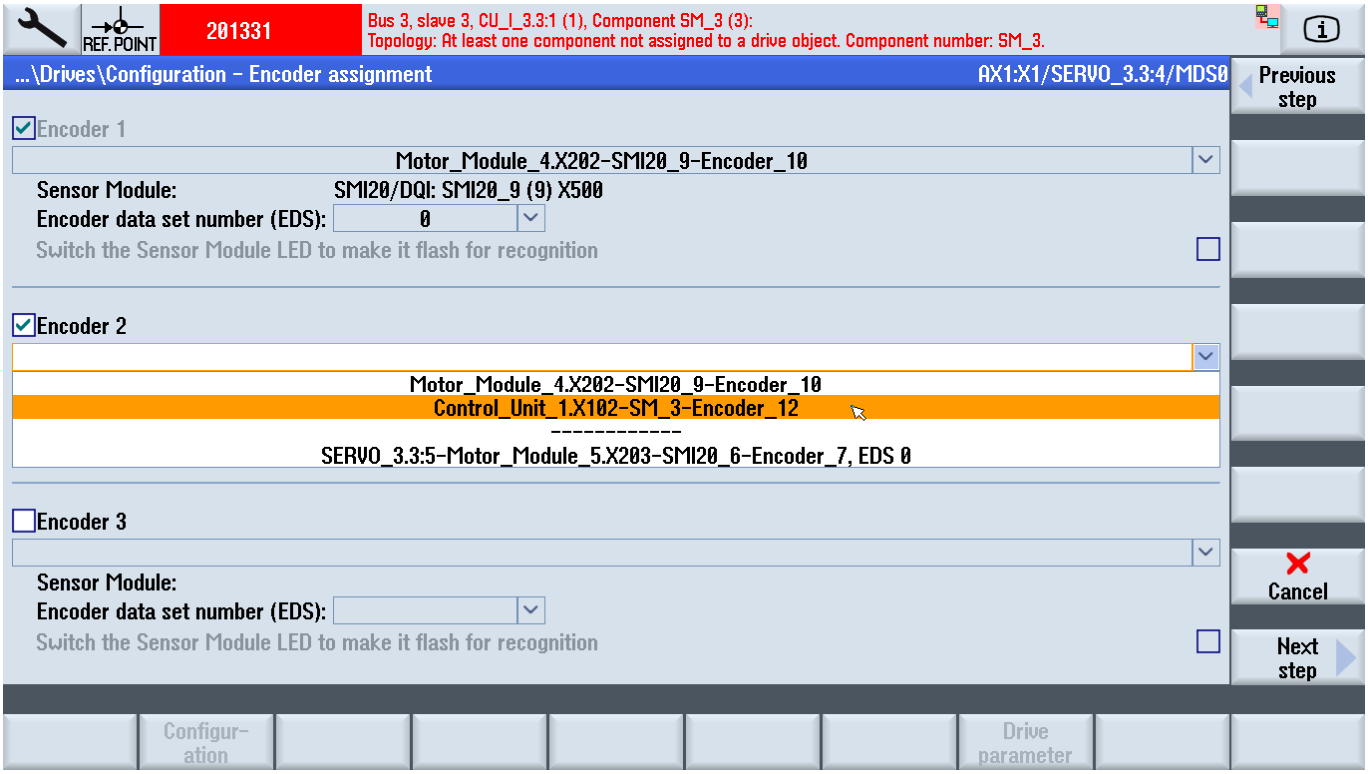
> OA: 'Setup' > SK: 'Drive system'
> SK: 'Drives'

Select Servo_3.3:4 (X1) with **SK: Drive + / Drive -**

> SK: 'Change'

The sequence starts with the components detected automatically,
e.g. Motor Module, motor, and brake. Confirm these with **SK: 'Next'**

In the encoder screen form, select the second encoder SM_13.Encoder_14 from the list.



After again pressing **SK: 'Next step'** select **Encoder 2** from the list.
We have the encoder type with **code number 3001**.

All additional settings involving the object are not changed; at the end, a summary of the object data is displayed, which must include the following values:

Drive: Drive object name: SERVO_3.3:4 Motor Module: Type (Order No.): 6SL3120-2TE15-0Axx Rated power: 2.70 kW Rated current: 5.00 A Code number: 10012 Component number: 4 Serial number: T-B82018847 Motor: Motor name: Motor_SMI_11 No motor holding brake available Motor type: 1FK7 synchronous motor Type (Order No.): 1FK7044-xAF7x-xxxx Encoder1: Encoder name: Encoder_10 Sensor Module: SMI20/DQI: SMI20_9 (9) X500 Serial No. SMx: T-B62010208 Serial No. encoder: - Encoder type: 2048, 1 Vpp, A/B C/D R Code number: 2001	Encoder2: Encoder name: Encoder_14 Sensor Module: SMC30: SM_13 (13) X500 Serial No. SMx: T-BN2068986 Serial No. encoder: - Encoder type: 1024 HTL A/B R Code number: 3001 Control mode/setpoints: Control mode: closed-loop speed control (with encoder) PROFIBUS PZD telegram: SIEMENS telegram 136, PZD-15/19 Function modules:

6. Assign axis

On the NC side, the second encoder is activated with the "Assign axis" function.

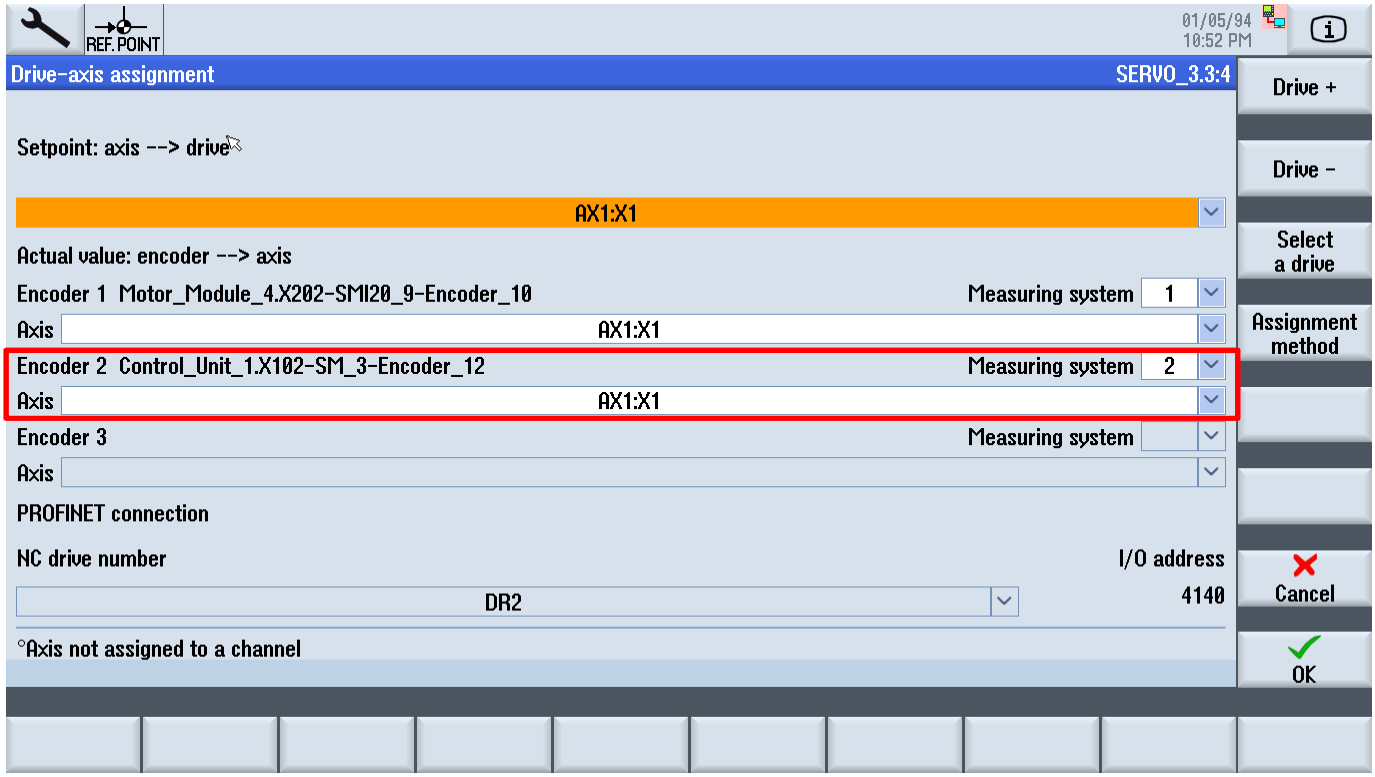
To do this, select the menu:

> OA: 'Setup' >SK: 'Drive system'
> SK: 'Drives' > SK: 'Assign axis'

Select drive Servo_3.3:3 (X1)

Press **SK: 'Change'**

Acknowledge with **SK: 'Accept'**



Used to configure a number of machine data items for specific axes.
You can of course check the values:

MD number	Name	Value
MD 30110	CTRL_OUT_MODULE_NR	1
MD 30220 [0/1]	ENC_MODULE_NR	1
MD 30200	NUM_ENCS	2
MD 30230 [0]	ENC_INPUT_NR	1
MD 30230 [1]	ENC_INPUT_NR	2
MD 30240 [0]	ENC_TYPE	1
MD 30240 [1]	ENC_TYPE	1
MD 31020 [0]	ENC_RESOL	2048
MD 31020 [1]	ENC_RESOL	1024
MD 34102	REFP_SYNC_ENCS	1

7. Optional: PLC encoder switchover

Switchover of the encoder to the position encoder used for the position control system takes place via the axis-specific PLC interface DB31..61.DBX 1.5/1.6

DB31-61.DBX1.5 = 1 Motor measuring system

DB31-61.DBX1.6 = 1 Direct measuring system

If both bits have the value "1", the motor measuring system is used.
If both bits have the value "0", the position control loop is open and the axis is parked.

8. Test When the C1 axis is traversed, the second encoder actual value X1 should change.

- Notice **So that the C1 axis can be traversed, you must change the following MD manually (!):**

MD 30200 = 0 NUM_ENCS

MD 30240 [0] = 0 ENC_TYPE

Encoder synchronization

To facilitate change-over between measuring systems in active operation, a limit value ENC_CHANGE_TOL relating to synchronization must not be exceeded.

This is achieved by setting MD 34102 = 1 REFP_SYNC_ENCS.

This causes the reference point value of the active position measuring system to be transferred to the non-active measuring system during referencing.
As a result, the two encoders are synchronized.

To stop this from happening, set

MD 30242 ENC_IS_INDEPENDENT = 1.

Appendix



SITRAIN © Siemens AG 2017

Notes

Appendix
G groups

SIEMENS

Group 1: Modally valid motion commands						
G function	No. [1]	Meaning	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G0	1.	Rapid traverse	+	m		
G1	2.	Linear interpolation (linear interpolation)	+	m	x	
G2	3.	Circular interpolation clockwise	+	m		
G3	4.	Circular interpolation counterclockwise	+	m		
CIP	5.	Circular interpolation through intermediate point	+	m		
ASPLINE	6.	Akima spline	+	m		
BSPLINE	7.	B-spline	+	m		
CSPLINE	8.	Cubic spline	+	m		
POLY	9.	Polynomial interpolation	+	m		
G33	10.	Thread cutting with constant lead	+	m		
G331	11.	Tapping	+	m		
G332	12.	Retraction (tapping)	+	m		
OEMIPO1 [5]	13.	Reserved	+	m		
OEMIPO2 [5]	14.	Reserved	+	m		
CT	15.	Circle with tangential transition	+	m		
G34	16.	Thread cutting with linear increasing lead	+	m		
G35	17.	Thread cutting with linear decreasing lead	+	m		
INVCW	18.	Involute interpolation clockwise	+	m		
INVCCW	19.	Involute interpolation counter-clockwise	+	m		
If no function from the group is programmed with modal G functions, the default setting, which can be changed in the machine data (MD20150 $MN_$MC_GCODE_RESET_VALUES), applies:						

Course NC-84SL-SIP Page 10 - 52 SITRAIN © Siemens AG 2017

Legend

1) **No.**

Internal number (e.g. for PLC interface): Configurability of the G function as a delete setting for the function group on run-up, reset or part program end (with MD20150 $MC_GCODE_RESET_VALUES): + configurable

2) **MD 20150**

Configurability of the G function as a delete setting for the function group on power up, reset or end of part program with

MD20150 $MC_GCODE_RESET_VALUES:

\+ Configurable

\- Not configurable

3) **W** Activity (German: **W**irksamkeit)

\- m modal

\- s non-modal (default setting).

If no function from the group is programmed with modal G functions, the default setting, which can be changed using the machine data (MD20150 $MN_$MC_GCODE_RESET_VALUES), applies.

4) **STD**: Standard settings

\- **SAG**: Siemens AG default setting

\- **MM**: Machine manufacturer's default setting
(please see machine manufacturer's specifications)

Group 2: Non-modally valid motions, dwell time						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G4	1.	Dwell time preset	-	n		
G63	2.	Tapping without synchronization	-	n		
G74	3.	Reference point approach with synchronization	-	n		
G75	4.	Fixed-point approach	-	n		
REPOSL	5.	Linear repositioning	-	n		
REPOSQ	6.	Repositioning in a quadrant	-	n		
REPOSH	7.	Repositioning in semicircle	-	n		
REPOSA	8.	Linear repositioning with all axes	-	n		
REPOSQA	9.	Linear repositioning with all axes, geometry axes in quadrant	-	n		
REPOSHA	10.	Repositioning with all axes; geometry axes in semicircle	-	n		
G147	11.	Approach contour with straight line	-	n		
G247	12.	Approach contour with quadrant	-	n		
G347	13.	Approach contour with semicircle	-	n		
G148	14.	Leave contour with straight line	-	n		
G248	15.	Leave contour with quadrant	-	n		
G348	16.	Leave contour with semicircle	-	n		
G5	17.	Oblique plunge-cut grinding	-	n		
G7	18.	Compensatory motion during oblique plunge-cut grinding	-	n		

Group 3: Programmable frame, working area limitation and pole programming						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
TRANS	1.	TRANSLATION: Programmable offset	-	n		
ROT	2.	ROTATION: Programmable rotation	-	n		
SCALE	3.	SCALE: Programmable scaling	-	n		
MIRROR	4.	MIRROR: Programmable mirroring	-	n		
ATRANS	5.	Additive TRANSLATION: Additive programmable offset	-	n		
AROT	6.	Additive ROTATION: Programmable rotation	-	n		
ASCALE	7.	Additive SCALE: Programmable scaling	-	n		
AMIRROR	8.	Additive MIRROR: Programmable mirroring	-	n		
	9.	Unassigned				
G25	10.	Minimum working area limitation/spindle speed limitation	-	n		
G26	11.	Maximum working area limitation/spindle speed limitation	-	n		
G110	12.	Pole programming relative to the last programmed setpoint position	-	n		
G111	13.	Polar programming relative to origin of current workpiece coordinate system	-	n		
G112	14.	Pole programming relative to the last valid pole	-	n		
G58	15.	Programmable offset, absolute axial substitution	-	n		
G59	16.	Programmable offset, additive axial substitution	-	n		
ROTS	17.	Rotation with solid angle	-	n		
AROTS	18.	Additive rotation with solid angle	-	n		

Group 4: FIFO						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
STARTFIFO	1.	Start FIFO Execute and simultaneously fill preprocessing memory	+	m	x	
STOPFIFO	2.	STOP FIFO, stop machining; fill preprocessing memory until STARTFIFO is detected, FIFO is full or end of program	+	m		
FIFOCTRL	3.	Activation of automatic preprocessing memory control	+	m		

Group 6: Plane selection						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G17	1.	Plane selection 1st - 2nd geometry axis	+	m	x	
G18	2.	Plane selection 3rd - 1st geometry axis	+	m		
G19	3.	Plane selection 2nd - 3rd geometry axis	+	m		

Group 7: Tool radius compensation						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G40	1.	No tool radius compensation	+	m	x	
G41	2.	Tool radius compensation left of contour	-	m		
G42	3.	Tool radius compensation right of contour	-	m		

Group 8: Settable zero offset						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G500	1.	Deactivation of adjustable work offset (G54 to G57, G505 to G599)	+	m	x	
G54	2.	1st Settable zero offset	+	m		
G55	3.	2nd adjustable work offset	+	m		
G56	4.	3rd adjustable work offset	+	m		
G57	5.	4th adjustable work offset	+	m		
G505	6.	5th adjustable work offset	+	m		
...	...	...	+	m		
G599	100.	99th adjustable work offset	+	m		
Each of the G functions in this group is used to activate an adjustable user frame $P_UIFR[ ]. G54 corresponds to frame $P_UIFR[1], G505 corresponds to frame $P_UIFR[5]. The number of adjustable user frames and, therefore, the number of G functions in this group, can be parameterized using machine data MD28080 $MC_MM_NUM_USER_FRAMES.						

Group 9: Frame suppression						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G53	1.	Suppression of current frames: Programmable frame including system frame for TOROT and TOFRAME and active adjustable frame (G54 to G57, G505 to G599).	-	n		
SUPA	2.	As for G153 including suppression of system frames for actual-value setting, scratching, ext. work offset, PAROT including handwheel offsets (DRF), [external work offset], overlaid movement	-	n		
G153	3.	As for G53 including suppression of all channel-specific and/or NCU-global basic frames	-	n		

Group 10: Exact stop - continuous-path mode						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G60	1.	Exact stop	+	m	x	
G64	2.	Continuous-path mode	+	m		
G641	3.	Continuous-path mode with smoothing as per distance criterion (= programmable rounding clearance)	+	m		
G642	4.	Continuous-path mode with smoothing within the defined tolerances	+	m		
G643	5.	Continuous-path mode with smoothing within the defined tolerances (block-internal)	+	m		
G644	6.	Continuous-path mode with smoothing with maximum possible dynamic response	+	m		
G645	7.	Continuous-path mode with smoothing and tangential block transitions within the defined tolerances	+	m		

Group 11: Exact stop, non-modal						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G9	1.	Exact stop	-	n		

Group 12: Block change criteria at exact stop (G60/G9)						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G601	1.	Block change at exact stop fine	+	m	x	
G602	2.	Block change at exact stop coarse	+	m		
G603	3.	Block change at IPO - end of block	+	m		

Group 13: Workpiece measuring inch/metric						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G70	1.	Input system inches (length)	+	m		
G71	2.	Input system metric mm (lengths)	+	m	x	
G700	3.	Input system inch, inch/min (lengths + velocity + system variable)	+	m		
G710	4.	Input system metric mm, mm/min (lengths + velocity + system variable)	+	m		

Group 14: Workpiece measuring absolute/incremental						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G90	1.	Absolute dimension	+	m	x	
G91	2.	Incremental dimension input	+	m		

Group 15: Feed type						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G93	1.	Inverse-time feedrate 1/rpm	+	m		
G94	2.	Linear feedrate in mm/min, inch/min	+	m	x	
G95	3.	Revolutional feedrate in mm/rev, inch/rev	+	m		
G96	4.	Constant cutting rate and type of feedrate as for G95 ON	+	m		
G97	5.	Constant cutting rate and type of feedrate as for G95 OFF	+	m		
G931	6.	Feedrate specification by means of traversing time, deactivate constant path velocity	+	m		
G961	7.	Constant cutting rate and type of feedrate as for G94 ON	+	m		
G971	8.	Constant cutting rate and type of feedrate as for G94 OFF	+	m		
G942	9.	Freeze linear feedrate and constant cutting rate or spindle speed	+	m		
G952	10.	Freeze revolutional feedrate and constant cutting rate or spindle speed	+	m		
G962	11.	Linear feedrate or revolutional feedrate and constant cutting rate	+	m		
G972	12.	Freeze linear feedrate or revolutional feedrate and constant cutting rate	+	m		
G973	13	Revolutional feedrate without spindle speed limitation (G97 without LIMS for ISO mode)	+	m		

Group 16: Feedrate override on inside and outside curvature						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
CFC	1.	Constant feedrate at contour effective for internal and external radius	+	m	x	
CFTCP	2.	Constant feedrate in tool center point (center point path)	+	m		
CFIN	3.	Constant feedrate for internal radius only, acceleration for external radius	+	m		

Group 17: Approach and retraction response, tool offset						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
NORM	1.	Normal position at starting and end points	+	m	x	
KONT	2.	Travel around contour at starting and end points	+	m		
KONTT	3.	Approach/retraction with constant tangent	+	m		
KONTC	4.	Approach/retraction with constant curvature	+	m		

Group 18: Corner behavior, tool offset						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G450	1.	Transition circle (tool travels around workpiece corners on a circular path)	+	m	x	
G451	2.	Intersection of equidistant paths (tool backs off from the workpiece corner)	+	m		

Group 19: Curve transition at beginning of spline						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
BNAT	1.	Natural transition to first spline block	+	m	x	
BTAN	2.	Tangential transition to first spline block	+	m		
BAUTO	3.	Definition of the first spline section by means of the next 3 points	+	m		

Group 20: Curve transition at end of spline						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
ENAT	1.	Natural transition to next traversing block	+	m	x	
ETAN	2.	Tangential transition to next traversing block	+	m		
EAUTO	3.	Definition of the last spline section by means of the last 3 points	+	m		

Group 21: Acceleration profile						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
BRISK	1.	Fast non-smoothed path acceleration	+	m	x	
SOFT	2.	Soft smoothed path acceleration	+	m		
DRIVE	3.	Velocity-dependent path acceleration	+	m		

Group 22: Tool offset type						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
CUT2D	1.	2½D tool offset determined by G17-G19	+	m	x	
CUT2DF	2.	2½D tool offset determined by frame The tool offset is effective in relation to the current frame (inclined plane).	+	m		
CUT3DC [5]	3.	3D tool offset circumferential milling	+	m		
CUT3DF [5]	4.	3D tool offset face milling with non-constant tool orientation	+	m		
CUT3DFS [5]	5.	3D tool offset face milling with constant tool orientation independent of active frame	+	m		
CUT3DFF [5]	6.	3D tool offset face milling with fixed tool orientation dependent on active frame	+	m		
CUT3DCC [5]	7.	3D tool offset circumferential milling with limitation surfaces	+	m		
CUT3DCCD [5]	8.	3D tool offset circumferential milling with limitation surfaces and differential tool	+	m		

Group 23: Collision monitoring at inside contours						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
CDOF	1.	Collision detection OFF	+	m	x	
CDON	2.	Collision detection ON	+	m		
CDOF2	3.	Collision detection OFF (currently only for CUT3DC)	+	m		

Group 24: Feedforward control						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
FFWOF	1.	Feedforward control OFF	+	m	x	
FFWON	2.	Feedforward control ON	+	m		

Group 25: Tool orientation reference						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
ORIWKS [5]	1.	Tool orientation in workpiece coordinate system (WCS)	+	m	x	
ORIMKS [5]	2.	Tool orientation in machine coordinate system (MCS)	+	m		

Group 26: Repositioning point for REPOS						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
RMB	1.	Reapproach to start of block position	+	m		
RMI	2.	Reapproach to interruption point	+	m	x	
RME	3.	Repositioning to end-of-block position	+	m		
RMN	4.	Reapproach to nearest path point	+	m		

Group 27: Tool offset for change in orientation at outside corners						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
ORIC [5]	1.	Orientation changes at outside corners are superimposed on the circle block to be inserted	+	m	x	
ORID [5]	2.	Orientation changes are performed before the circle block	+	m		

Group 28: Working area limitation						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
WALIMON	1.	Working area limitation ON	+	m	x	
WALIMOF	2.	Working area limitation OFF	+	m		

Group 29: Radius/diameter programming						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
DIAMOF	1.	Modal channel-specific diameter programming OFF Deactivation activates channel-specific radius programming.	+	m	x	
DIAMON	2.	Modal independent channel-specific diameter programming ON The effect is independent of the programmed dimensions mode (G90/G91).	+	m		
DIAM90	3.	Modal dependent channel-specific diameter programming ON The effect is dependent on the programmed dimensions mode (G90/G91).	+	m		
DIAMCYCOF	4.	Modal channel-specific diameter programming during cycle processing OFF	+	m		

Group 30: NC block compression						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
COMPOF [5]	1.	NC block compression OFF	+	m	x	
COMPON [5]	2.	Compressor function COMPON ON	+	m		
COMPCURV [5]	3.	Compressor function COMPCURV ON	+	m		
COMPCAD [5]	4.	Compressor function COMPCAD ON	+	m		

Group 31: OEM G function group						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G810 [5]	1.	OEM G function	-	m		
G811 [5]	2.	OEM G function	-	m		
G812 [5]	3.	OEM G function	-	m		
G813 [5]	4.	OEM G function	-	m		
G814 [5]	5.	OEM G function	-	m		
G815 [5]	6.	OEM G function	-	m		
G816 [5]	7.	OEM G function	-	m		
G817 [5]	8.	OEM G function	-	m		
G818 [5]	9.	OEM G function	-	m		
G819 [5]	10.	OEM G function	-	m		
Two G function groups are reserved for the OEM user. This enables the OEM to program functions that can be customized.						

Group 32: OEM G function group						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G820 [5]	1.	OEM - G function	-	m		
G821 [5]	2.	OEM G function	-	m		
G822 [5]	3.	OEM G function	-	m		
G823 [5]	4.	OEM G function	-	m		
G824 [5]	5.	OEM G function	-	m		
G825 [5]	6.	OEM G function	-	m		
G826 [5]	7.	OEM G function	-	m		
G827 [5]	8.	OEM G function	-	m		
G828 [5]	9.	OEM G function	-	m		
G829 [5]	10.	OEM G function	-	m		
Two G function groups are reserved for the OEM user. This enables the OEM to program functions that can be customized.						

Group 33: Settable fine tool offset						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
FTOCOF [5]	1.	Online fine tool offset OFF	+	m	x	
FTOCON [5]	2.	Online fine tool offset ON	-	m		

Group 34: Tool orientation smoothing						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
OSOF [5]	1.	Tool orientation smoothing OFF	+	m	x	
OSC [5]	2.	Continuous tool orientation smoothing	+	m		
OSS [5]	3.	Tool orientation smoothing at end of block	+	m		
OSSE [5]	4.	Tool orientation smoothing at start and end of block	+	m		
OSD [5]	5	Block-internal smoothing with specification of path length	+	m		
OST [5]	6	Block-internal smoothing with specification of angular tolerance	+	m		

Group 35: Punching and nibbling						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
SPOF [5]	1.	Stroke OFF, nibbling and punching OFF	+	m	x	
SON [5]	2.	Nibbling ON	+	m		
PON [5]	3.	Punching ON	+	m		
SONS [5]	4.	Nibbling ON in interpolation cycle	-	m		
PONS [5]	5.	Punching ON in interpolation cycle	-	m		

Group 36: Punching with delay						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
PDELAYON [5]	1.	Punching with delay ON	+	m	x	
PDELAYOF [5]	2.	Punching with delay OFF	+	m		

Group 37: Feed profile						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
FNORM [5]	1.	Feed normal (as per DIN 66025)	+	m	x	
FLIN [5]	2.	Feed linear variable	+	m		
FCUB [5]	3.	Feedrate variable according to cubic spline	+	m		

Group 38: Assignment of fast inputs/outputs for punching/nibbling						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
SPIF1 [5]	1.	Fast NCK inputs/outputs for punching/nibbling byte 1	+	m	x	
SPIF2 [5]	2.	Fast NCK inputs/outputs for punching/nibbling byte 2	+	m		

Group 39: Programmable contour accuracy						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
CPRECOF	1.	Programmable contour precision OFF	+	m	x	
CPRECON	2.	Programmable contour precision ON	+	m		

Group 40: Tool radius compensation constant						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
CUTCONOF	1.	Constant tool radius compensation OFF	+	m	x	
CUTCONON	2.	Constant tool radius compensation ON	+	m		

Group 41: Interruptible thread cutting						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
LFOF	1.	Interruptible thread cutting OFF	+	m	x	
LFON	2.	Interruptible thread cutting ON	+	m		

Group 42: Toolholder						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
TCOABS	1.	Determine tool length components from the current tool orientation	+	m	x	
TCOFR	2.	Determine tool length components from the orientation of the active frame	+	m		
TCOFRZ	3.	Determine tool orientation of an active frame on selection of tool, tool points in Z direction	+	m		
TCOFRY	4.	Determine tool orientation of an active frame on selection of tool, tool points in Y direction	+	m		
TCOFRX	5.	Determine tool orientation of an active frame on selection of tool, tool points in X direction		m		

Group 43: SAR approach direction						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G140	1.	SAR approach direction defined by G41/G42	+	m	x	
G141	2.	SAR approach direction to left of contour	+	m		
G142	3.	SAR approach direction to right of contour	+	m		
G143	4.	SAR approach direction tangent-dependent	+	m		

Group 44: SAR path segmentation						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G340	1.	Spatial approach block; in other words, infeed depth and approach in plane in one block	+	m	x	
G341	2.	Start with infeed on perpendicular axis (Z), then approach in plane	+	m		

Group 45: Path reference for FGROUP axes						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
SPATH	1.	Path reference for FGROUP axes is arc length	+	m	x	
UPATH	2.	Path reference for FGROUP axes is curve parameter	+	m		

Group 46: Plane selection for fast retraction						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
LFTXT	1.	The plane is determined from the path tangent and the current tool orientation	+	m	x	
LFWP	2.	The plane is determined by the current working plane (G17/G18/G19)	+	m		
LFPOS	3.	Axial retraction to a position	+	m		

Group 47: Mode switchover for external NC code						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G290	1.	Activate SINUMERIK language mode	+	m	x	
G291	2.	Activate ISO language mode	+	m		

Group 48: Approach and retraction response with tool radius compensation						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
G460	1.	Collision detection for approach and retraction block ON	+	m	x	
G461	2.	Extend border block with arc if no intersection in TRC block	+	m		
G462	3.	Extend border block with straight line if no intersection in TRC block	+	m		

Group 49: Point-to-point motion						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
CP	1.	Path motion	+	m	x	
PTP	2.	Point-to-point motion (synchronized axis motion)	+	m		
PTPG0	3.	Point-to-point motion only with G0, otherwise path motion CP	+	m		

Group 50: Orientation programming						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
ORIEULER	1.	Orientation angle via Euler angle	+	m	x	
ORIRPY	2.	Orientation angle via RPY angle (rotation sequence XYZ)	+	m		
ORIVIRT1	3.	Orientation angle via virtual orientation axes (definition 1)	+	m		
ORIVIRT2	4.	Orientation angle via virtual orientation axes (definition 2)	+	m		
ORIAXPOS	5.	Orientation angle via virtual orientation axes with rotary axis positions	+	m		
ORIRPY2	6.	Orientation angle via RPY angle (rotation sequence ZYX)	+	m		

Group 51: Interpolation type for orientation programming						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
ORIVECT	1.	Large-radius circular interpolation (identical to ORIPLANE)	+	m	x	
ORIAXES	2.	Linear interpolation of machine axes or orientation axes	+	m		
ORIPATH	3.	Tool orientation trajectory referred to path	+	m		
ORIPLANE	4.	Interpolation in plane (identical to ORIVECT)	+	m		
ORICONCW	5.	Interpolation on a peripheral surface of the cone in clockwise direction	+	m		
ORICONCCW	6.	Interpolation on the peripheral surface of a taper in the counter-clockwise direction	+	m		
ORICONIO	7.	Interpolation on a conical peripheral surface with intermediate orientation setting	+	m		
ORICONTO	8.	Interpolation on a peripheral surface of the cone with tangential transition	+	m		
ORICURVE	9.	Interpolation with additional space curve for orientation	+	m		
ORIPATHS	10.	Tool orientation in relation to path, blips in the orientation characteristic are smoothed	+	m		

Group 52: Frame rotation in relation to workpiece						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
PAROTOF	1.	Frame rotation in relation to workpiece OFF	+	m	x	
PAROT	2.	Frame rotation in relation to workpiece ON The workpiece coordinate system is aligned on the workpiece.	+	m		

Group 53: Frame rotation in relation to tool						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
TOROTOF	1.	Frame rotation in relation to tool OFF	+	m	x	
TOROT	2.	Align the Z axis of the workpiece coordinate system parallel to the workpiece orientation by rotating the frame	+	m		
TOROTZ	3.	As TOROT	+	m		
TOROTY	4.	Align the Y axis of the workpiece coordinate system parallel to the workpiece orientation by rotating the frame	+	m		
TOROTX	5.	Align the X axis of the workpiece coordinate system parallel to the workpiece orientation by rotating the frame	+	m		
TOFRAME	6.	Align the Z axis of the workpiece coordinate system parallel to the workpiece orientation by rotating the frame	+	m		
TOFRAMEZ	7.	As TOFRAME	+	m		
TOFRAMEY	8.	Align the Y axis of the workpiece coordinate system parallel to the workpiece orientation by rotating the frame	+	m		
TOFRAMEX	9.	Align the X axis of the workpiece coordinate system parallel to the workpiece orientation by rotating the frame	+	m		

Group 54: Vector rotation for polynomial programming						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
ORIROTA	1.	Vector rotation absolute	+	m	x	
ORIROTR	2.	Vector rotation relative	+	m		
ORIROTT	3.	Vector rotation tangential	+	m		
ORIROTC	4.	Tangential rotational vector in relation to path tangent	+	m		

Group 55: Rapid traverse with/without linear interpolation						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
RTLION	1.	Rapid traverse motion with linear interpolation ON	+	m	x	
RTLIOF	2.	Rapid traverse motion with linear interpolation OFF Rapid traverse motion is achieved with single-axis interpolation.	+	m		

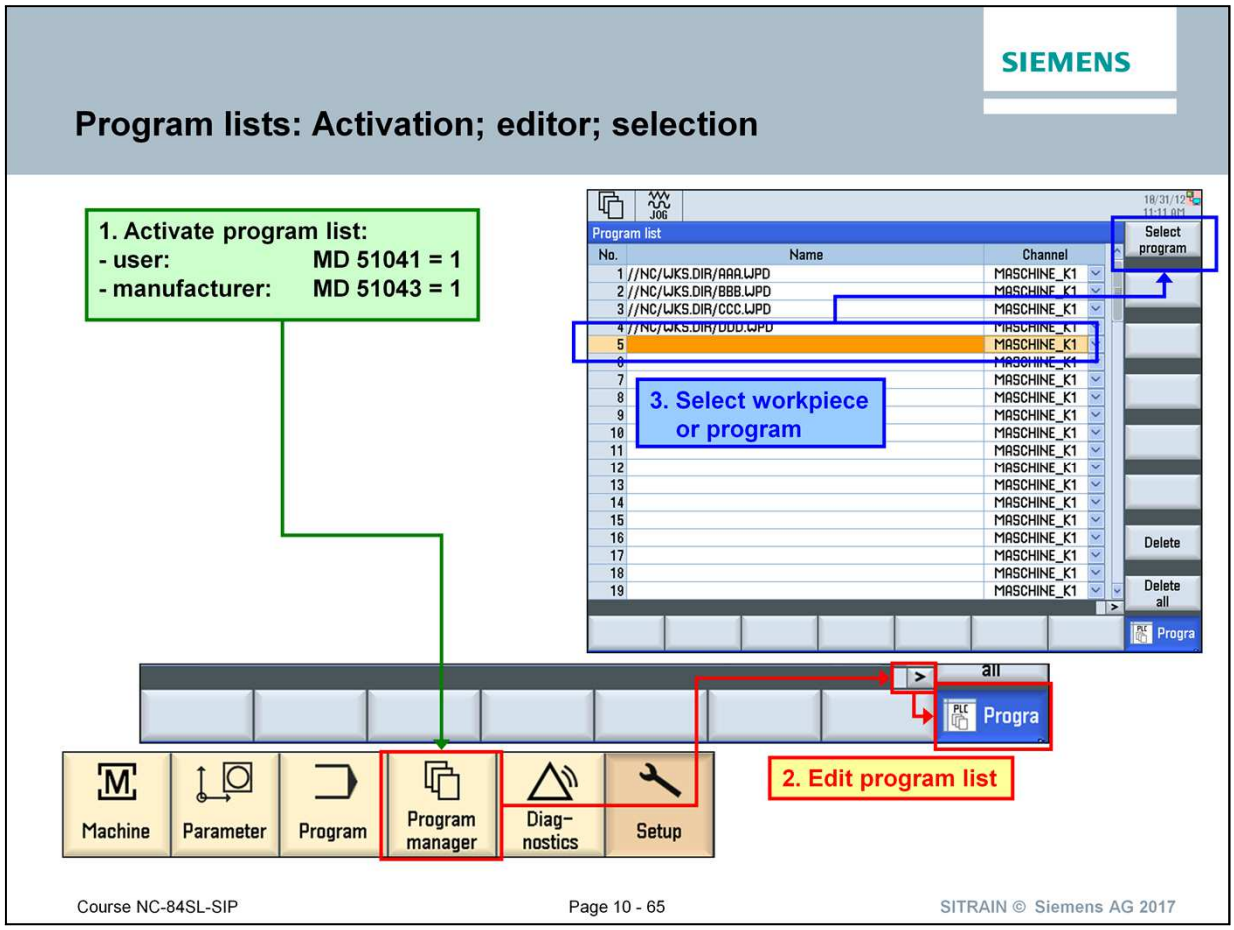
Group 56: Inclusion of tool wear						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
TOWSTD	1.	Initial setting value for offsets in tool length	+	m	x	
TOWMCS	2.	Wear values in the machine coordinate system	+	m		
TOWWCS	3.	Wear values in the workpiece coordinate system	+	m		
TOWBCS	4.	Wear values in the basic coordinate system (BCS)	+	m		
TOWTCS	5.	Wear values in the tool coordinate system (toolholder ref. point T at the tool holder)	+	m		
TOWKCS	6.	Wear values in the coordinate system of the tool head for kinetic transformation (differs from machine coordinate system through tool rotation)	+	m		

Group 57: Corner deceleration						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
FENDNORM	1.	Corner deceleration OFF	+	m	x	
G62	2.	Corner deceleration at inside corners when tool radius compensation is active (G41/G42)	+	m		
G621	3.	Corner deceleration at all corners	+	m		

Group 59: Dynamic response mode for path interpolation						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
DYNNORM	1.	Standard dynamic, as previously	+	m	x	
DYNPOS	2.	Positioning mode, tapping	+	m		
DYNROUGH	3.	Roughing	+	m		
DYNSEMIFIN	4.	Finishing	+	m		
DYNFINISH	5.	Smooth-finishing	+	m		

Group 60: Working area limitation						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
WALCS0	1.	Workpiece coordinate system working area limitation OFF	+	m	x	
WALCS1	2.	WCS working area limitation group 1 active	+	m		
WALCS2	3.	WCS working area limitation group 2 active	+	m		
WALCS3	4	WCS working area limitation group 3 active	+	m		
WALCS4	5	WCS working area limitation group 4 active	+	m		
WALCS5	6	WCS working area limitation group 5 active	+	m		
WALCS6	7	WCS working area limitation group 6 active	+	m		
WALCS7	8	WCS working area limitation group 7 active	+	m		
WALCS8	9	WCS working area limitation group 8 active	+	m		
WALCS9	10	WCS working area limitation group 9 active	+	m		
WALCS10	11	WCS working area limitation group 10 active	+	m		

Group 61: Tool orientation smoothing						
G function	No. [1]	Significance	MD20150 [2]	W [3]	STD [4]	
					SAG	MM
ORISOF	1.	Tool orientation smoothing OFF	+	m	x	
ORISON	2.	Tool orientation smoothing ON	+	m		



Program list

The program list is used for selecting programs via the PLC.

- Activation

The following machine data bits are used for activation:

MD 51041 = 1 ENABLE_PROGLIST_USER

MD 51043 = 1 ENABLE_PROGLIST_MANUFACTURER

The two lists can be found at

/card/user/sinumerik/hmi/plc/programlist/plc_proglist_user.ppl

or

/card/oem/sinumerik/hmi/plc/programlist/plc_proglist_manufacturer.ppl

- Editing

You can find the editor for the program list as follows:

-> OA: Program manager -> ETC key -> SK: Progra

(should be SK: “Prog. list”)

- Selection

You can make a selection as follows:

- Complete workpiece directories; example:

1 //NC/WKS.DIR/AAA.WPD

This assumes that there is a job list with the same name that controls program selection for the individual channels.

Advantage: Exactly one line per workpiece!

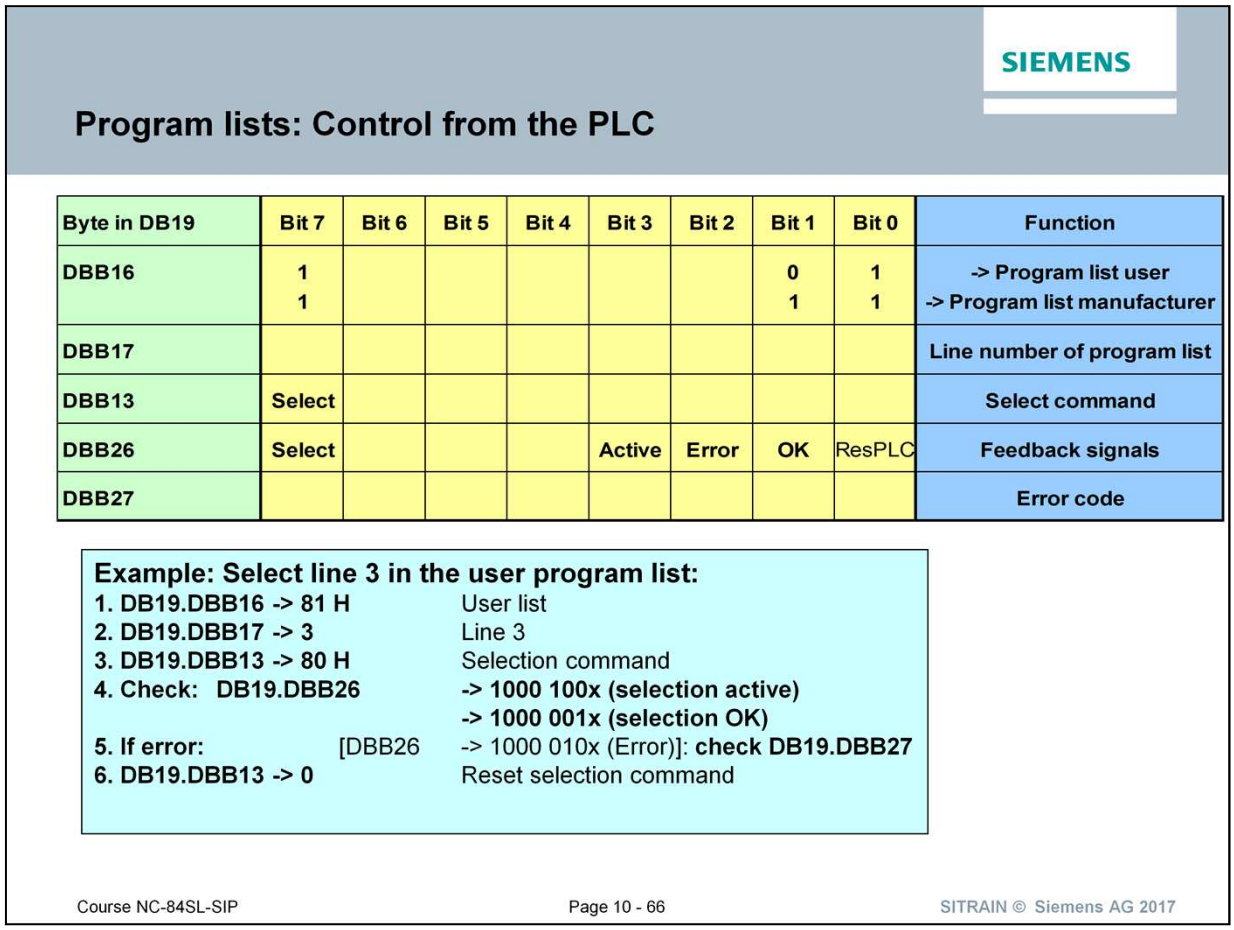
- The alternative would be to make one entry for each channel in the program list; example:

1 //NC/WKS.DIR/AAA.WPD/K1.MPF Channel 1

2 //NC/WKS.DIR/AAA.WPD/K2.MPF Channel 2

Disadvantage: Multiple entries required for each workpiece;

-> makes control via the PLC complicated! (Exception: just 1 channel!)



Program list: Control from the PLC

An interface in DB19 is used to control the program list from the PLC:

- DBB16 Select user or manufacturer list:

= 1 proglist_user
= 3 proglist_manufacturer

- DBB17 Line numbers in program list: 1-100

NOTICE In contrast to the documentation, the number range **1 - 100** applies to both cases!

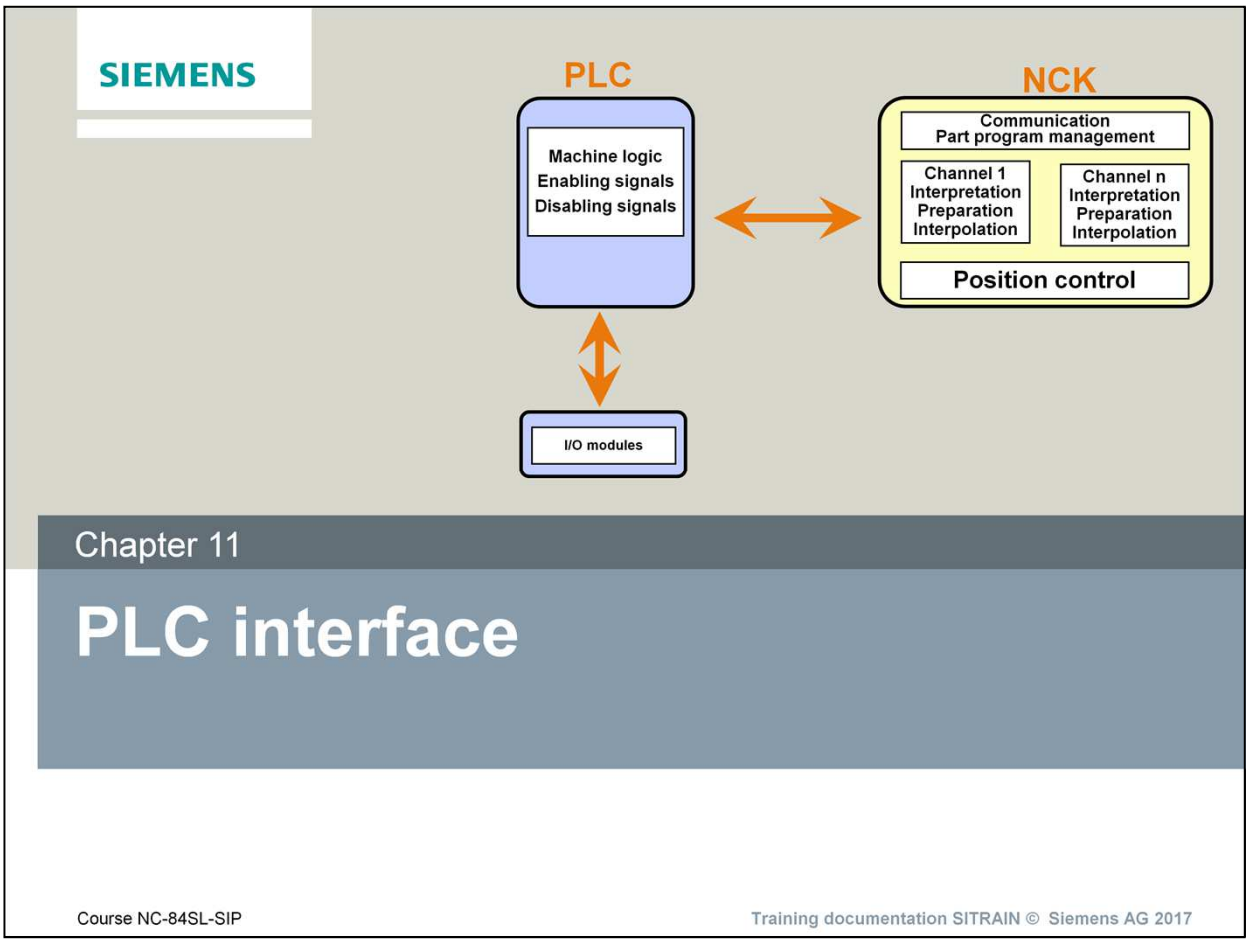
- DBB13 Bit 7: Selection command **This bit must be reset before a new select command can be issued!**

- DBB26 Feedback signals

Bit 1: OK
Bit 2: Error -> Evaluate DBB27!
Bit 3: Active
Bit 7: Selection

- DBB27 Error identifiers:

Value	Significance
0	No error
1	Invalid number for control file. (value in DB19.DBB16 < 127 or invalid)
2	Unable to read DB19.DBB15
3	Control file /BD.DIR/PLC_IN_OUT_xxx.TEA not found. (value in DB19.DBB16 invalid)
4	Invalid index in control file. (incorrect value in DB19.DBB17)
5	Selected job list in control file could not be opened.
6	Error in job list (job list interpreter returns error)
7	Job list interpreter returns empty job list
9	Error while executing job list



Learning objectives 2
Structure of the basic program 3
Overview of blocks 4
Overview of interface DBs 5
Machine control panel 6
Interface signals 8
Generating data sets 12
Backing up the PLC program 13
Alarm texts 14
User alarms 15
Alarm text converter 20
Using the alarm text 21
Alarm text converter 22
Alarm text help 23
Exercise: 27
Additional exercise: 34
Auxiliary functions 36
Help functions 37
Auxiliary functions 39
Exercise: 48

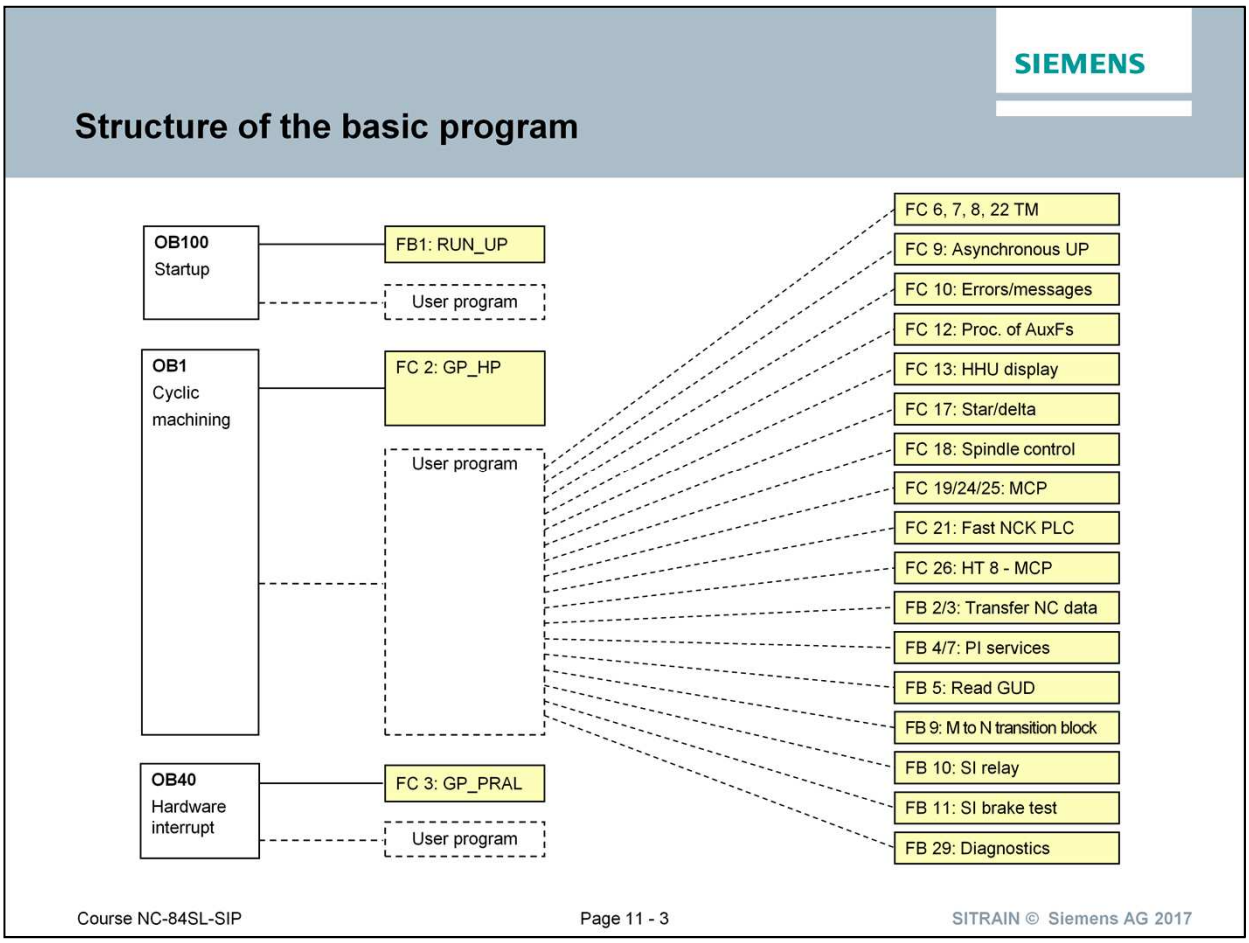
Learning objectives

- You will have an overview of the structure of the basic program
- You will be familiar with the interface DBs
- You will be familiar with the enable signals
- You will be able to generate error and system status messages in SINUMERIK Operate
- You will be familiar with processing the auxiliary functions and their interface



SITRAIN © Siemens AG 2017

Notes



PLC program

The PLC program is clearly structured.

It must include the following organization blocks:

- Startup OB 100
- Cyclical mode (OB 1)
- Hardware interrupt processing OB 40

In OB 1, 40, and 100, the respective part of the SIEMENS basic program must be called by the user.

Restart

During a restart, the NCK and PLC are synchronized. The system and user data blocks are checked to ensure that they are complete and the most important basic program parameters are checked to ensure that they are plausible. If an error occurs, the basic program transfers an error code to the diagnostic buffer and puts the PLC into the stop state. After system initialization, the operating system executes **OB 100** and always starts cyclic operation at the beginning of **OB 1**.

The PLC is synchronized with the HMI and NCK during run-up. After a correct boot procedure and the first complete OB 1 cycle, the PLC and NCK constantly exchange sign-of-life signals. If no sign-of-life signal is received from the NC, the interface is neutralized and the "NCK CPU ready" signal in DB10 is set to zero.

Overview of blocks

Blocktype	reserved for SIEMENS	free for user
FB	0 - 29 1000-1023	30 – 999 1024 – upper limit
FC	0 - 29 1000-1023	30 – 999 1024 – upper limit
DB	1 – 62 71 – 80 1000 - 1099	63-70 1100 – upper limit
Timer	-	1 - 512

SITRAIN © Siemens AG 2017

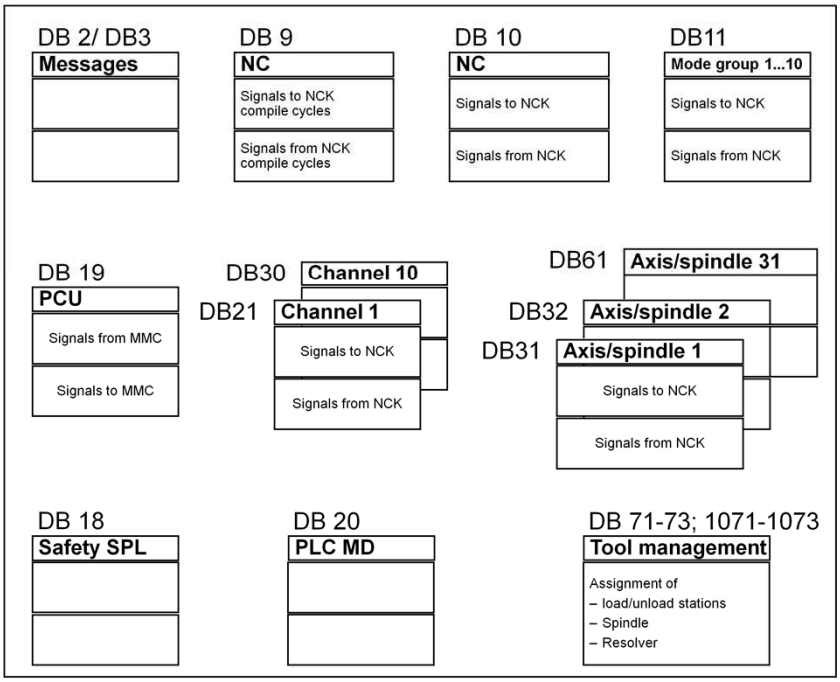
Reserved blocks

The SIEMENS basic program needs FB/FC/DB for cyclic signal exchange between NC/PLC. All non-reserved blocks are available for use.

DBs

The data blocks of channels, axes/spindles and tool management functions that are not activated may be assigned as desired by the user.

Overview of interface DBs



 SITRAIN © Siemens AG 2017

NCK/PLC interface

The NCK/PLC interface comprises a data interface and a function interface. The data interface contains status and control signals as well as auxiliary and G functions.

The basic program generates these data blocks using the current NC machine data (number of channels, axes/spindles, etc.) during system run-up.

Data interface

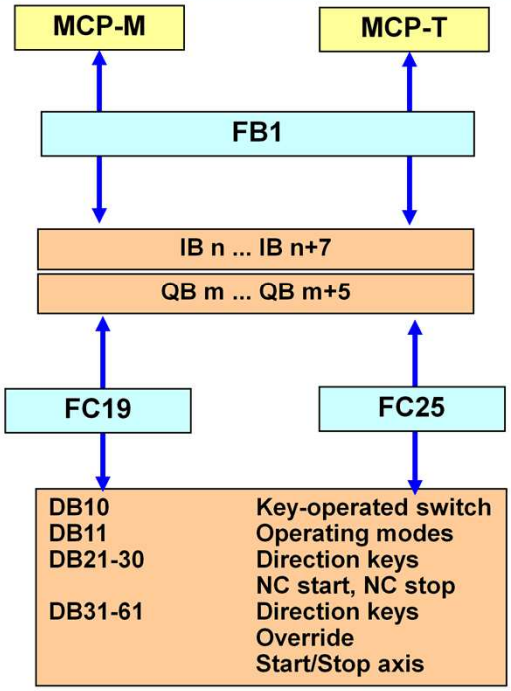
The data interface comprises a number of data blocks and is divided into the following groups:

- NCK-specific signals
- Mode-group-specific signals
- Channel-specific signals
- Axis/spindle/drive-specific signals

Function interface

Comprises FBs and FCs via which commands are transferred from the PLC to the NCK.

Machine control panel



Course NC-84SL-SIP Page 11 - 6 SITRAIN © Siemens AG 2017

FB 1

FB1 is called absolutely in OB100. The addresses for the signals of the 1st machine control panel are set in two formal parameters in FB1.
MCP1In: POINTER Start address input signals MCP 1
MCP1Out: POINTER Start address output signals MCP 1

The default setting is assigned address zero.
In a further parameter of the FB1, the bus that is connected to the MCP is queried.

MCPBusType:=B#16#**33** Bus type for PROFIBUS control panel,
MCPBusType:=B#16#**55** Bus type for Industrial Ethernet control panel.

FC19/25

FC19 or FC25 is called absolutely or conditionally in OB1. It has 5 formal parameters for transferring MCP signals from the process image memory to the respective data blocks.

BAGNo:=	B#16#1	//Mode group no. 1
ChanNo:=	B#16#1	//Channel no. 1
SpindlelFNo:=	B#16#4	//e.g. 4th axis is the spindle
FeedHold:=	DB21.DBx6.0	//Feedrate stop, modal
SpindleHold:=	DB34.DBX4.3	//Spindle stop, modal

Note

If MCP310 is used, FC 24 must be used.
FC19/25 are designed for MCP 483.

Machine control panel
Interface signals

SIEMENS

Signals from machine control panel (keys)								
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
IB n + 0	Spindle speed override				Operating mode			
	D	C	B	A	JOG	TEACH IN	MDA	AUTO
IB n + 1	Machine function							
	REPOS	REF	var. INC	10000 INC	1000 INC	100 INC	10 INC	1 INC
IB n + 2	Key-switch position 0	Key-switch position 2	Spindle start	*Spindle stop	Feed start	*Feed stop	NC Start	*NC Stop
IB n + 3		Key-switch position 1		Feedrate override				
	Reset		Single block	E	D	C	B	A
IB n + 4	Direction keys			Keyswitch position 3	Direction keys			
	+ R15	- R13	Rapid traverse R14		x R1	4th axis R4	7th axis R7	R10
IB n + 5	Axis selection							
	Y R2	Z R3	5th axis R5	Traverse command MCS/WCS R12	R11	R9	8th axis R8	6th axis R6
IB n + 6	Unassigned customer keys							
	T9	T10	T11	T12	T13	T14	T15	
IB n + 7	Unassigned customer keys							
	T1	T2	T3	T4	T5	T6	T7	T8

SITRAIN © Siemens AG 2017

Signals to machine control panel (LEDs)								
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
QB n + 0	Machine function				Operating mode			
	1000 INC	100 INC	10 INC	1 INC	JOG	TEACH IN	MDA	AUTO
QB n + 1	Feed start	*Feed stop	NC Start	*NC Stop	Machine function			
					REPOS	REF	var. INC	10000 INC
QB n + 2	Axis selection					Single block	Spindle start	*Spindle stop
	Direction key - R13	X R1	4th axis R4	7th axis R7	R10			
QB n + 3	Axis selection							
	Z R3	5th axis R5	Travel command MCS/WCS R12	R11	R9	8th axis R8	6th axis R6	Direction key + R15
QB n + 4	Unassigned customer keys							
	T9	T10	T11	T12	T13	T14	T15	R2
QB n + 5	Unassigned customer keys							
	T1	T2	T3	T4	T5	T6	T7	T8

Interface signals Axis/spindle

SIEMENS

In the PLC interface for axes/spindles, the following signals must be set:

IS "Controller enable"	(DB31, ... DBX2.1)
IS "Pulse enable"	(DB31, ... DBX21.7)
IS "Position measur. system 1 or 2"	(DB31, ... DBX1.5, DBX 1.6)

The following interface signals must not be set if the axis/spindle is to be moved:

IS "Feedrate/spindle speed override switch"	(DB31, ... DBB0) not to 0 %
IS "Axis/spindle disable"	(DB31, ... DBX1.3)
IS "Follow-up mode"	(DB31, ... DBX1.4)
IS "Feed stop/Spindle stop"	(DB31, ... DBX4.3)
IS "Traversing key disable"	(DB31, ... DBX4.4)
IS "Read-in disable"	(DB21, ... DBX6.1)
IS "Feedrate disable"	(DB21, ... DBX6.0)

Course NC-84SL-SIP Page 11 - 8 SITRAIN © Siemens AG 2017

Controller release

The position control loop is closed on a 0-to-1 edge transition. Edge transition from 1-to-0 leads to rapid deceleration and the position control loop is opened. Actual value detection continues and the controller enable of the drive is canceled.

The controller enable can also be reset within the control in the event of various faults, e.g. on the position measuring system.

Pulse enable

When a 1 signal is present, pulse enable is active for the relevant drive. If pulse enable is reset for a traversing axis, the axis coasts to a standstill.

Position measuring system 1/2

With this interface signal, a measuring system is selected for position control. The motor measuring system or the direct measuring system can be selected.

Axis disable

When a 1 signal is present, no setpoints are output to the position controller. This means that no traversing movement takes place; only the display changes. This signal can be used for running in and testing part programs.

Follow-up mode

The position setpoint constantly tracks the actual value. Standstill and clamping tolerance monitoring are not effective.

Read-in disable

The "Read-in disable" signal is only effective in AUTOMATIC and MDI modes. When a 1 signal is present, data transfer of the next part program block to the interpolator is disabled.

Feed disable

This signal is effective in one channel for all operating modes. When the signal state is 0, the axes can be traversed if a travel request is present. When the signal state changes to 1, the axes are decelerated at the contour, the following error is reduced, and position control remains active.

Interface signals
Hardware/software

SIEMENS

Errors

OK

DB10	Signals from the NC (NC → PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB103 Operating software → PLC	Operating software, battery alarm /Z1-A2/	Operating software, temperature limit /Z1-A2/	AT box ready /Z1-A2/	Operating software, fan monitoring	Operating software, hard disk monitoring			Remote diagnostics active /Z1-A2/
DBB104 GP → PLC	NC CPU ready /Z1-A2/	1. OB1 cycle		Op2Key ready	Op1Key ready	HHU ready	MCP 2 ready	MCP 1 ready
DBB108	NC ready /Z1-A2/	Drive ready /Z1-A2/	Drives in cyclic operation /Z1-A2/		Operating software CPU at OPI /Z1-A2/ /-B3/	Operating software CPU at MPI ready /Z1-A2/ /-B3/	Operating software 2 CPU ready E_MM2_Ready /Z1-A2/ /-B3/	
DBB109	NC battery alarm /Z1-A2/	Air temperature alarm /Z1-A2/	NCU heat sink temp. alarm /Z1-A2/	Reserved				NC alarm is active /Z1-A2/

DB31 - DB61	Signals from the axis/spindle (NC → PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB94 Operating mechanism	Variable signaling function 5)	$n_{act} = n_{set}$ /Z1-A2/	$\|n_{act}\| < n_x$ /Z1-A2/	$\|n_{act}\| < n_{min}$ /Z1-A2/	$M_d < M_{dx}$ /Z1-A2/	Run-up completed	Temperature pre-alarm	
							Heat sink	Motor

SITRAIN © Siemens AG 2017

CPU ready

These signals are found for the individual components under:

- **NCK-CPU** **DB10.DBX 104.7**
- **Drive** **DB10.DBX 108.6**
- **HMI-CPU** **DB10.DBX 108.3**

Ready

Ready signals of other operator components

- **HHU** **DB10.DBX 104.2** Handheld unit / HT
- **MCP 1 / 2** **DBX 104.0 / 104.1** Machine control panel

Temperature, fan, battery alarm

They are the signals:

NC	**DB10.DBX 109.5**	Heat sink temp.
	DB10.DBX 109.6	Air temp. alarm
	DB10.DBX 109.7	Battery alarm
HMI	**DB10.DBX 103.6**	HMI temp. limit
	DB10.DBX 103.4	HMI fan monitoring
	DB10.DBX 103.7	HMI battery alarm
Motor	**DB30+n.DBX 94.0**	Temp. prewarning
MoMo	**DB30+n.DBX 94.1**	Temp. prewarning heat sink

Hard disk **HMI** **DB10.DBX 103.3** HMI hard disk monitoring

Software **NCK** **DB10.DBX 108.7** NCK Ready

Drive **DB10.DBX 108.6** Drive Ready (only comes in after DC link!)

Interface signals
TCU changeover

SIEMENS

DB19	Signals from the OP (operating software → PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB114	ParTcuIndex TCU index call waiting interface							
DBB115	ParHt2Index HT2 index log on interface							
DBB118	TCU index 1st online interface							
DBB119	TCU index 2nd online interface							
DBB126	Reserved	TCU1_ SHIFT_ LOCK	MMC1_ CHANGE_ DENIED /Z2-B3/	MMC1_ ACTIVE_ CHANGED /Z2-B3/	MMC1_ ACTIVE_ PERM /Z2-B3/	MMC1_ ACTIVE_ REQ /Z2-B3/	MMC1_ MSTT_ SHIFT- LOCK /Z2-B3/	MMC1_ SHIFT_ LOCK /Z2-B3/
DBB136	Reserved	TCU2_ SHIFT_ LOCK	MMC2_ CHANGE_ DENIED /Z2-B3/	MMC2_ ACTIVE_ CHANGED /Z2-B3/	MMC2_ ACTIVE_ PERM /Z2-B3/	MMC2_ ACTIVE_ REQ /Z2-B3/	MMC2_ MSTT_ SHIFT_ LOCK /Z2-B3/	MMC2_ SHIFT_ LOCK /Z2-B3/

Course NC-84SL-SIP Page 11 - 10 SITRAIN © Siemens AG 2017

DB19.DBX126.6 TCU changeover disable offers the option of dynamically disabling changeover from one TCU to the next when the system is running via the PLC. For the duration of the disable, a user authorization request to change user authorizations between TCUs will be ignored by the system and rejected. The "Changeover disable" function is always active; therefore, it does not have to be activated separately. The bit address for the first HMI is DB19.DBB126.6 and DB19.DBB136.6 for the second HMI.

TCU_INDEX The PLC m:n online interface is expanded so that byte DB19.DBB118 is still added to the first interface and byte DB19.DBB119 to the second. These bytes take the index of the active TCU (of the active OP) for the appropriate HMI. The byte is called TCU_INDEX. The TCU_INDEX byte is described by the relevant HMI, with the TCU index configured for the active TCU. The HMI assigns the configured index of the TCU or PCU whose OP currently has user authorization in the shadow grouping. If no OP is active, a value of 0 is entered in TCU_INDEX. If no TCU index is configured for the active TCU (active OP), the value 255 = undefined is transferred as the TCU index. This means that the values 0 and 255 may not be configured as the TCU index.

Comment on PCU These signals are irrelevant if a PCU is mounted on the OP (no TCU).

Interface signals
New in 840D sl

SIEMENS

DB31 - DB61	Signals to the axis/spindle (PLC → NC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB20 Operating mechanism			Open motor holding brake /FBSIsl/				Ramp-function generator disable 1)	
DBB21 Operating mechanism	Pulse enable	Speed controller integrator disable /Z1-A2/	Motor being selected /Z1-A2/	Motor/drive data set: Request interface (interface definition: DB31, ...DBX130.0 - 4)				
				E	D	C	B	A
DBB92 Drive	Drive operation enabled		Motor holding brake opened	Drive-autonomous motion active1)			Ramp-function-generator disable active /FB1/A2/	
DBB93 Drive	Pulses enabled /Z1-A3/	Speed controller integrator disabled /Z1-A2/	Drive ready /Z1-A2/	Motor/drive data set: Display interface (interface definition: DB31, ...DBX130.0 - 4)				
				E	D	C	B	A

SITRAIN © Siemens AG 2017

Holding brake

The holding brake actuated by the Motor Module can be opened via a signal.

DB 30+n.DBX 20.5 Holding brake open (n= axis number)

Feedback from the brake is

DB 30+n.DBX 92.5 Holding brake open

Motor selection and parameter set changeover

This is a special feature; the full 5-bit combination comprising the 2 bits for motor selection and the 3 bits for parameter set changeover are used internally in SINAMICS for the drive data set (DDS) selection.

DB 30+n.DBX 21.0 ... 2 Selection of drive data set

DB 30+n.DBX 21.3 ... 4 Selection of motor data set

Corresponding feedback signals are

DB 30+n.DBX 93.0 ... 2 Feedback of drive data set

DB 30+n.DBX 93.3 ... 4 Feedback of motor data set

Special feature

The corresponding SINAMICS parameters must be supplied according to fixed rules, e.g. motor data set number

MDS=p186[DDS]

or encoder data set number

EDS=p187[DDS]

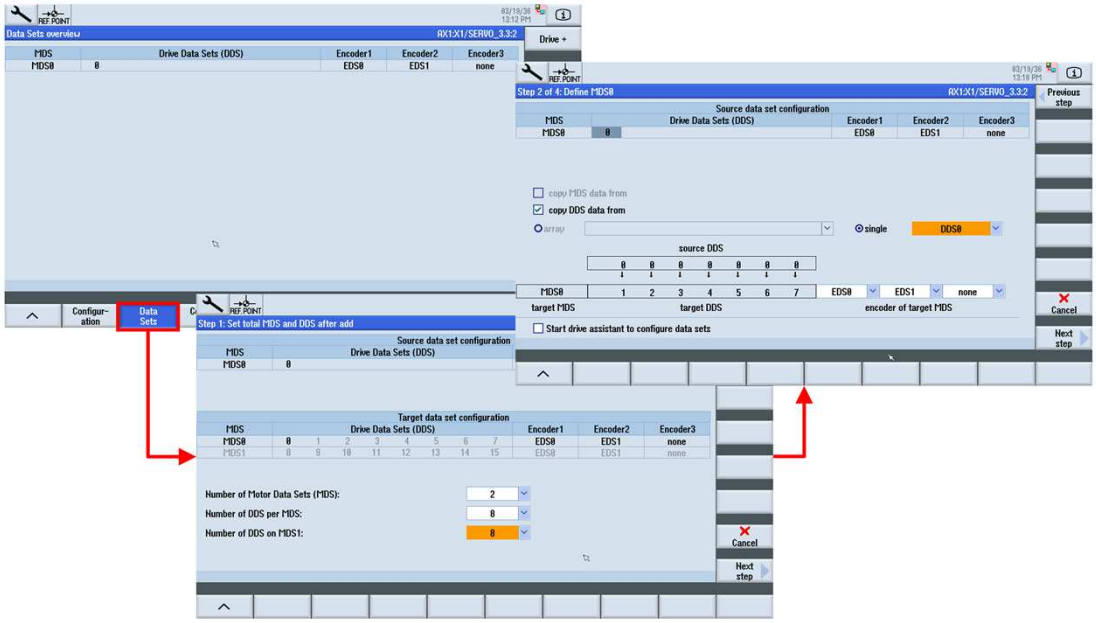
The reason is that the drive data sets (DDS) with the same 3 least significant bits mean the same motor (and, therefore, encoder).

Therefore, the following must be identical:

p186[0] = ... = p186[7]; p186[8]= ... =p186[15], etc.

p187[0] = ... = p187[7]; p187[8]= ... =p187[15], etc.

Generating data sets



SITRAIN © Siemens AG 2017

Generating data sets

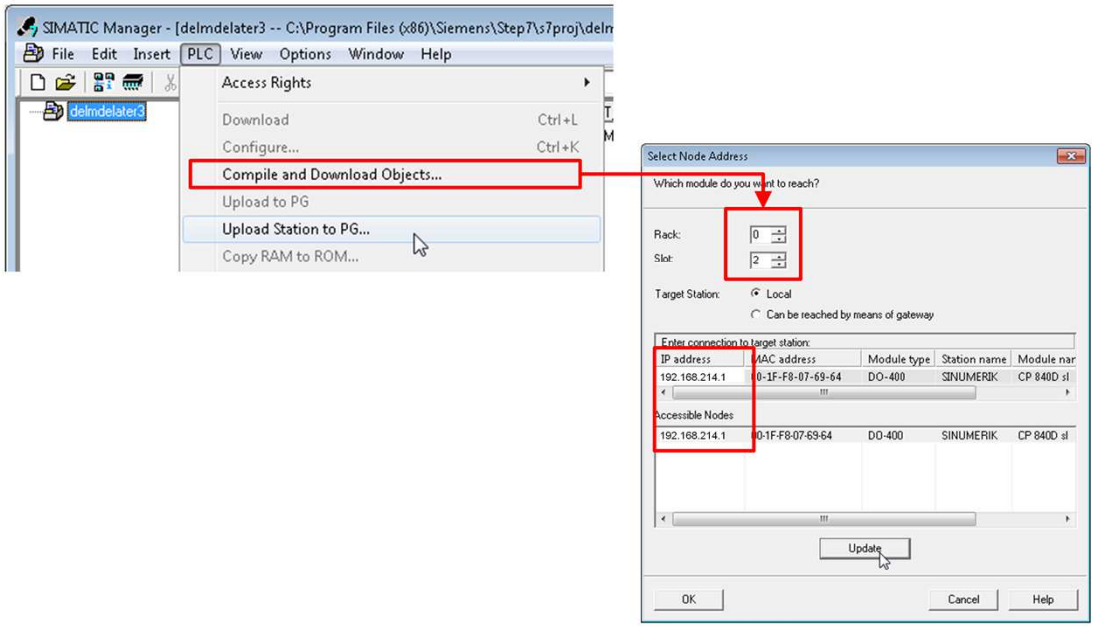
Data sets are generated using the functions that are available in the "Data sets" menu:

OA: Setup -> Drive system -> Drives -> Data sets

Functions

Here, the number of motor data sets and the relevant number of drive data sets per motor data set can be changed.
Newly created data sets can be generated (copied) from existing data sets and then modified.

Backing up the PLC program



SITRAIN © Siemens AG 2017

Load station in PG

To save the PLC program in a SIMATIC project, choose *Target system -> Upload Station to PG*. The blocks and the hardware configuration are transferred to the project.

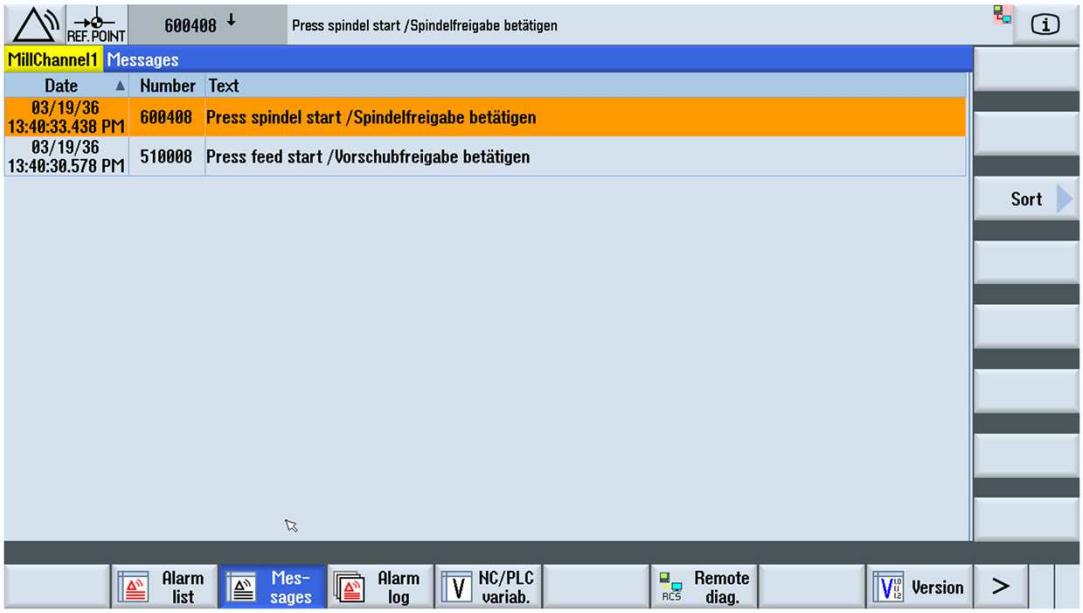
In the "Select node address" screen form, select:

Module rack *0* and **slot *2***

IP address

The IP address of the NCU must be entered in the IP address field.
(The **Display** button has no function with NCU 7x0.3.)

Alarm texts

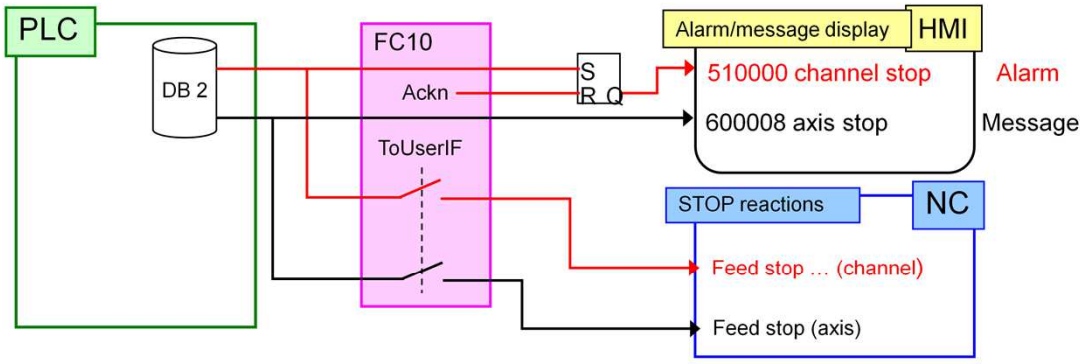


SITRAIN © Siemens AG 2017

The alarm and message texts are displayed in the Diagnostics operating area. The alarm protocol and the status/control function for the PLC can also be selected here.

The alarm texts can be entered using an input screen on the HMI.

User alarms
FC 10 alarm display (example)



Display of alarms on the HMI (SINUMERIK Operate or HMI Advanced)

```
CALL FC 10
   ToUserIF   := TRUE            ; transfer interlocks to channel / axis
   Quit       := DB19.DBX20.2    ; acknowledgment signal (CANCEL key)
U DB19.DBX20.2                   ; reset acknowledgment bit
R DB19.DBX20.2
```

 SITRAIN © Siemens AG 2017

FC 10

Using the function FC 10, alarms (error and system status messages) can be signaled to the HMI.

Only the alarm number is communicated; the texts must be stored in the operator control system. The specific number depends on the bit set in DB2.

Categories

For displaying and acknowledging alarms, there are two categories:

System status messages

Clear themselves automatically when the triggering bit is reset again in DB2.

Error messages

These must always be acknowledged. The error acknowledgment key is defined by calling FC10.

Parameters

FC10 has 2 formal parameters:

ToUserIF

:=TRUE

A STOP response (e.g. feed disable, read-in disable, feed/spindle stop, etc.) is triggered in the NC.

The response depends on the bit used in DB2 (and therefore on the alarm number).
The corresponding STOP signals in the standard channel / axis interface DB21 … / DB31 … are used.

:=FALSE

No STOP response is triggered in the NC. The user must program the STOP response himself.

Quit



This signal clears alarms (=error messages).

In the example, the key CANCEL :=DB 19.DBX20.2. Bit DB19.DBX20.2 must be reset by the user.

User alarms
Error and system status messages (DB2)

SIEMENS

DB2	Signals for PLC messages (PLC→MMC), /P3/							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
	Channel 1							
0	510007	510006	510005	510004	510003	510002	510001	510000
	Feed disable (alarm no.: 510000-510015)							
1	510015	510014	510013	510012	510011	510010	510009	510008
2	Feed and read-in disable byte1 (alarm no.: 510100-510131)							
3	Feed and read-in disable byte 2 (alarm no.: 510108-510115)							

	User area 0 Bytes 1 - 8							
180	700007	700006	700005	700004	700003	700002	700001	700000
...	User area 0 (alarm no.: 700000-700063)							
187	700063	700062	700061	700060	700059	700058	700057	700056
188-195	User area 1 Bytes 1 - 8 (alarm no.: 700100-700163)							
...								
372-379	User area 24 Bytes 1 - 8 (alarm no.: 702400-702463)							

Course NC-84SL-SIP Page 11 - 16 SITRAIN © Siemens AG 2017

DB2

DB2 is the alarm/message interface of the PLC with the HMI.

DB2 specifies which bit triggers which message/interrupt number. The bits in DB2 can also be used for automatic error response (STOP signals, such as feed stop, etc.) for defined channels and axes (if ToUserIf=TRUE in FC10).

The corresponding bits in DB 2 are set by the PLC user program, which makes the alarm number appear with text on the HMI (by means of FC10).

Note

The size of the user area for alarms is set in parameter **MsgUser** of FB1 (caution: if the value entered is less than 10, the value is increased to 10). FB1 is called in OB100. A maximum of 32 x 8 bytes can be set (value range: 0…32). If the MsgUser parameter is changed, DB2 and DB3 (copy of DB2 for interrupt acknowledgment) must be cleared so that the controller can create DB2 and DB3 again.

User alarms
DB 2 with FB1 param. "ExtendAlMsg"=True for sl

SIEMENS

DB2	Signals for PLC events (PLC → HMI) FB1 parameter "ExtendAlMsg" == TRUE							
Byte (message type)	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
	Signals WITHOUT display of a fault/operating message (DBB0 - 309)							
	Channel 1							
0	Feed disable							
1	Feed disable							
2	Read-in disable							
3	Read-in disable							
4	Start disable							
5	Start disable							
6	Feed stop, geo axis 1, byte 1							
7	Feed stop, geo axis 1, byte 2							
	...							
310 (FM)	510007	510006	510005	510004	510003	510002	510001	510000
311 (OM)	510015	510014	510013	510012	510011	510010	510009	510008
312 (FM)	Feed and read-in disable, byte 1 (event no.: 510100-510107)							
313 (FM)	Feed and read-in disable, byte 2 (event no.: 510108-510115)							
314 (OM)	Feed and read-in disable, byte 3 (event no.: 510116-510123)							
315 (OM)	Feed and read-in disable, byte 4 (event no.: 510124-510131)							
316 (FM)	Read-in disable, byte 1 (event no.: 510200-510207)							
317 (FM)	Read-in disable, byte 2 (event no.: 510208-510215)							
318 (OM)	Read-in disable, byte 3 (event no.: 510216-510223)							

Course NC-84SL-SIP Page 11 - 17 SITRAIN © Siemens AG 2017

FB1 parameter "ExtendAlMsg" = TRUE

When this parameter is activated, the new DB2 structure becomes effective. The new structure has the following features:

10 channels, 31 axes Interlocks for the full number of NC channels and axes are supported.

Interlocks without display

Additional bit fields that do not issue any interrupts or messages are available for disable and stop signals.

Parameterizable alarm texts

Additional integer value %Z for alarm texts in the user area 0 (DB2, Array Zinfo0)

Parameterizable alarm numbers

Second decade of the user alarm numbers in the range 0-9 variable (DB5, Array UserDek2No).

Note This function is only available for Solution Line.

Documentation This functionality is explained in detail in the basic program description P3. Simply search for **ExtendAlMsg.**

There, you will find a procedure for converting the PLC program from the old to the new structure, using the UDT1002.

User alarms
Error and system status messages: FC10

SIEMENS

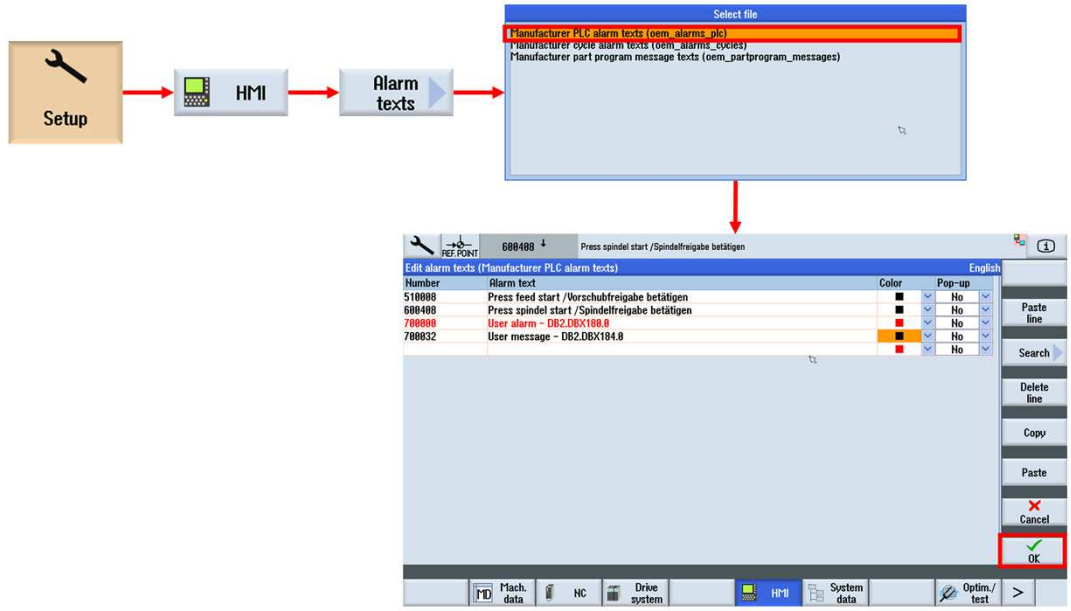
Definition of error and operational messages /P3/							
Byte no. of DB2 / Error message EM or operational message OM							
7 / EM	6 / EM	5 / OM	4 / OM	3 / EM	2 / EM	1 / OM	0 / EM
15 / OM	14 / EM	13 / OM	12 / EM	11 / OM	10 / EM	9 / OM	8 / OM
23 / OM	22 / OM	21 / EM	20 / EM	19 / OM	18 / EM	17 / OM	16 / EM
31 / OM	30 / EM	29 / OM	28 / EM	27 / OM	26 / OM	25 / EM	24 / EM
				35 / OM	34 / EM	33 / OM	32 / EM
151 / OM	150 / EM	149 / OM	148 / EM	147 / OM	146 / EM	145 / OM	144 / EM
159 / OM	158 / EM	157 / OM	156 / EM	155 / OM	154 / EM	153 / OM	152 / EM
187 / OM	186 / OM	185 / OM	184 / OM	183 / EM	182 / EM	181 / EM	180 / EM
195 / OM	194 / OM	193 / OM	192 / OM	191 / EM	190 / EM	189 / EM	188 / EM

Course NC-84SL-SIP Page 11 - 18 SITRAIN © Siemens AG 2017

Whether the set bit will be processed further as an error message (with acknowledgment) or a system status message (without acknowledgment) is preset.

Note This structure only applies if the following parameter of the FB1 is the default: **ExtendAlMsg=FALSE** (see figure). The definition always refers to a byte in DB2.

User alarms
Creating an alarm/message text



 SITRAIN © Siemens AG 2017

The operating area *Setup* > *HMI* > *Alarm texts* is used to select the types of alarm text to be created. The next screen form is used to enter the new alarm texts. These take effect once entry is complete. The color options at the end of the entry line enable you to choose whether an event will be displayed black or red.

The PopUp column

Defines whether the alarm will be displayed in a window on the screen or just in the alarm line. The alarm texts are stored for the selected language.

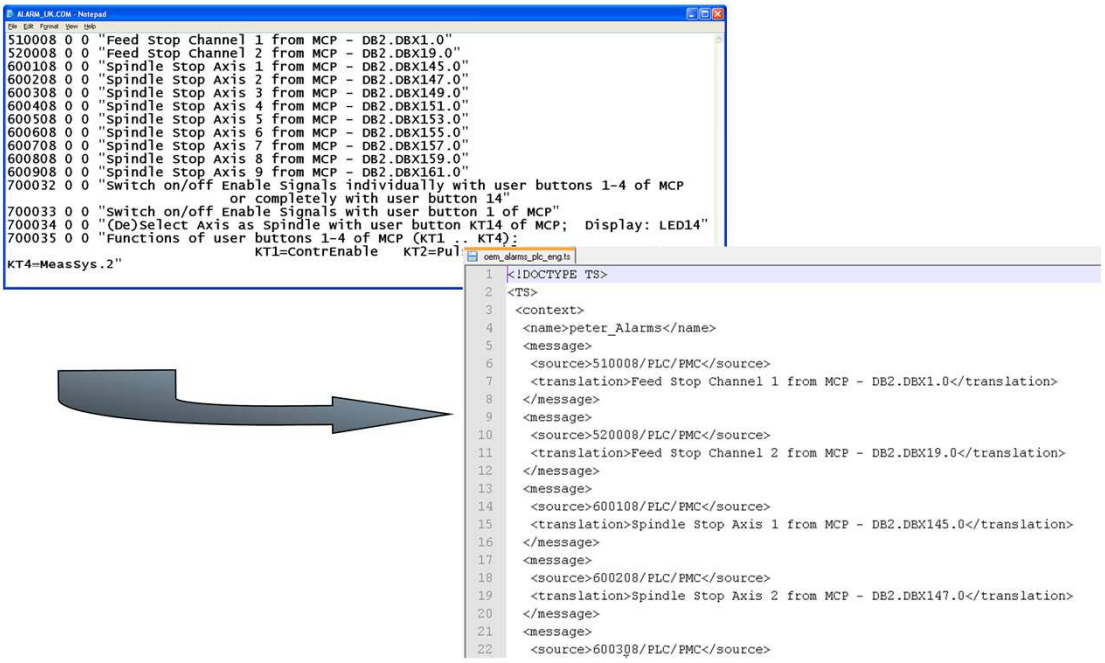
Storage on CF card

Alarm texts are stored on the CF card in XML format. They are located in the path
/card/oem/sinumerik/hmi/lng and there as the file
oem_alarms_plc_deu.ts or with the relevant language code.
The ts file is a Qt translation source file, the qm is a Qt message file that has been created.

umerik_alarm_oem_plc_pmc.html | 2: oem_alarms_plc_deu.ts

```
<!DOCTYPE TS>
<TS>
  <context>
    <name>slaeconv</name>
    <message>
      <source>510008/PLC/PMC</source>
      <translation>PRESS FEED START / Vorschub start druecken</translation>
    </message>
    <message>
      <source>600408/PLC/PMC</source>
      <translation>PRESS SPINDEL START / SPINDELFREIGABE DRUECKEN</translation>
    </message>
    <message>
      <source>700000/PLC/PMC</source>
      <translation>Anwenderalarm 1 DB2.DBX180.0</translation>
    </message>
    <message>
      <source>700032/PLC/PMC</source>
      <translation>Anwendermeldung 1 DB2.DBX184.0</translation>
    </message>
    <message>
```

Alarm text converter



Alarm text converter The alarm text converter is used to convert alarm text files that were created as normal text into alarm text files for SINUMERIK Operate.

Source files The entries in this file must be made in the following notation

<alarm_number 0 0 ``<text>`` that is, e.g.:

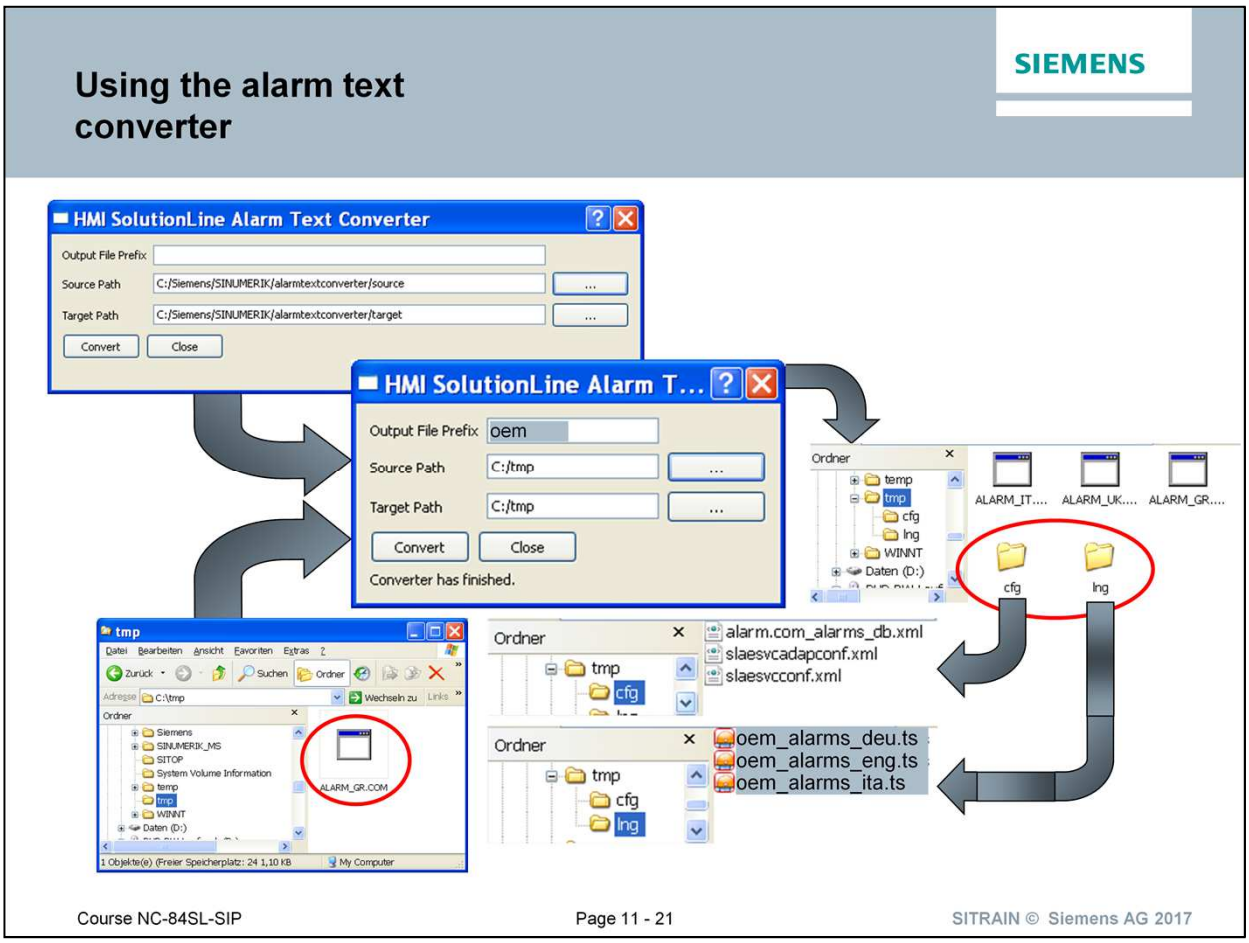
510008 0 0 ``FEED Stop …``

File name extension The source files have the following file name extension .COM

Languages The language IDs for the 5 standard European languages are:

German	gr	French	fr
English	uk	Italian	it
Spanish	sp		

XML/TS files The files are xml files (configuration) and ts files (texts).



First start Start the converter via the Start menu.

Preparing the sources Copy the alarm texts to be converted in all languages to the directory specified under "Source Path" (e.g. via Windows Explorer).

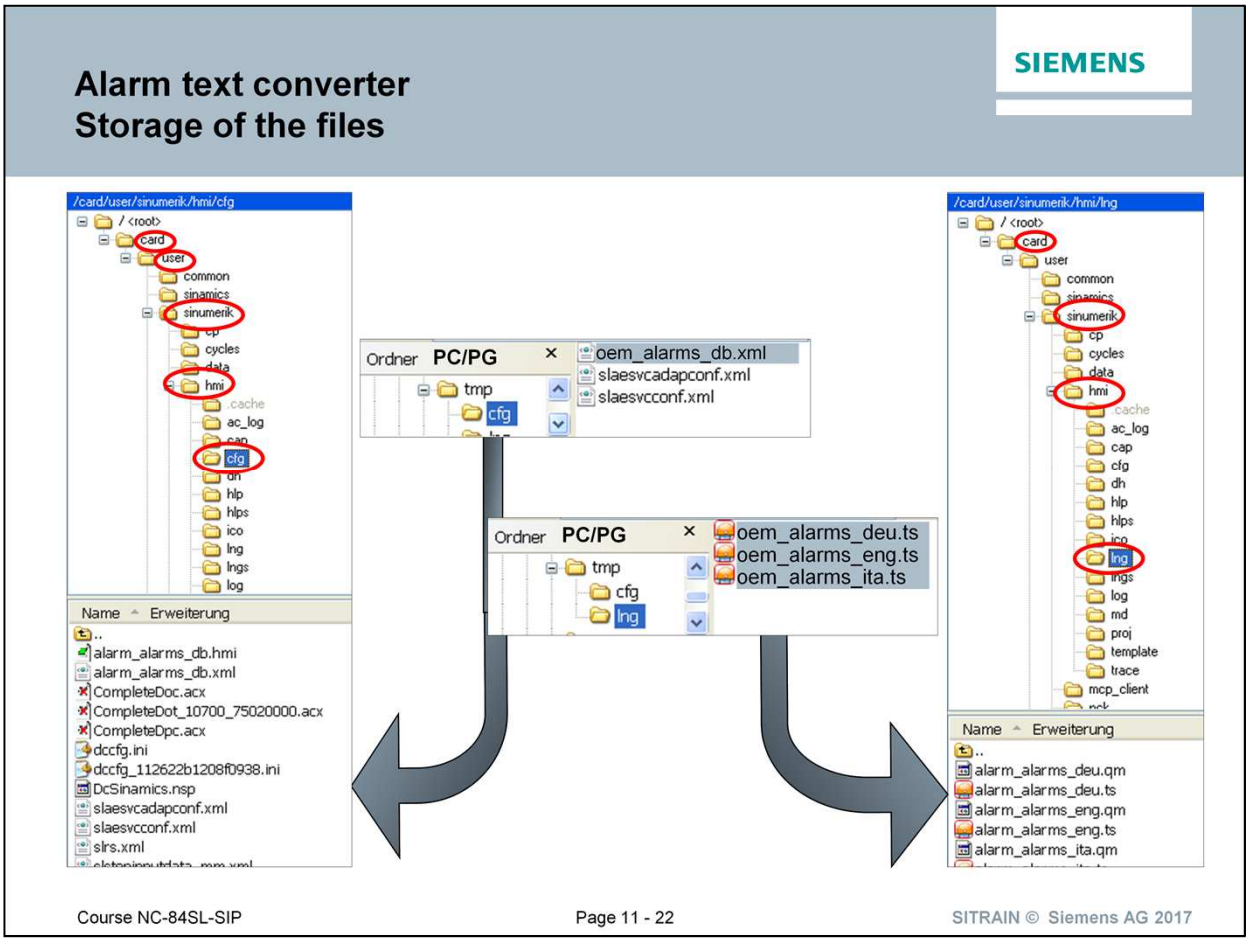
Note The file names of the alarm texts must contain a valid language code. The file name extension must be ".com", e.g. "ALARM_GR.COM".

Index texts If you also want to convert index texts, i.e. texts that are indexed via a parameter value (syntax: "%1<file selector>",
e.g. alarm no. 10203 "Channel %1 NC start without reference point (action=%2<ALNX>)"), copy the index texts to an additional directory, which will need to be set up under the "Source Path" with the name of the file selector (e.g. '..\source\ALNX').

Note
- Note that the names are case-sensitive.
- If you are using more than one file selector, create the appropriate number of subdirectories (all under the "Source Path").

Output File Prefix Enter the first part of the file name (prefix) for the SINUMERIK Operate alarm texts in the "Output File Prefix" entry field. This file name is used to distinguish between the alarm texts of different manufacturers. For this reason, the name should be easily recognizable, e.g. contain the name of the manufacturer.

Note: The prefix must not start with a number.



Conversion result

The converter generates two new subdirectories underneath the directory specified under 'Target Path'.

1. **'..\lng**': This contains the newly generated alarm and index texts (<prefix>_alarms_???.ts) for SINUMERIK Operate and
2. **'..\cfg'**, This contains the language-neutral database **<prefix>_alarms_db.xml**, and the newly generated configuration files '**slaesvcconf.xml**, and '**slaesvcadapconf.xml**'.

Now copy all the newly generated files to the relevant '.../lng' and '.../cfg' directories in the 'addon', 'oem', or 'user' path of the SINUMERIK Operate. An example would be /card/oem/SINUMERIK/hmi/lng. WinSCP, for instance, can be used for this.

operate on PCU

There are the paths:
c:\ProgramData\siemens\MotionControl\oem\sinumerik\hmi\cfg and
c:\ProgramData\siemens\MotionControl\oem\sinumerik\hmi\lng

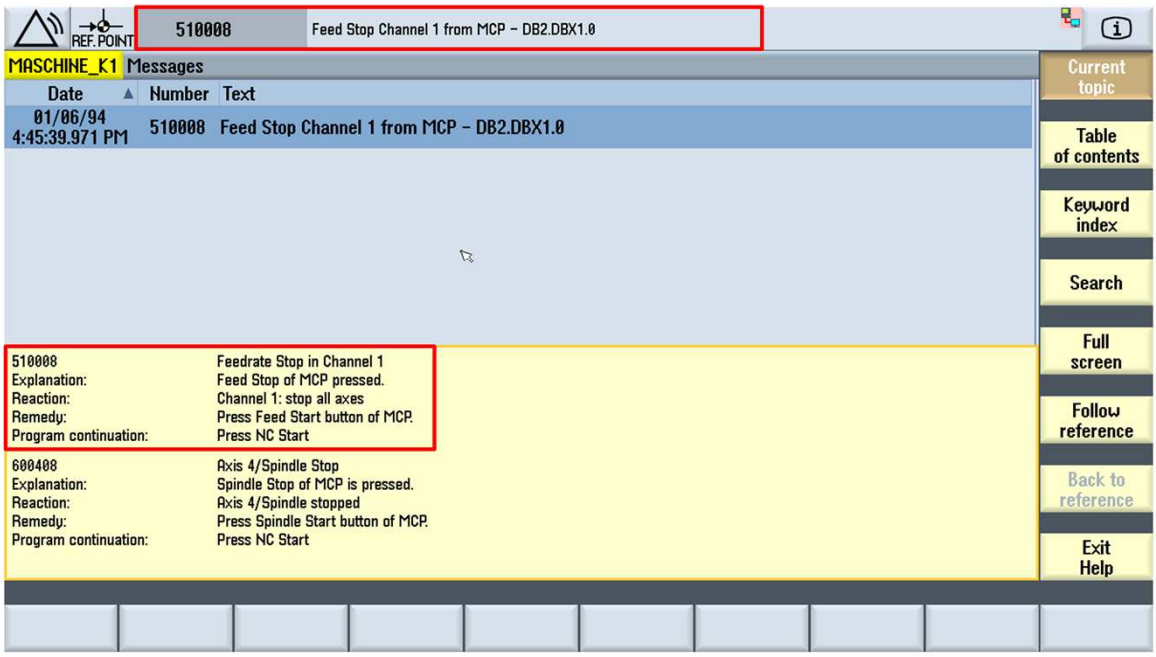
xml already exists

If the 'slaesvcconf.xml' configuration file already exists on the target, copy the furthest indented row from the configuration file that has just been generated to the file which already exists.
The same applies to the file slaesvcadapconf.xml

Example

```
<?xml version="1.0" encoding="UTF-8" standalone="yes" ?>
<CONFIGURATION>
    <AlarmTexts>
        <BaseNames>
            <oem_Alarms type="QString" value=„oem_alarms" />
        </BaseNames>
    </AlarmTexts>
</CONFIGURATION>
```

Alarm text help



 SITRAIN © Siemens AG 2017

Overview

A separate online help can be set up for user-specific PLC alarms. This can be opened context-sensitively from the alarm list with the "i" key when alarms occur.

Procedure

To set up a separate online help for user-specific PLC alarms, you perform the following steps:

Create the "**sinumerik_alarm_oem_plc_pmc.html**" HTML file.
This file contains the infotexts for the user-specific PLC alarms. The alarm number must be used as the HTML anchor.

A separate file called "sinumerik_alarm_oem_plc_pmc.html" must be created for each national language supported. The file name "sinumerik_alarm_oem_plc_pmc.html" must not be changed; it is the same for all national languages.

The files "sinumerik_alarm_oem_plc_pmc.html" are stored in language-specific directories, such as:

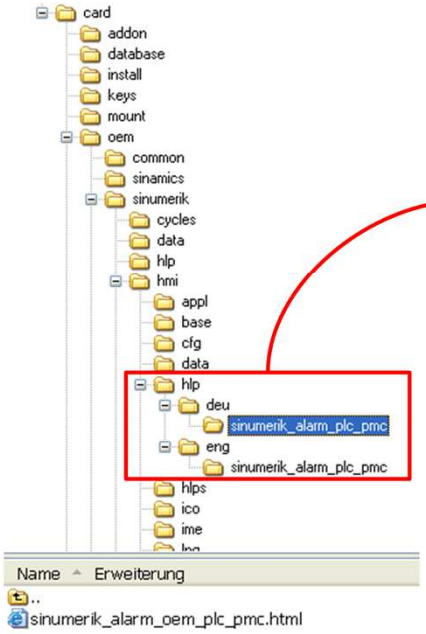
operate on NCU **/card/oem/sinumerik/hmi/hlp/<lng>/sinumerik_alarm_plc_pmc/ sinumerik_alarm_oem_plc_pmc.html**

operate on PCU **c:\ProgramData\siemens\MotionControl\oem\sinumerik\hmi\hlp\ <lng>\sinumerik_alarm_plc_pmc\sinumerik_alarm_oem_plc_pmc.html**
where (<lng>=deu, eng ...depending on the national languages supported).

NOTE!

You also need to delete the files called "**slhlp_sinumerik_alarm_plc_pmc_*.hmi**" in directory **/siemens/sinumerik/sys_cache/hmi//hlp**
and then restart the HMI. The deleted files are compilations that are regenerated from the HTML help file on restarting.

Alarm text help
Language codes



Language	Language code	Standard languages
Chinese (simplified)	chs	X
Chinese (traditional)	cht	
Danish	dan	
German	deu	X
English	eng	X
Finnish	fin	
French	fra	X
Indonesian	ind	
Italian	ita	X
Japanese	jpn	
Korean	kor	
Malaysian	msl	
Dutch	nld	
Polish	plk	

Subdirectories matching the language codes are generated in the oem/sinumerik/hmi/hlp folder. A subdirectory matching the alarm text type is then created inside these, e.g.:

sinumerik_alarm_plc_pmc

in which the alarm text file will ultimately be stored.

The alarm text file itself is an HTML file which can be created using suitable editors.

Alarm text help html file

SIEMENS

```
<!DOCTYPE HTML PUBLIC "-//W3C//DTD HTML 4.01 Transitional//EN">
<html>
<head>
  <meta http-equiv="Content-Type" content="text/html; charset=" utf-8="">
</head>

<body>

<table>

  <tbody>
    <tr>
      <td width="20%"><b><a name="510008">510008</a></b></td>
      <td width="80%"><b>Feed Stop Chanel 1</b></td>
    </tr>

    <tr>
      <td valign="top" width="20%"><b>Definitions:</b></td>
      <td width="80%">Feed Stop on MCP active.</td>
    </tr>

    <tr>
      <td valign="top" width="20%"><b>Reaction:</b></td>
      <td width="80%">CHanel 1: Stop all Axis</td>
    </tr>

    <tr>
      <td valign="top" width="20%"><b>Remedy:</b></td>
      <td width="80%">Operate feed start key on MCP.</td>
    </tr>

    <tr>
      <td valign="top" width="20%"><b>Program continuation:</b></td>
      <td width="80%">Press NC-start-key</td>
    </tr>

  </tbody>
</table>

<p></p>
```

Course NC-84SL-SIP Page 11 - 25 SITRAIN © Siemens AG 2017

The HTML file has a standard structure. The first component is a header:

<head><meta http-equiv="Content-Type" content="text/html; charset="UTF-8"/><title></title></head>

The file format is specified at the start and end of the HTML file in the relevant tags:

<HTML>

</HTML>

Next comes the file header, i.e. the title bar of the HTML page:

<head><title> </title> </head>

This is followed by the actual content of the page in the "Body" tag:

<body>

:

</body>

Help for the alarm text is stored in a table definition.

Alarm text help
Table definition

SIEMENS

```
<table>  <!-- Alarm 700000 -->
    <tr>
        <td width="15%">
            <b><a name="700000">700000</a></b>
        </td>

        <td width="85%">
            <b>This is the help for user alarm 700000</b>
        </td>
    </tr>

    <tr>
        <td valign="top" width="15%">
            <b>Explanation:</b>
        </td>
        <td width="85%">This is the explanation for user alarm 700000.</td>
    </tr>

    <tr>
        <td valign="top" width="15%"><b>Remedy:</b></td>
        <td width="85%">This is the remedy for alarm 700000 - Eliminate reason for alarm.</td>
    </tr>
</table>   <!-- Alarm 700000 end -->
```

Course NC-84SL-SIP Page 11 - 26 SITRAIN © Siemens AG 2017

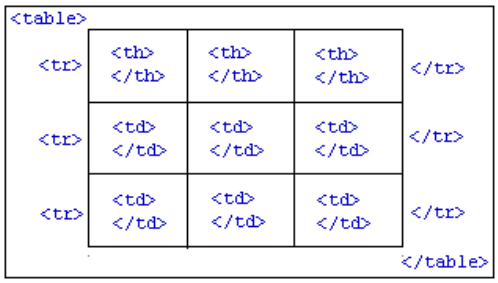
<table> Introduces a table.
If the table is to contain visible grid lines, you must enter a *border* attribute in the introductory *<table>* tag and allocate it a value higher than 0. The value specified will determine the width of the border in pixels. To create an "invisible" table without any visible borders or grid lines, do not specify a border or (this is the neater approach) specify *border="0"*.

<tr> Introduces a new table row *(tr = table row).* The cells (columns) are then defined for the respective row. A closing </tr> tag is added at the end of a table row.

A table can contain header cells and normal data cells. Text in header cells is accentuated (usually in bold characters and centered).

<th> Introduces a header cell.

<td> Introduces a normal data cell *(th = table header, td = table data)*. In each case, the content of a cell is entered after the tag. Although the associated **</th>** and **</td>** end tags are, strictly speaking, optional, we strongly recommend always using them in any case.



<a> This is known as the anchor tag and contains either a reference or just text.

<b> This is a tag for text formatting, b stands for black (i.e. bold).

Exercise: Backing up the PLC program

Creating an S7 project

Create a new S7 project. Assign a project name of your choosing.

Ethernet cable

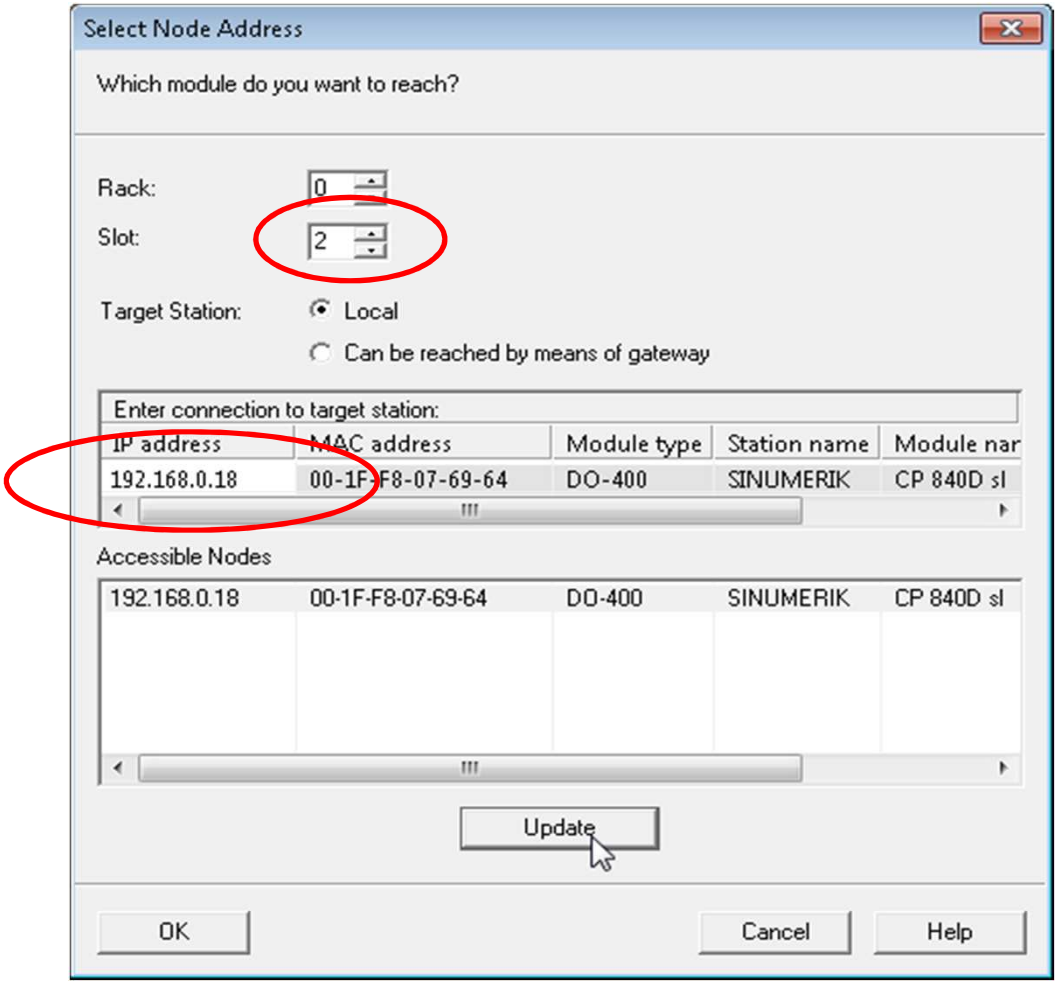
Establish a connection between PG (Ethernet interface) and 840D sl (Ethernet interface **X120** or **X127**).

Backing up a PLC program

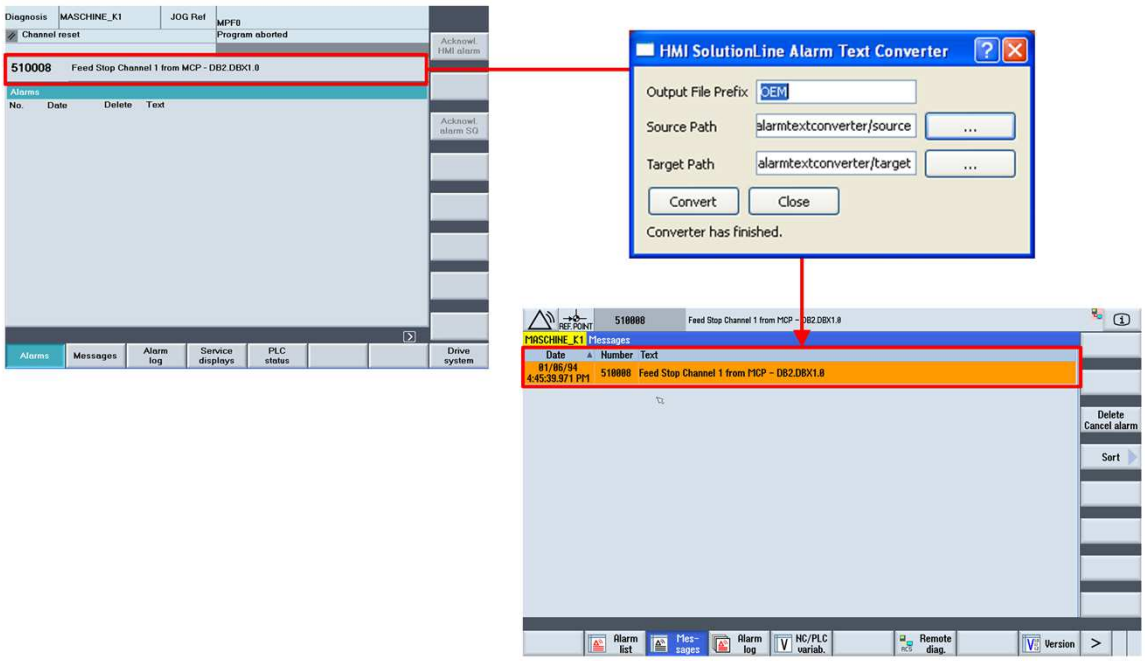
To back up the PLC program on the PG, choose *Target system* > *Upload station to PG*.

In the next dialog box, you must specify slot 2 and the corresponding IP address of the NCU:

- X120: IP = 192.168.214.1
- X127: IP = 192.168.215.1



Exercise:
Using the alarm text converter



Course NC-84SL-SIP Page 11 - 28 SITRAIN © Siemens AG 2017

Objective

In this exercise, we are going to use the alarm text converter to generate texts for SINUMERIK Operate from text files.

So that language selection works correctly, the texts for German (.._GR.COM) and English (.._UK.COM) are converted.

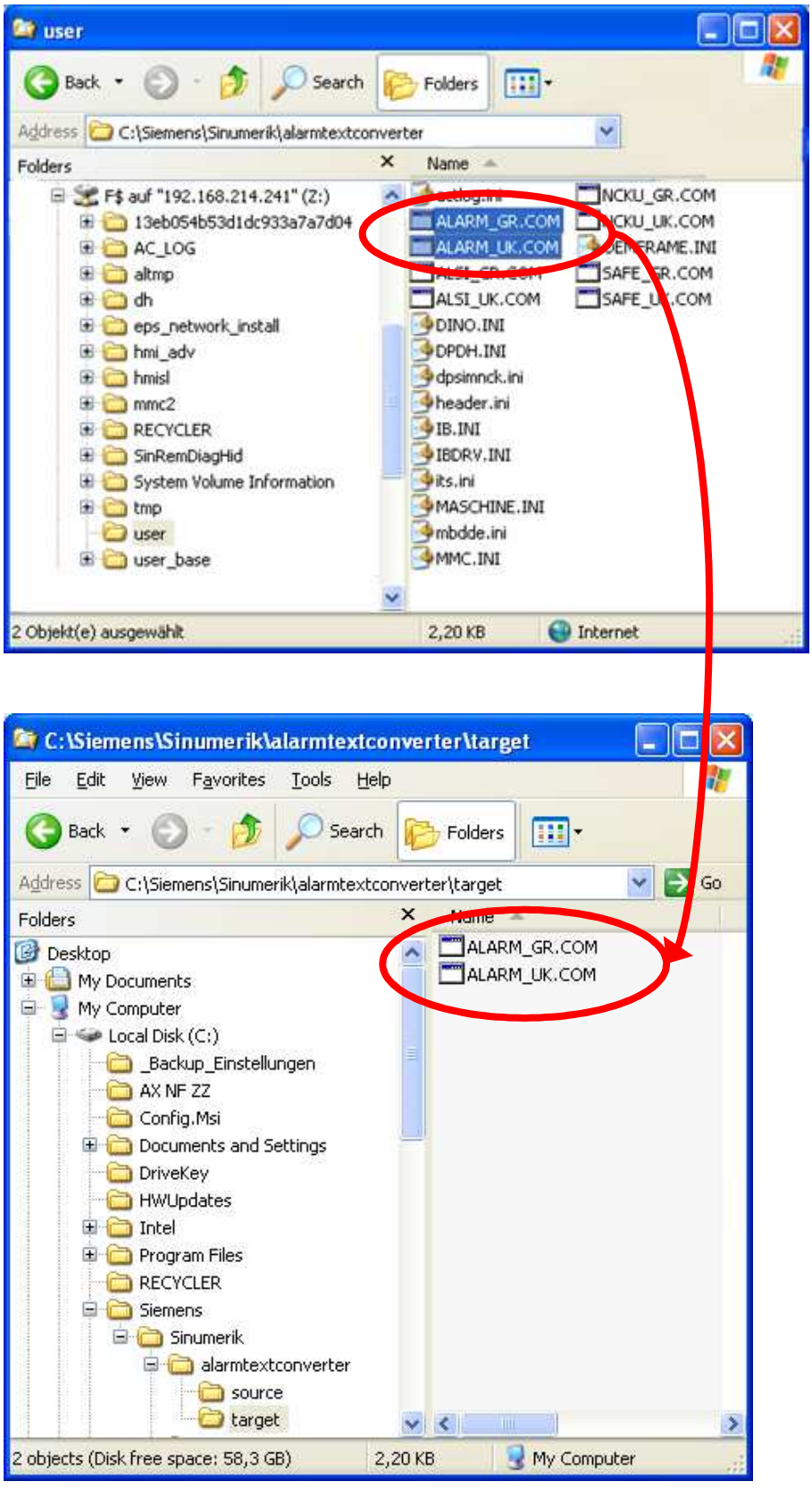
Steps

1. Copy the alarm text files to the converter's **source** directory.
2. Convert the alarm texts with the converter.
3. Copy the files created by the converter from the **target** path to the CF card.
4. Call FC10 in OB1
5. Re-parameterize the FC19 call
   (assign FeedHold and SpindleHold to messages in DB2).
6. Power ON and test the alarm text display.

Procedure:

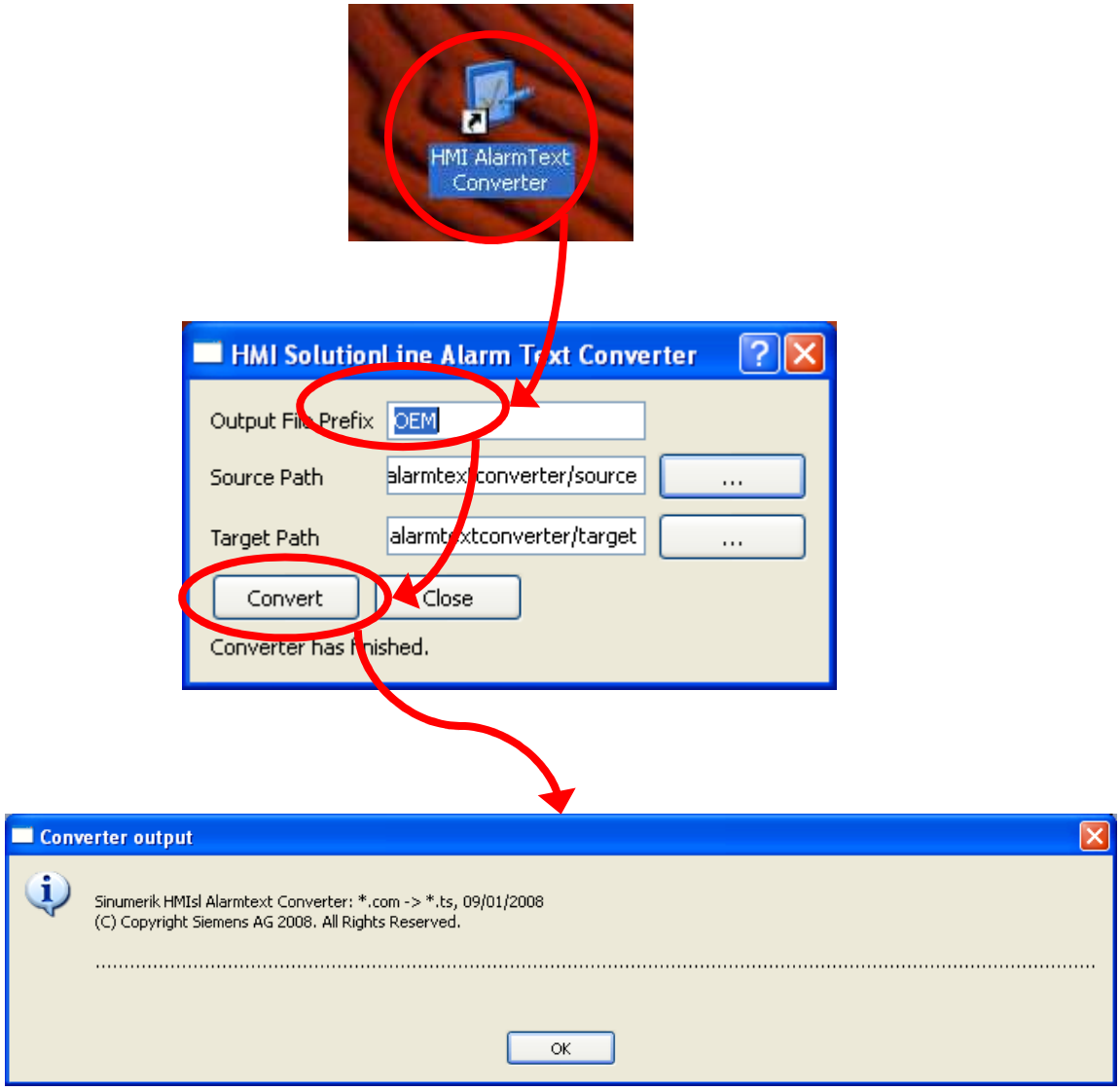
1. Copy alarm texts to source directory

Copy the alarm text files **ALARM_GR.COM** and **ALARM_UK.COM** from the **F:\user** directory of the PCU to the source text directory of the alarm text converter:



2. Convert alarm texts

Convert the alarm texts with the alarm text converter.
The prefix should be **"OEM"**.



3. Copy alarm texts from "target" to CF card

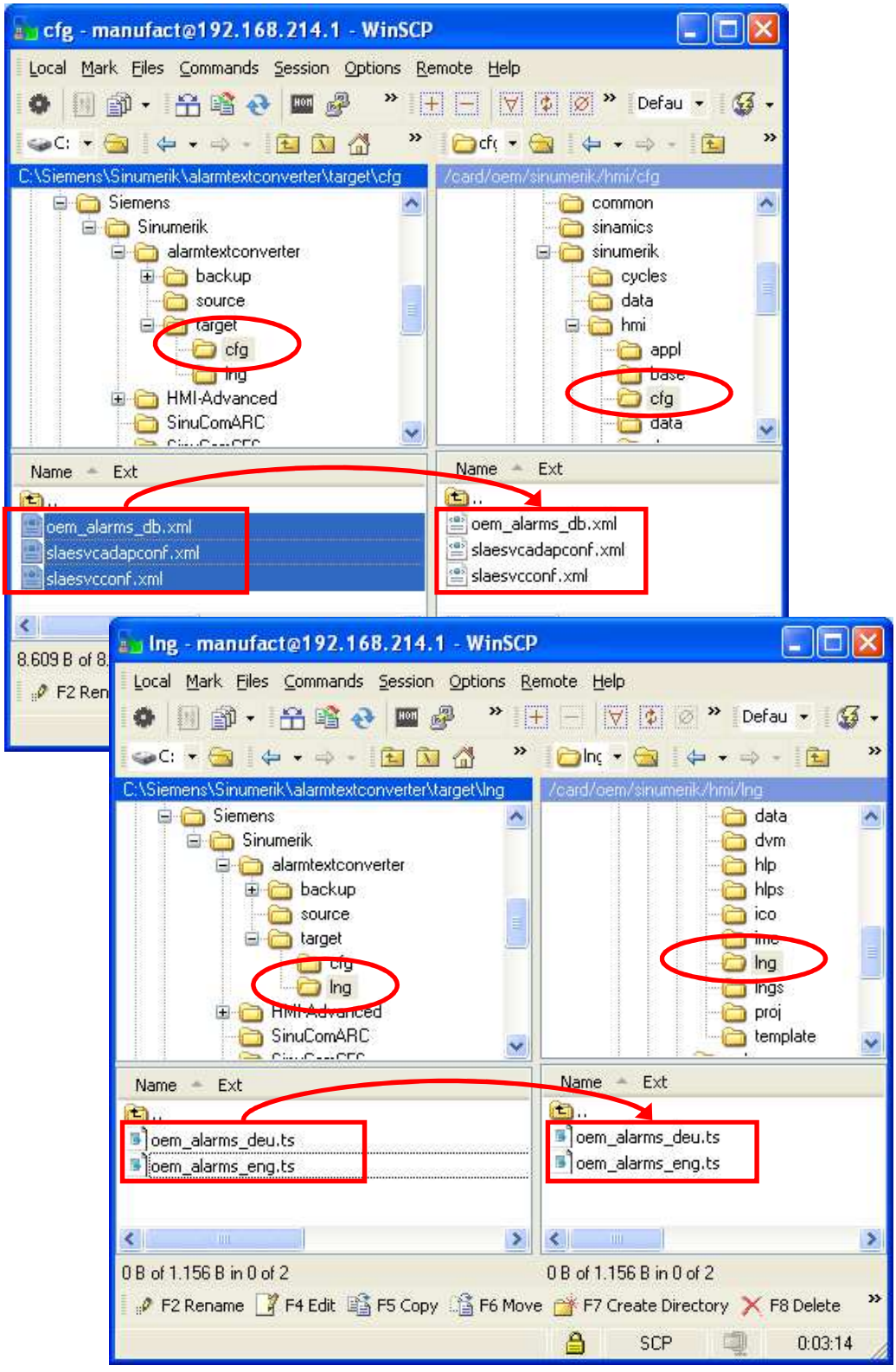
Use WinSCP (or RCS Commander) to copy the alarm texts from the **target** directory of the converter to the CF card.

Specifically, you must copy:

<converter path>\target\cfg*.* -> /card/oem/sinumerik/hmi/cfg/

<converter path>\target\lng*.* -> /card/oem/sinumerik/hmi/lng/

<converter path> is the installation path of the alarm text converter (default: C:\Siemens\SINUMERIK\alarmtextconverter).



4. FC10 call in OB1

Call FC10 for the alarm and message display in OB 1:

```
Netzwerk 3: Alarme und Meldungen
Kommentar:

      CALL  FC    10
       ToUserIF:=TRUE
       Quit    :=DB19.DBX20.2

      U     DB19.DBX   20.2
      R     DB19.DBX   20.2
```

5. Re-parameterize the FC19 call

Re-parameterize the FC19 call (MCP) to make the following test of the alarm display more straightforward.

Example: FeedHold -> 1st message feed stop (channel 1)
SpindleHold -> 1st message spindle stop (axis 4)

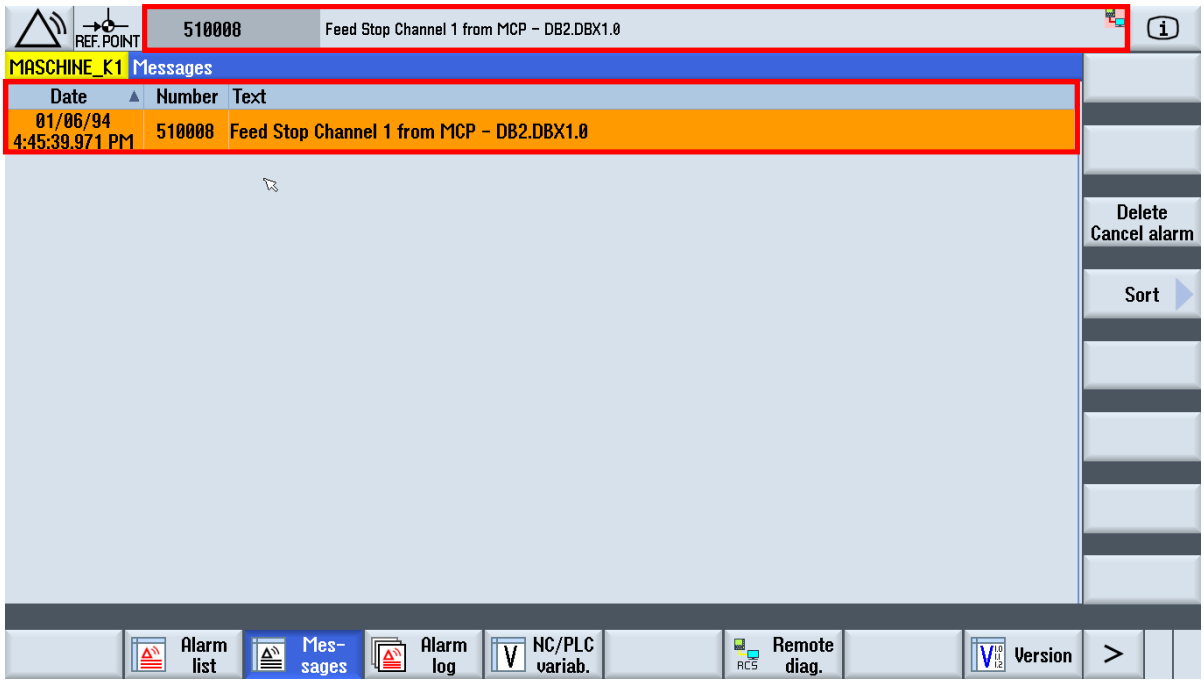
```
Netzwerk 2: MSTT
Kommentar:

     CALL  FC     19
      BAGNo       :=B#16#1
      ChanNo      :=B#16#1
      SpindleIFNo:=B#16#4
      FeedHold    :=DB2.DBX1.0
      SpindleHold:=DB2.DBX151.0
```

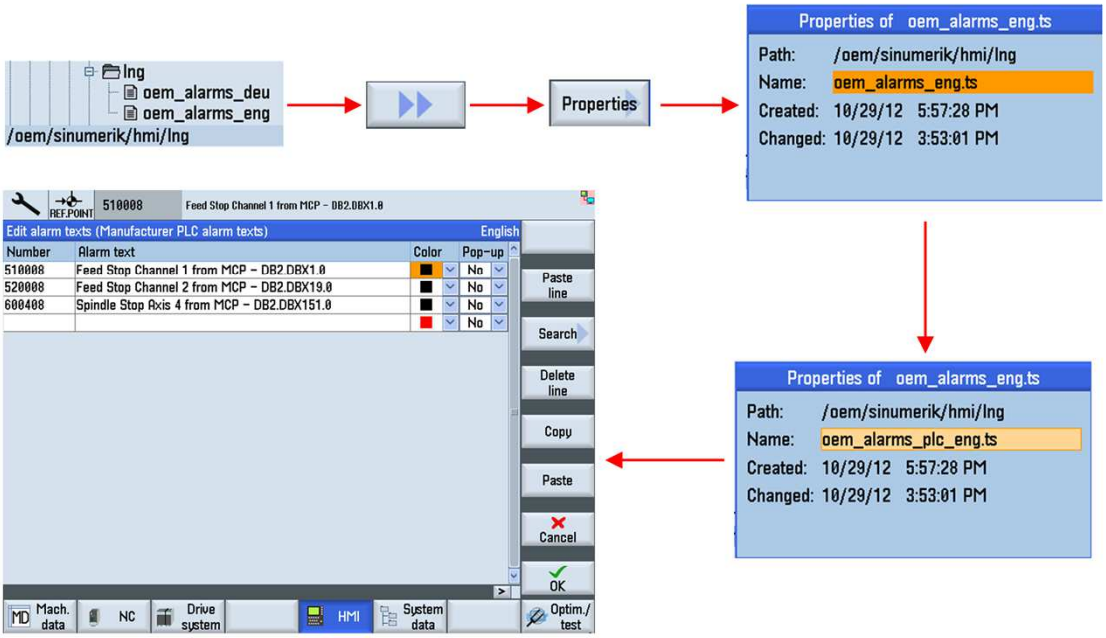
6. Power ON and test

Switch the control system off and on again.

Test the alarm display,
e.g. alarm 510008 "Feed stop ... of MCP.."
simply by pressing the Start / Stop buttons for feed or spindle on the MCP.



Additional exercise: Renaming the alarm text file for processing



SITRAIN © Siemens AG 2017

Objective

Unfortunately, the files provided by the alarm text converter cannot be accessed via a user-friendly editing screen form as is the case with alarms created on the control. However, if the conversion results (*.ts) are stored under the appropriate file name, they will be displayed correctly in the editor.

Steps

To do this, you must

1. rename the .ts files to **oem_alarms_plc_XXX.ts** (XXX=language code) and
2. delete the files you stored in directory …\cfg, and
3. open the alarm text editor once after restarting the HMI.

The texts will be displayed again when you close the editor!

Additional exercise: Changing alarm texts

Objective: One of the existing alarm texts is to be edited with the integrated editor in SINUMERIK Operate.

Sequence: Open the Start-up operating area and



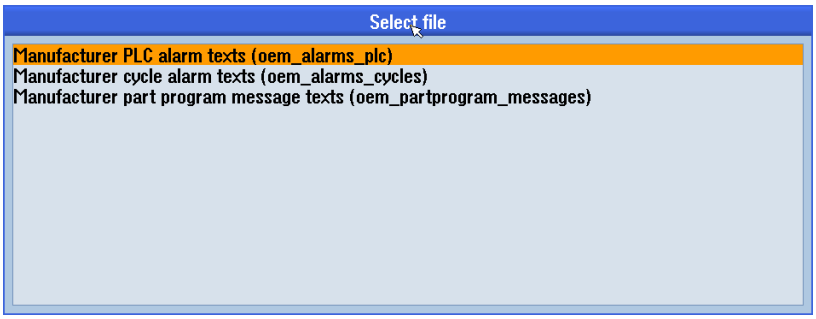
select the HMI menu.



Followed by alarm texts.



You can now select the individual areas for which you wish to edit user alarm texts.

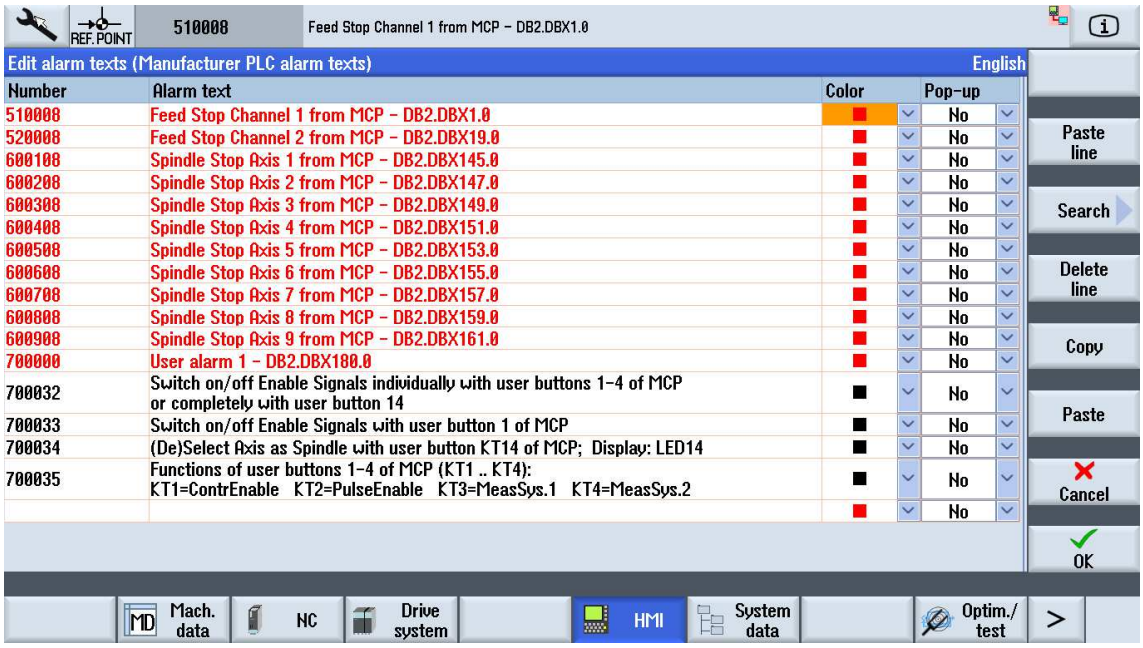


Select the

PLC user alarm texts

here.

You can now edit the entries in the editor window.
You can delete entries that are not required with **Delete line**, for example.

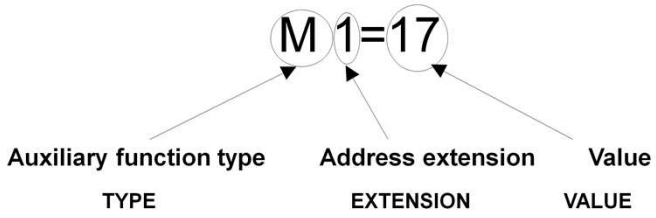


Next, restart SINUMERIK Operate.

Auxiliary functions

The following types of auxiliary function are available:

- **Miscellaneous function M**
- **Spindle function S**
- **Auxiliary function H**
- **Tool number T**
- **Tool offset D, DL**
- **Feedrate F**



Course NC-84SL-SIP Page 11 - 36 SITRAIN © Siemens AG 2017

Preassigned help function assignments

Group 1: The auxiliary functions M0, M1, M2, M17, and M30 are assigned to group 1 by default.

Group 2: The M functions M3, M4, M5, and M70 of the master spindle are assigned to group 2 by default.

Group 3: The S functions of the master spindle are assigned to group 3 by default.

Preassigned group properties

Auxiliary functions exhibit the following behavior if not defined in MD 11110: AUXFU_GROUP_SPEC[n]:

Normal acknowledgment by PLC

Transfer time:

- At end of block for group 1
- Prior to motion for group 2
- During motion for group 3

Implicit preprocessing stop

Machine data MD 10713: M_NO_FCT_STOPRE is used to define 15 M functions to execute an implicit preprocessing stop.
With an M function configured in MD 10713: M_NO_FCT_STOPRE, interpretation of the following part program line is delayed until the block containing the M function has been executed fully, i.e. acknowledged by the PLC.

Help functions
Overview and programming

SIEMENS

Miscellaneous function M	max. 5 per block M3, M2 = 5 M=QU(130)	M [0-99] = 0 - 99999999
Spindle function S	max. 3 per block S1000 S2 = 50.4 S[SPINO] = SPEED	S0 - S999999.999
Auxiliary function H	max. 3 per block H10 H1 = -1.45 H[NO] = VALUE	H [0..99] = -99999.999 - 99999.999
Tool number T	max. 1 per block T5 or T = "IDENT_NO"	T0 - T32000
Tool offset D	max. 1 per block D1 or D = DNOm	D0 - D9
Feedrate F, FA	max. 6 per block F1000 FA[X] = 100 F3 = 10	F0.001 - 999999.99
	max. 10 aux. functions per block	

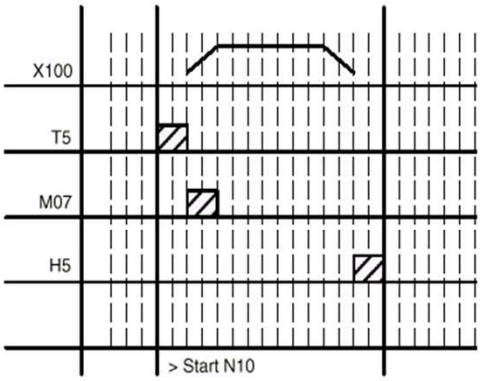
SITRAIN © Siemens AG 2017

Note

1) The type for the values of the H functions (integer or real) can be selected by the user in MD22110: AUXFU_H_TYPE_INT as of SW version 5.
2) Because of the limited display options on the operator panels, the values of type REAL are really limited to -999999999.9999 to 999999999.9999. Calculations in the NC kernel are performed with full REAL accuracy.
3) If MD 22110: AUXFU_H_TYPE_INT = 1 is set, the suggested REAL values are rounded before being transferred to the PLC. The PLC program must interpret the value in accordance with the MD setting.
4) If tool management is active, the meaning of the address extension can be controlled with MD.

Help functions
Transfer to the PLC

SIEMENS



Output prior to motion
Output during motion
Output at end of block
No output to PLC

Course NC-84SL-SIP Page 11 - 38 SITRAIN © Siemens AG 2017

Time of transfer

Machine data can be set for each type of auxiliary function to specify the time at which the auxiliary function parameters should be transferred to the PLC.

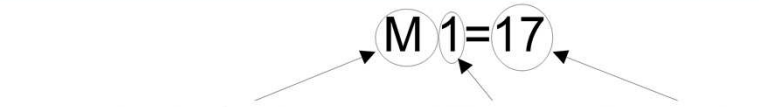
Machine data

MD 22200: AUXFU_M_SYNC_TYPE
MD 22210: AUXFU_S_SYNC_TYPE
MD 22220: AUXFU_T_SYNC_TYPE
MD 22230: AUXFU_H_SYNC_TYPE
MD 22240: AUXFU_F_SYNC_TYPE
MD 22250: AUXFU_D_SYNC_TYPE

0: Output prior to motion
1: Output during motion
2: Output at end of block
3: No output to the PLC

Auxiliary functions
Grouping

SIEMENS



MD 11100 AUXFU_MAXNUM_GROUP_ASSIGN = n	Indicates the total number of auxiliary functions to be split into groups. Maximum 256 definitions (n<=255).
MD 22000 AUXFU_ASSIGN_GROUP [0… n-1]	Number of auxiliary function group k can be selected in range 1 … 168. User ranges: **5 .. 9** and **13 ..71**
MD 22010 AUXFU_ASSIGN_TYPE [0 … n-1]	Definition of auxiliary function type **(TYPE)** of this definition (e.g. M)
MD 22020 AUXFU_ASSIGN_EXTENSION [0 … n-1]	Address extension **(EXTENSION)** used for this definition.
MD 22030 AUXFU_ASSIGN_VALUE [0 … n-1]	Auxiliary function value **(VALUE)**. The value -1 means: All AuxF with the same **TYPE** and **EXTENSION** belong to the group.
MD 22035 AUXFU_ASSIGN_SPEC[0 … n-1]	Behavior of auxiliary function with regard to transfer to the PLC (e.g. time of output: start of motion)

Course NC-84SL-SIP Page 11 - 39 SITRAIN © Siemens AG 2017

Grouping

The auxiliary functions are grouped:
- to output the help functions to the PLC during block search,
- to monitor the output time (e.g. end of block) and
- in the part program so that only one auxiliary function is programmed per group in the block.

Transfer properties

MD 22035 AUXFU_ASSIGN_SPEC

This MD is used to specify the output characteristics of the auxiliary functions in a group definition.

Bit 0=1 --> output duration 1 OB1 pass (normal auxiliary function)
Bit 1=1 --> output duration 1 OB40 pass, alarm-controlled (fast auxiliary function)
Bit 2 reserved
Bit 3=1 --> No output to PLC (must only be set as single bit)
Bit 4 reserved
Bit 5=1 --> Output before the motion
Bit 6=1 --> Output during the motion
Bit 7=1 --> Output at the end of the block

The MD must be defined for each existing auxiliary function group definition.

Alternative

MD 11110 AUXFU_GROUP_SPEC

This machine data can be used as an alternative (instead of MD 22035) if you do not require channel-specific transfer properties.

Example

AUXFU_ASSIGN_SPEC[6]=21H

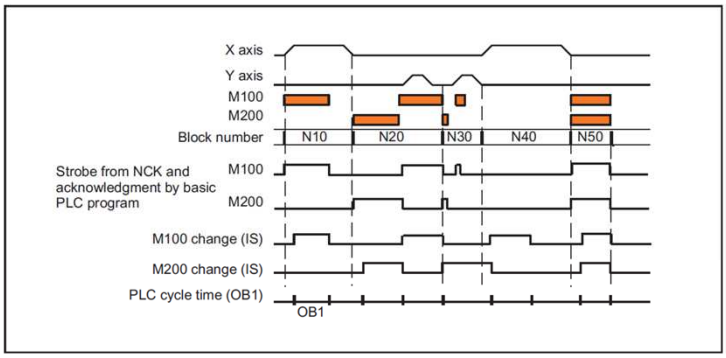
The signals of auxiliary function group 7 are output in the OB1 cycle prior to motion.

AUXFU_GROUP_SPEC[8]=82H

The signals of auxiliary function group 9 are output for the duration of OB 40 execution at the end of the block.

Auxiliary functions
Block change behavior

SIEMENS



Three potential block change responses:

- Block change with standard acknowledgment.
  Wait for acknowledgment before new block. Occurs in the OB1 cycle.
- Block change with fast acknowledgment with block change delay.
  Wait for acknowledgment before new block. Occurs in the OB40 cycle.
- Block change with fast acknowledgment without block change delay
  No wait for acknowledgment. Occurs in the OB40 cycle.

SITRAIN © Siemens AG 2017

Block change delay

MD 22100: AUXFU_QUICK_BLOCKCHANGE
= 0 with block change delay
= 1 without block change delay

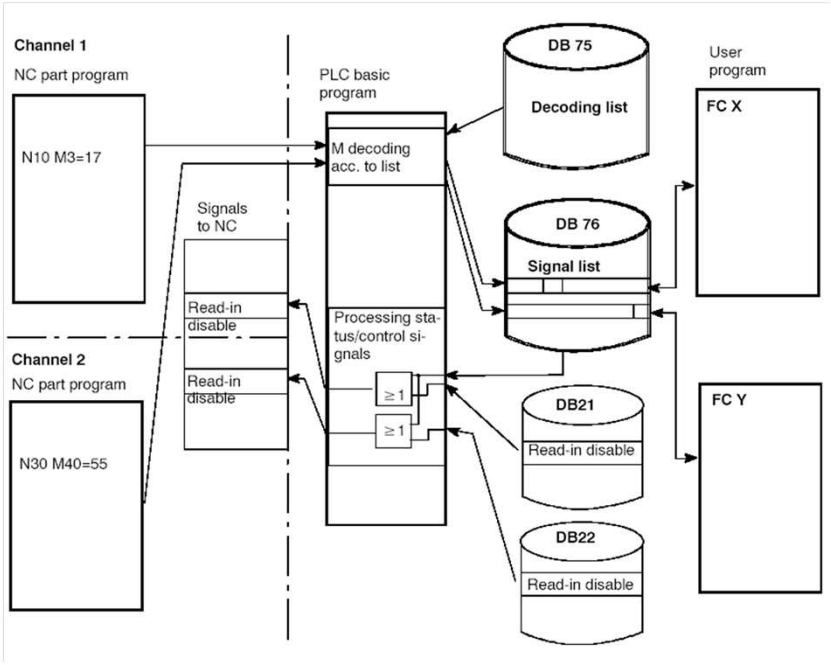
Note:

High-speed acknowledgment can either be set by means of NC programming (e.g. M=QU(10)) or M grouping with appropriate configuration of MD 22035 AUXFU_ASSIGN_SPEC.

Examples

N10 G94 G01 X50 M100	Ausgabe von M100 während der Bewegung.
N20 Y5 M100 M200	Ausgabe von M200 vor der Bewegung.
N30 Y0 M=QU(100) M=QU(200) N40 X0	Ausgabe als schnelle Hilfsfunktionen
N50 M100 M200	Keine Achsbewegungen, sofortige Ausgabe
M17	

Auxiliary functions
M decoding DB75 – DB76



Course NC-84SL-SIP Page 11 - 41 SITRAIN © Siemens AG 2017

When the M decoding function is activated in accordance with the list via the GP parameter of FB 1 "ListMDecGrp", the basic program can decode up to 256 M functions with an address extension.

When the bit is set in the signal list, the read-in disable is also set by the basic program. The read-in disable is reset once the user has reset all the bits in the signal list, thus acknowledging them.

The output of an M function decoded in the list as a high-speed auxiliary function does not result in a read-in disable.

Auxiliary functions
M decoding list DB75 – DB76

SIEMENS

DB 75

LAD/STL/FBD - [DB75 -- NC_S7APT\SINUMERIK840D\CPU 315-2 DP]

File Edit Insert PLC Debug View Options Window Help

Address	Name	Type	Initial value
0.0		STRUCT	
+0.0	MSigGrp	ARRAY[1..16]	
*0.0		STRUCT	
+0.0	MExtAdr	INT	0
+2.0	MFirstAdr	DINT	L#0
+6.0	MLastAdr	DINT	L#0
=10.0		END_STRUCT	
=160.0		END_STRUCT	

Example:
M20=100...M20=115 are to be decoded.
Group 1 shall be used.

MSigGrp[1].MExtAdr := 20;
MSigGrp[1].MFirstAdr:= L#100;
MSigGrp[1].MLastAdr := L#115;

Entered in the source mdeclist.STL > Compiled.

In FB1 parameter ListMDecGrp := 1 specified for first group.

The basic program generates DB76.
Byte 0 contains M20=100 107
Byte 1 contains M20=108...115.

Value 21 means: M20=100, M20=105 was called in the NC program.

DB76

0.0	STAT0[-32768]	BYTE	B#16#0	B#16#21
1.0	STAT0[-32767]	BYTE	B#16#0	B#16#0
2.0	STAT0[-32766]	BYTE	B#16#0	B#16#0

Course NC-84SL-SIP Page 11 - 42 SITRAIN © Siemens AG 2017

DB75

Signal	Type	Value range	Notes
MExtAdr	Int	0 to 99	Extended M address
MFirstAdr	DInt	0 to 99.999.999	First M address in group
MLastAdr	Dint	0 to 99.999.999	Last M address in group

DB76

DB for the auxiliary function signals.

This is generated automatically by the basic program.

A "1" signal in DB76 triggers a read-in stop of the NC and must be reset specifically by the PLC user program.

Auxiliary functions
Interface: Change signals

SIEMENS

DB 21 - 30	Signals from NCK channel (NCK→PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB **58**				M fct. 5 change /H2/	M fct. 4 change /H2/	M fct. 3 change /H2/	M fct. 2 change /H2/	M fct. 1 change /H2/
DBB **59**				M fct. 5 not decoded	M fct. 4 not decoded	M fct. 3 not decoded	M fct. 2 not decoded	M fct. 1 not decoded
DBB **60**		S fct. 3 quick	S fct. 2 quick	S fct. 1 quick		S fct. 3 change /H2/	S fct. 2 change /H2/	S fct. 1 change /H2/
DBB **61**		T fct 3 quick	T fct. 2 quick	T fct. 1 quick		T fct. 3 change/H2/ (SW 2 and higher)	T fct. 2 change/H2/ (SW 2 and higher)	T fct. 1 change /H2/
DBB **62**		D fct. 3 quick	D fct. 2 quick	D fct. 1 quick		D fct. 3 change/H2/ (SW 2 and higher)	D fct. 2 change/H2/ (SW 2 and higher)	D fct. 1 change /H2/
DBB **63**				DL fct. quick				DL fct. change
DBB **64**		H fct. 3 quick	H fct. 2 quick	H fct. 1 quick		H fct. 3 change /H2/	H fct. 2 change /H2/	H fct. 1 change /H2/
DBB **65**			F fct. 6 change /H2/	F fct. 5 change /H2/	F fct. 4 change /H2/	F fct. 3 change /H2/	F fct. 2 change /H2/	F fct. 1 change /H2/
DBB **66**				M fct. 5 quick	M fct. 4 quick	M fct. 3 quick	M fct. 2 quick	M fct. 1 quick
DBB **67**			F fct. 6 quick	F fct. 5 quick	F fct. 4 quick	F fct. 3 quick	F fct. 2 quick	F fct. 1 quick

SITRAIN © Siemens AG 2017

With just a few exceptions, the interface signals of the M functions can be found in the channel data block (DB21-DB30). The exceptions here are signals evaluated on an axis-specific basis, e.g. M3.

The modification signals are located in bytes 58 to 67. These are set to "1" for one cycle if an auxiliary function is triggered.

Auxiliary functions
Interface: Auxiliary functions M, S

SIEMENS

DB 21 - 30	Signals from NCK channel (NCK→PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBW **68**	Extended address M function 1 (binary) /H2/							
DBD **70**								
	M function 1 (binary) /H2/							
DBW **74**	Extended address M function 2 (binary) /H2/							
DBD **76**	M function 2 (binary) /H2/							
DBW **80**	Extended address M function 3 (binary) /H2/							
DBD **82**	M function 3 (binary) /H2/							
DBW **86**	Extended address M function 4 (binary) /H2/							
DBD **88**	M function 4 (binary) /H2/							
DBW **92**	Extended address M function 5 (binary) /H2/							
DBD **94**	M function 5 (binary) /H2/							
DBW **98**	Extended address S function 1 (binary) /H2/							
DBD **100**	S function 1 (REAL format) /H2/							
DBW **104**	Extended address S function 2 (binary) /H2/							
DBD **106**	S function 2 (REAL format) /H2/							
DBW **110**	Extended address S function 3 (binary) /H2/							
DBD **112**	S function 3 (REAL format) /H2/							

SITRAIN © Siemens AG 2017

M functions are programmed in the part program in the INTEGER format (8 decades plus leading sign).

Auxiliary functions
Interface: Auxiliary functions T, D

SIEMENS

DB 21 - 30	Signals from NCK channel (NCK→PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBW **116**	Extended address T function 1 (16 bit Int)							
DBW **118** DBD **118**	T function 1 (binary) /H2/ For 8-decade T nos., T function 1 (32 bit DINT) is used in DBD 118 (see note)							
DBW **120**	Extended address T function 2 (16 bit Int)							
DBW **122**	T function 2 (Int)							
DBW **124**	Extended address T function 3 (16 bit Int)							
DBW **126**	T function 3 (Int)							
DBB **128**								
DBB **129**	D function 1 (binary) /H2/							
DBW **130** DBB **130**	For 5-decade D nos., D function 1 (16 bit DINT) is used in DBD 130 (see note) Extended address D function 2 (8 bit Int)							
DBB **131**	D function 2 (8 bit Int)							
DBB **132**	Extended address D function 3 (8 bit Int)							
DBB **133**	D function 3 (8 bit Int)							
DBW **134**	Extended address DL function (16 bit Int)							
DBD **136**	DL function (REAL)							

SITRAIN © Siemens AG 2017

Programmed T functions are not output to the PLC when tool management is active.
8 decade T numbers are only available as T function 1.

Programmed D functions with name (e.g. D=EDGE_1) cannot be output to the PLC in ASCII format.

5-decade D numbers are only available as "D function 1".

The REAL format corresponds to the floating-point representation in STEP 7 (24-bit mantissa and 8-bit exponent). This floating point format supplies a maximum of 7 valid places.

Auxiliary functions
Interface: Auxiliary functions H, F

SIEMENS

DB 21 - 30	Signals from NCK channel (NCK→PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBW **140**	Extended address H function 1 (binary) /H2/							
DBD **142**	H function 1 (REAL or Dint) /H2/							
DBW **146**	Extended address H function 2 (binary) /H2/							
DBD **148**	H function 2 (REAL or Dint) /H2/							
DBW **152**	Extended address H function 3 (binary) /H2/							
DBD **154**	H function 3 (REAL or Dint) /H2/							
DBW **158**	Extended address F function 1 (binary) /H2/							
DBD **160**	F function 1 (REAL format) /H2/							
DBW **164**	Extended address F function 2 (binary) /H2/							
DBD **166**	F function 2 (REAL format) /H2/							
DBW **170**	Extended address F function 3 (binary) /H2/							
DBD **172**	F function 3 (REAL format) /H2/							
DBW **176**	Extended address F function 4 (binary) /H2/							
DBD **178**	F function 4 (REAL format) /H2/							
DBW **182**	Extended address F function 5 (binary) /H2/							
DBD **184**	F function 5 (REAL format) /H2/							
DBW **188**	Extended address F function 6 (binary) /H2/							
DBD **190**	F function 6 (REAL format) /H2/							

SITRAIN © Siemens AG 2017

F functions are programmed in the part program in REAL format.

The extended address of the F function contains an identifier with the following meaning:

0: Path feed,
1 - 31: Machine axis number for feedrate for positioning axes.

The AUX function data type is dependent on MD 22110: AUXFU_H_TYPE_INT.

Auxiliary functions
Interface: Decoded M commands

SIEMENS

DB 21 - 30	Signals from NCK channel (NCK→PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB 194	Dynamic M functions							
	M07	M06	M05 *	M04 *	M03 *	M02	M01	M00
DBB 195	Dynamic M functions /H2/							
	M15	M14	M13	M12	M11	M10	M09	M08
DBB 196	Dynamic M functions /H2/							
	M23	M22	M21	M20	M19	M18	M17	M16
DBB 197	Dynamic M functions /H2/							
	M31	M30	M29	M28	M27	M26	M25	M24
DBB 198	Dynamic M functions /H2/							
	M39	M38	M37	M36	M35	M34	M33	M32
DBB 199	Dynamic M functions /H2/							
	M47	M46	M45	M44	M43	M42	M41	M40
DBB 200	Dynamic M functions /H2/							
	M55	M54	M53	M52	M51	M50	M49	M48
DBB 201	Dynamic M functions /H2/							
	M63	M62	M61	M60	M59	M58	M57	M56
DBB 202	Dynamic M functions /H2/							
	M71	M70 *	M69	M68	M67	M66	M65	M64
DBB 203	Dynamic M functions /H2/							
	M79	M78	M77	M76	M75	M74	M73	M72
DBB 204	Dynamic M functions /H2/							
	M87	M86	M85	M84	M83	M82	M81	M80
DBB 205	Dynamic M functions /H2/							
	M95	M94	M93	M92	M91	M90	M89	M88
DBB 206	Dynamic M functions /H2/							
					M99	M98	M97	M96

SITRAIN © Siemens AG 2017

Spindle

M functions marked with * are not decoded in this bit field if a spindle is configured. In this case, these M functions are available extended M functions in DB21-30.DBB68 ff. and in the relevant axis DB DB31-61.DBB86 ff.

"Dynamic"

Dynamic M functions (M00 to M99) are set to "1" via the basic PLC program in one cycle.

The PLC operator can create static M functions from the dynamic M functions.

Exercise: Grouping M functions

Assign the 4 M commands M50...M53 to the following groups:

M50, M51	Group 5	Output prior to motion
M52	Group 6	Output during motion
M53	Group 7	Output after motion

Output to the PLC should be at high speed.

To check the part programming, switch on the outputs on the simulator with the M commands.

M50 Reset the outputs A32.0 / A32.1 / A32.2
M51 Set A32.0
M52 Set A32.1
M53 Set A32.2

Machine data

MD 11100 = Number of auxiliary functions 4

MD 22035[0]= Transfer fast, prior to block motion
MD 22035[1]= Transfer fast, prior to block motion
MD 22035[2]= Transfer fast, during block motion
MD 22035[3]= Transfer fast, after block motion

MD 22000[0]= Auxiliary function group 5
MD 22010[0]= Type (M command)
MD 22020[0]= Extension (zero)
MD 22030[0]= M command value (M50)

MD 22000[1]= Auxiliary function group 5
MD 22010[1]= Type (M command)
MD 22020[1]= Extension (zero)
MD 22030[1]= M command value (M51)

MD 22000[2]= Auxiliary function group 6
MD 22010[2]= Type (M command)
MD 22020[2]= Extension (zero)
MD 22030[2]= M command value (M52)

MD 22000[3]= Auxiliary function group 7
MD 22010[3]= Type (M command)
MD 22020[3]= Extension (zero)
MD 22030[3]= M command value (M53)

Interface

M50...53 DB21.DBX200.2...200.5

PLC: OB1

```
U   DB21.DBX  200.3     // AuxF is acknowledged in OB40 but transfer to the
S   Q    32.0           // interface does not occur until the next OB1 cycle
A   DB21.DBX  200.4
S   Q    32.1
A   DB21.DBX  200.5
S   Q    32.2
O   DB21.DBX   7.7      // Reset also with Reset (channel 1)
O   DB21.DBX  200.2
R   Q    32.0
R   Q    32.1
R   Q    32.2
```

NC program

```
G0 G90 X0
G1 X500 F6000 M51 M52 M53
M0
M50
```

Exercise: M functions with decoding list

In your PLC program, decode the M commands with a decoding list (DB75/76).

M400...M415 No output to simulator

M3=20...27 AB37

M7=50...57 AB38

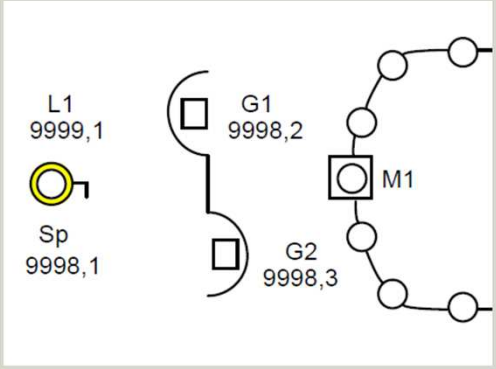
FB1 ListMDecGrp:=3

DB75 The file mdeclist.stl for creating DB75 is located in the source files of the basic PLC program.

0.0	MSigGrp[1].MExtAdr	INT	0	0
2.0	MSigGrp[1].MFirstAdr	DINT	L#0	L#400
6.0	MSigGrp[1].MLastAdr	DINT	L#0	L#415
10.0	MSigGrp[2].MExtAdr	INT	0	3
12.0	MSigGrp[2].MFirstAdr	DINT	L#0	L#20
16.0	MSigGrp[2].MLastAdr	DINT	L#0	L#27
20.0	MSigGrp[3].MExtAdr	INT	0	7
22.0	MSigGrp[3].MFirstAdr	DINT	L#0	L#50
26.0	MSigGrp[3].MLastAdr	DINT	L#0	L#57
30.0	MSigGrp[4].MExtAdr	INT	0	0
32.0	MSigGrp[4].MFirstAdr	DINT	L#0	L#0

DB76 Evaluation of the M commands as bit information. DB76 is automatically generated according to the settings in DB75.

SIEMENS



Chapter 12

Basic settings for tool management

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
What is tool management? 3
What are the usual types of tool magazine? 4
Structure of tool management 5
Basic machine data for tool management 6
Machine data for tool management: Number of tools, etc. 7
Machine data for tool management: Configuration 8
Selecting the magazine configuration 9
Steps for commissioning the magazine configuration 10
Activating TO data 11
Backing up TO data on network / USB 12
Magazine configuration ini file 13
Result: Tool magazine and tool view in SINUMERIK Operate 14
Tool change sequence for milling and turning machines with programGUIDE 15
Tool preparation and tool change with ShopMill/ShopTurn 16
Machine data for tool management: Tool change; reset behavior 17
Other machine data for technological M functions 18
Tool change cycle L6.SPF 19
Note for PLC programming of the individual basic functions 20
Exercise 1: Activate machine data for tool management 21
Exercise 2: Generate tool magazine configuration in the NC 24
Exercise 3: Activate tool change cycle L6 29

Learning objectives

The participant will ...

... Gain an overview of the basic settings for tool management in the SINUMERIK 840D solution system

... Learn how to make the basic settings for tool management based on a set example

Note

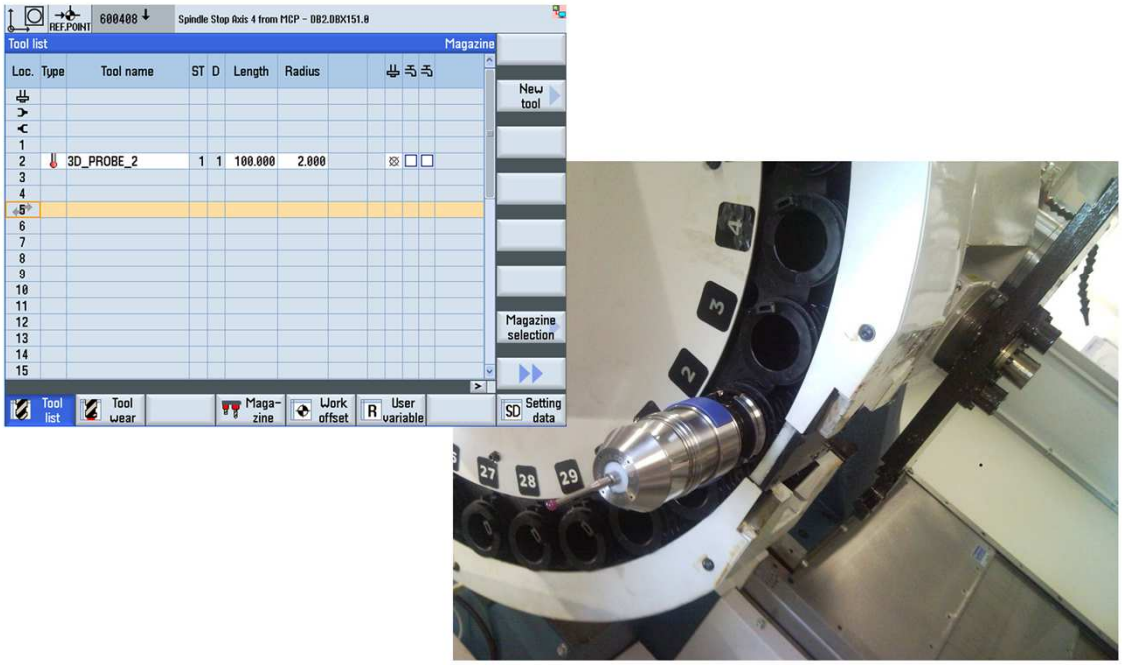
Basic functions, such as T,S,M, in SINUMERIK Operate require active tool management.

However, this chapter is not a substitute for the training course on tool management commissioning!

SITRAIN © Siemens AG 2017

Notes

What is tool management?



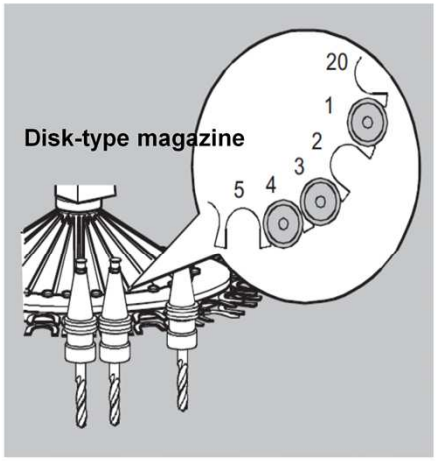
 SITRAIN © Siemens AG 2017

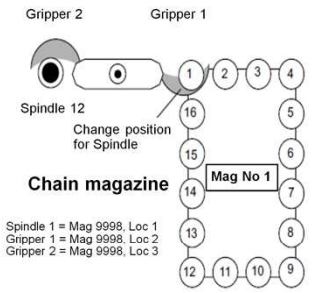
Tool management is needed in machining production to uniformly organize information about existing tools and to integrate it into the environment. The tool data are stored in a database and acquired and used with the tool management software. Unlike a general solution for managing resources, a tool management program contains specialized technical data fields, graphics, and parameters that are required for use in the production process.

Tool management is divided into the documentation for tools (master data) and logistics (motion data).

In tool management, logistics is concerned with the inventory and the location of the tools.

What are the usual types of tool magazine?









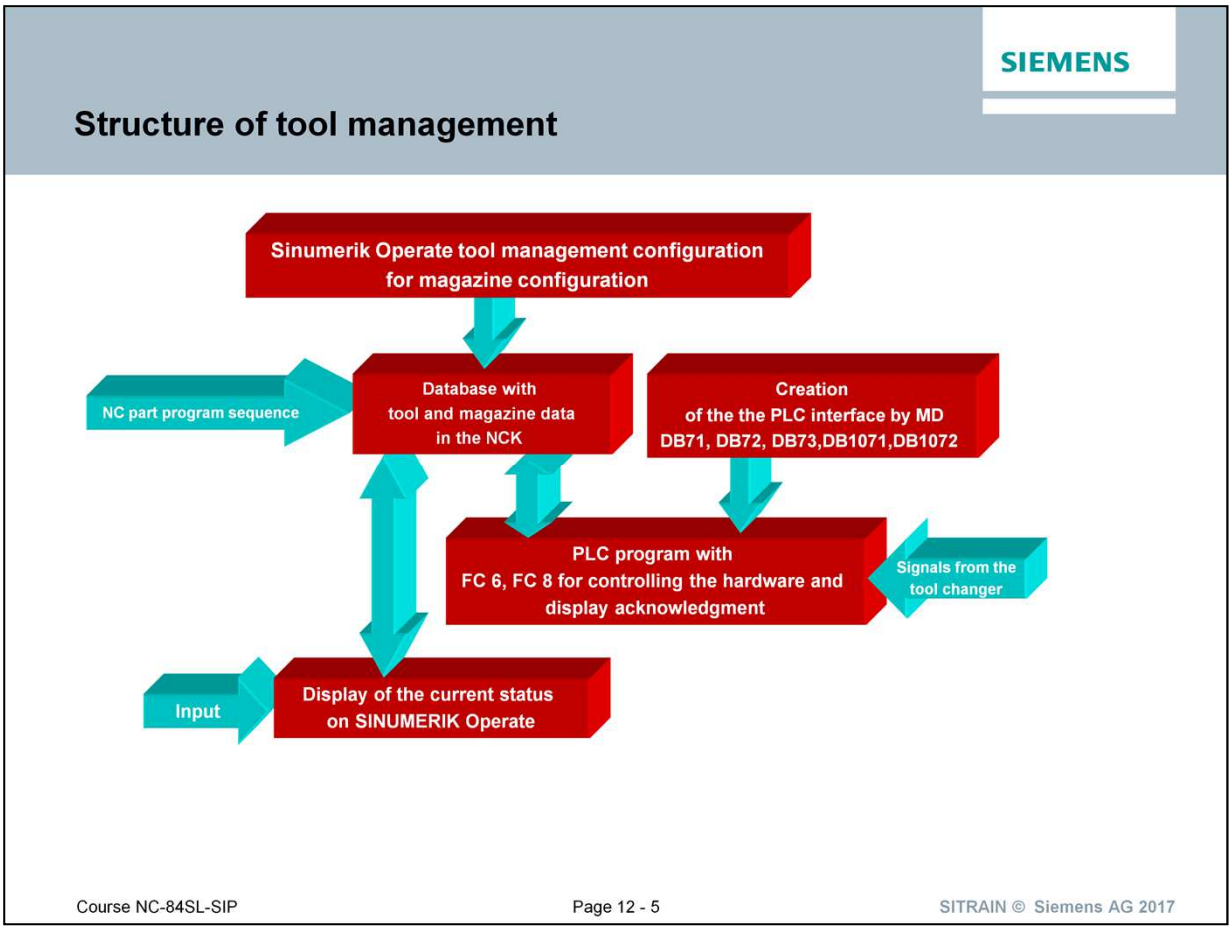
Course NC-84SL-SIP Page 12 - 4 SITRAIN © Siemens AG 2017

Chain magazine A chain magazine consists of a chain that has adapters for tools. The length of the chain determines the number of tools it can hold. Positioning moves a tool adapter, for example, to the change location or to the load location. The number of locations is stored with the definition of the management data.

Disk-type magazine The number of possible positions of a disk-type magazine depends on the size of the disk and the size of the tool adapters. A disk-type magazine is mechanically simpler than a tool chain, but the largest that can reasonably be built does not have as many magazine locations as a chain magazine.

Box magazine consists of multi-row tool adapter pallets. These can generally be completely prepared outside of the machine, which allows for relatively easy workpiece changes/tool set changes.

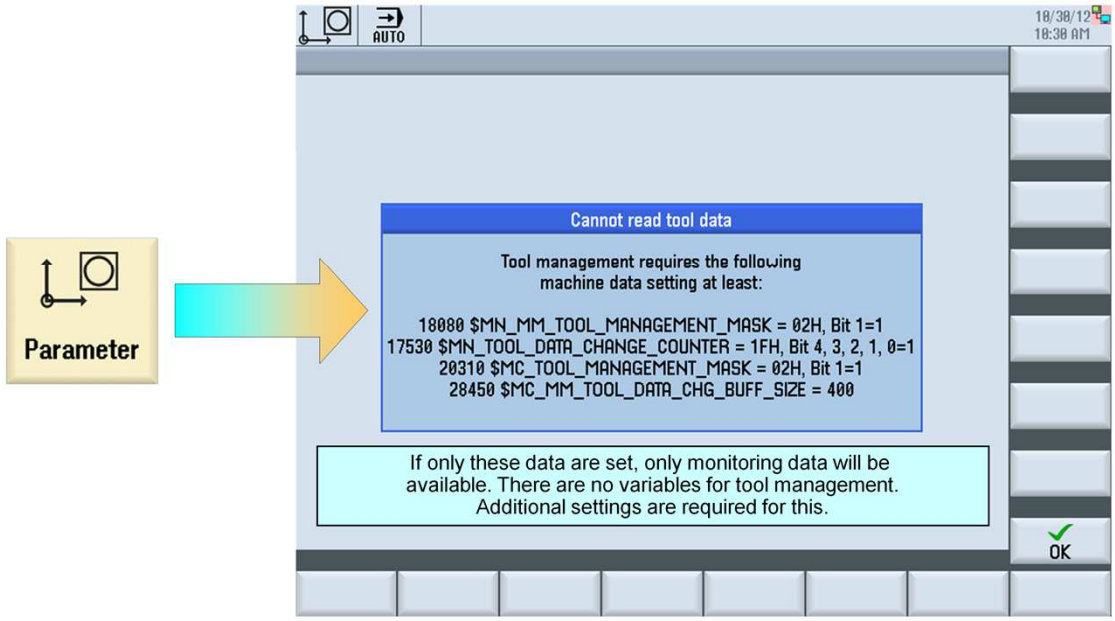
Tool turret A tool turret is the changing device typically used on a turning machine. The tools are mounted on and also remain on a drum-like adapter. A tool change does not first have to be prepared with a positioning task, but is performed directly when the "T" command is received. Depending on the tool that is selected, the turret retracts, rotates, and then moves back up to the workpiece for further machining. The tool change can be controlled either by the PLC or by the NC (e.g. using synchronized actions in the change cycle).



PLC interface

DB 71 for loading, unloading, and relocating tools.
DB 72 for tool change preparation and tool change
DB 73 for tool turret
DB 1071, DB 1072, DB 1073 for multitools

Basic machine data for tool management



 SITRAIN © Siemens AG 2017

When the operating area parameter is opened in SINUMERIK Operate, the screen shown above appears if tool management has not been commissioned. Entering the given machine data does not lead to the generation of system variables for tool management, but only generates monitoring data.

MD 19320 $ON_TECHNO_FUNCTION_MASK
Bit 25 = 1: Tool monitoring
Bit 4 = 1: Tool monitoring and magazine management

MD 18080 $MN_MM_TOOLMANAGEMENT_MASK = 40D h
Bit 0 = 1: Provide magazine management data
Bit 1 = 1 : Provide monitoring data.
Bit 3 = 1 : Consider adjacent location
Bit 10 = 1 : The multitool function is available

MD 20310 $MC_TOOL_MANAGEMENT_MASK = 2B h
Bit 0 = 1 : Magazine management is active
Bit 1 = 1 : Monitoring functions are active
Bit 3 = 1 : Adjacent location handling
Bit 9 = 1: Simulation of the PLC acknowledgments – with simulation = 1

Note Bits 0 - 3 in MD 18080 and MD20310 must always be the same.

MD 17530 $MN_TOOL_DATA_CHANGE_COUNTER = 1F h
Bit 0 = 1 : Tool status change
Bit 1 = 1 : Tool remaining quantity change
Bit 2 = 1 : Tool data in change service
Bit 3 = 1 : Magazine data in change service
Bit 4 = 1 : ISO data in change service

MD28450 $MC_MM_TOOL_DATA_CHG_BUF_SIZE = 400

Machine data for tool management: Number of tools, etc.

SIEMENS

- Number of tool magazines, magazine locations, tools, cutting edges:

MD 18084	$MC_MM_NUM_MAGAZINE = 3	Standard = 3, number of magazines that the NCK can manage. (at least 3 magazines incl. buffer and loading magazine.
MD 18086	$MN_MM_NUM_MAGAZINE_LOCATIONS =35	Standard = 30, number of all magazine locations that NCK can manage.
MD 18082	$MN_NUM_TOOL = 100	Standard = 30, number of tools that the NCK can manage. A tool has at least one cutting edge.
MD 18100	$MN_MM_NUM CUTTING_ EDGES_IN_TOA = 200	Standard = 30, number of tool cutting edges in the TO area

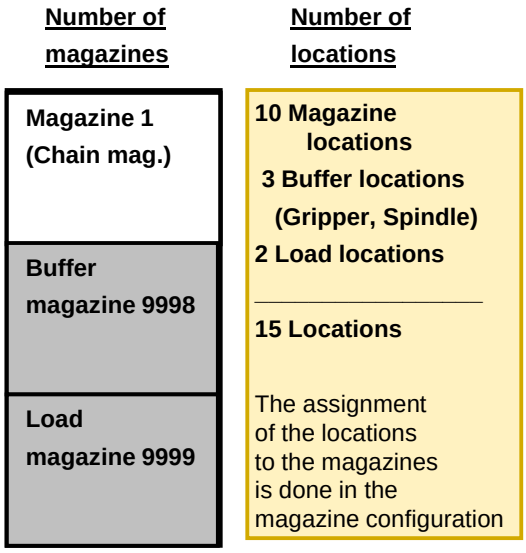
- Corresponding data for multitools:

MD 17504	$MN_MAX_TOOLS_PER_MULTITOOL = 6	Standard = 6, max. number of tool locations per multitool
MD 18083	$MN_NUM_MULTITOOL = 15	Standard = 15, number of multitools managed by the NCK
MD 18085	$MN_NUM_MULTITOOL_LOCATIONS = 90	Standard = 30, number of multitool locations managed by the NCK

SITRAIN © Siemens AG 2017

MD 18084
MD 18086

Note: Buffer and load locations must be added
MD 18084 Number of tool magazines
MD 18086 Number of magazine locations



NOTICE After changes have been made to data 18xxx or 28xxx, which configure the memory, the NC must be backed up and re-imported.

.

Machine data for tool management: Configuration

SIEMENS

- Option

MD 19320	$ON_TECHNO_FUNCTION_MASK	Bit 4: Tool management with magazine management Bit 25: Tool management without magazine management

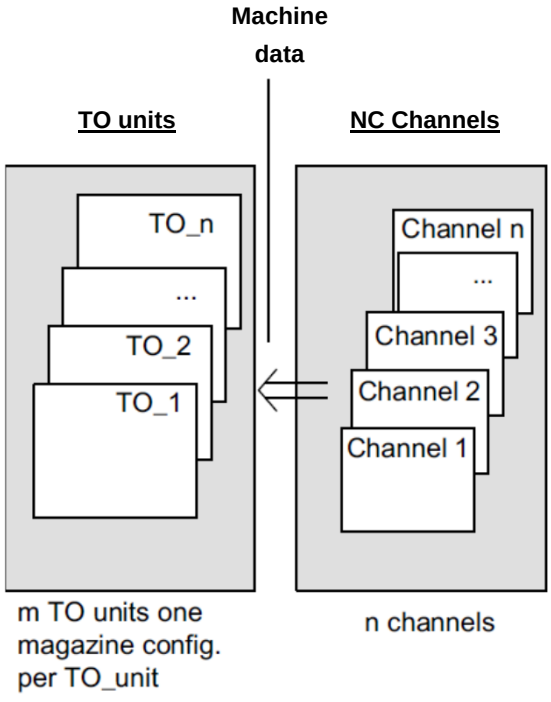
- Basic configuration

MD 17530	$MN_TOOL_DATA_CHANGE_COUNTER='H1F'	Identify tool data change for HMI: Bit 0: Tool status Bit 1: Tool remaining quantity Bit 2: Tool data Bit 3: Tool magazine data Bit 4: ISO data
MD 18080	$MN_MM_TOOL_MANAGEMENT_MASK='HB'	Reserve memory for tool management, examples: Bit 0: Magazine management Bit 1: Monitoring data Bit 3: Consider adjacent location
MD 20310	$MC_TOOL_MANAGEMENT_MASK	Activate tool management Bit 0: Magazine management data Bit 1: Monitoring data Bit 3: Consider adjacent location Bit 9: Simulate PLC acknowledgments Bit 14: Automatic tool change after reset and start (for TSM)
MD 28085	$MC_MM_LINK_TOA_UNIT = 1	The TO unit "1" is assigned to the channel.

Course NC-84SL-SIP Page 12 - 8 SITRAIN © Siemens AG 2017

MD 28085

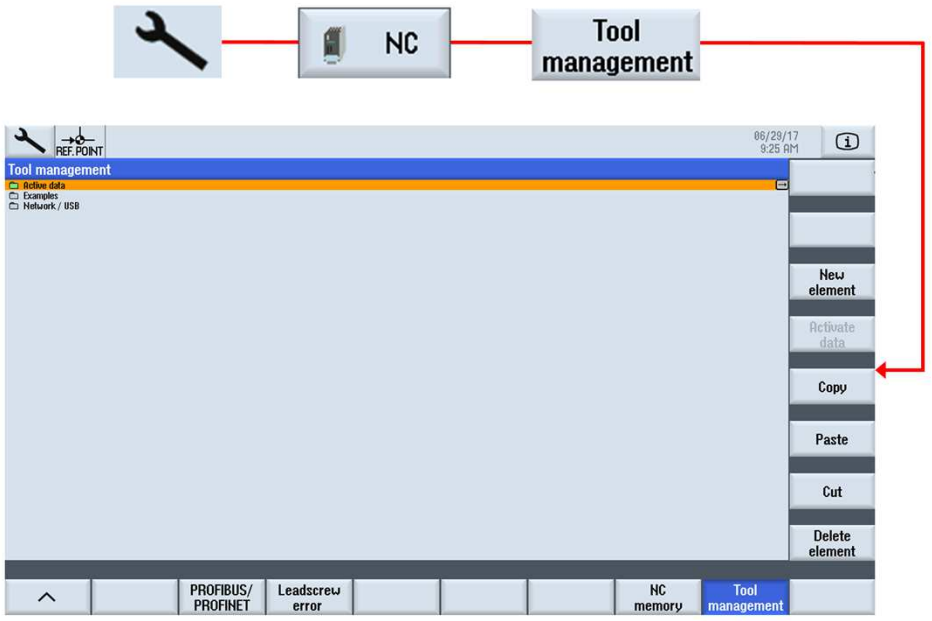
Assignment of the TO unit to the channel. The maximum number of TO units equals the number of channels.
You can also view a tool magazine as a TO unit.
If each machining channel has its own tool magazine, each tool magazine has data that is saved as a TO unit.
If all of the channels use the same tool magazine, the data 28085 has the same value in every channel, e.g. 1.



NOTICE

After changes have been made to machine data 18xxx or 28xxx, which configure the memory, the NC must be backed up and re-imported.

Selecting the magazine configuration



Course NC-84SL-SIP Page 12 - 9 SITRAIN © Siemens AG 2017

Objective

The aim of magazine configuration is to provide the configuration data for representation of the magazine under softkey: Parameter.

Selection

The "Magazine configuration" tool is selected in the following menu:

> OA: "Setup" > softkey: "NC" > softkey: "Tool management"

Magazine management

That requires activation of the magazine management: MD 18080 Bit 0 = 1 and MD 19320 Bit 4 = 1 (previous option bit for magazine management)

Warning

Because activation of the magazine configuration (**softkey: "Activate data"**) deletes data, a warning is displayed to us at this point.

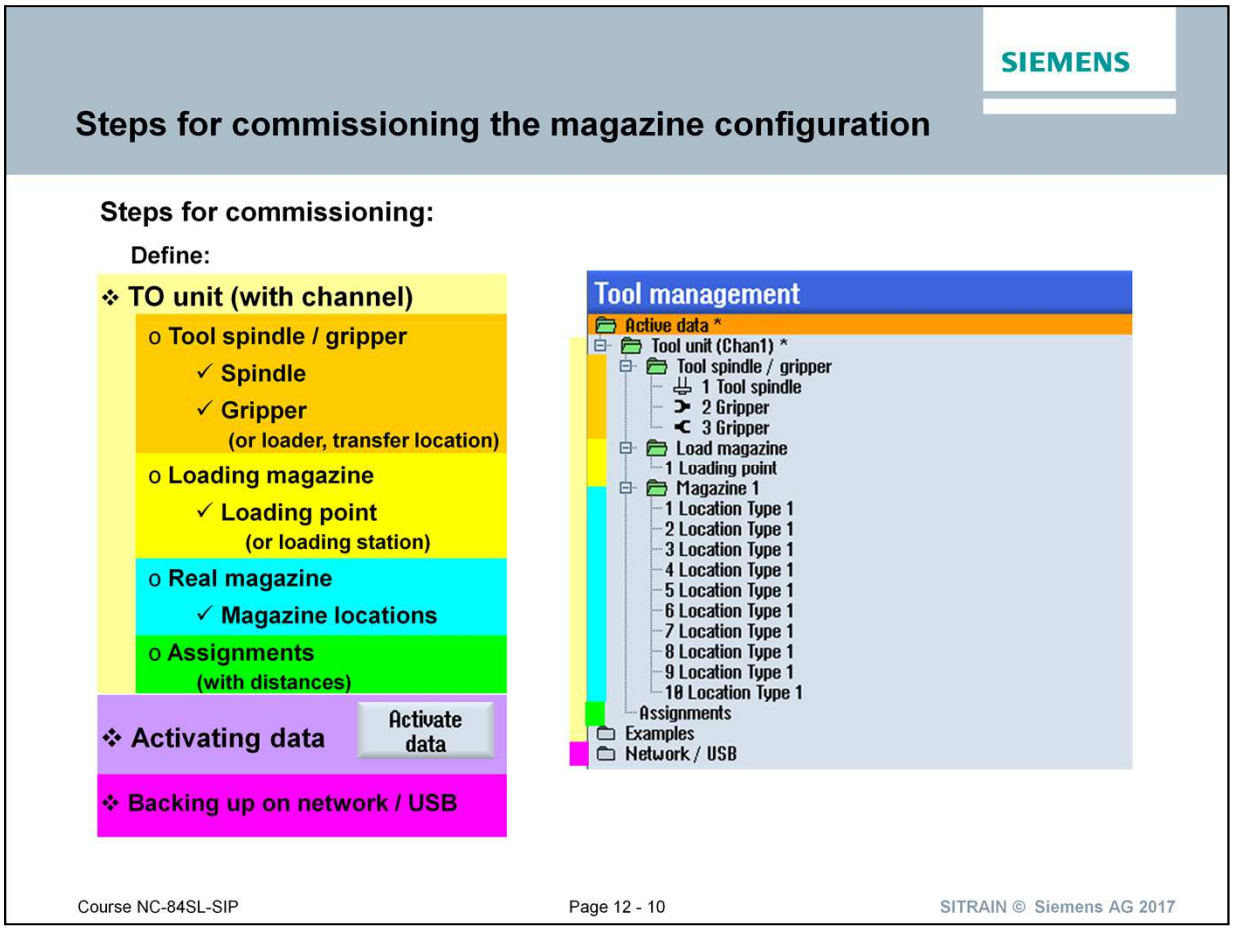
The affected data are:

- Hierarchies
- Wear groups
- Magazine status

  This is, for example, magazine assignment (**all existing tools!**)

Not supported

The magazine configuration in SINUMERIK Operate currently (SW 4.7) does not support definition of location types, for example!



How it works

The magazine configuration is defined step by step.

As we can see, a file "**new_TO_TMA.ini"** is built up (that is not visible at any time).

At the end, this file is started in an assigned channel of the TO unit as a program with the operating step:

> Softkey: "Activate data"

Individual steps

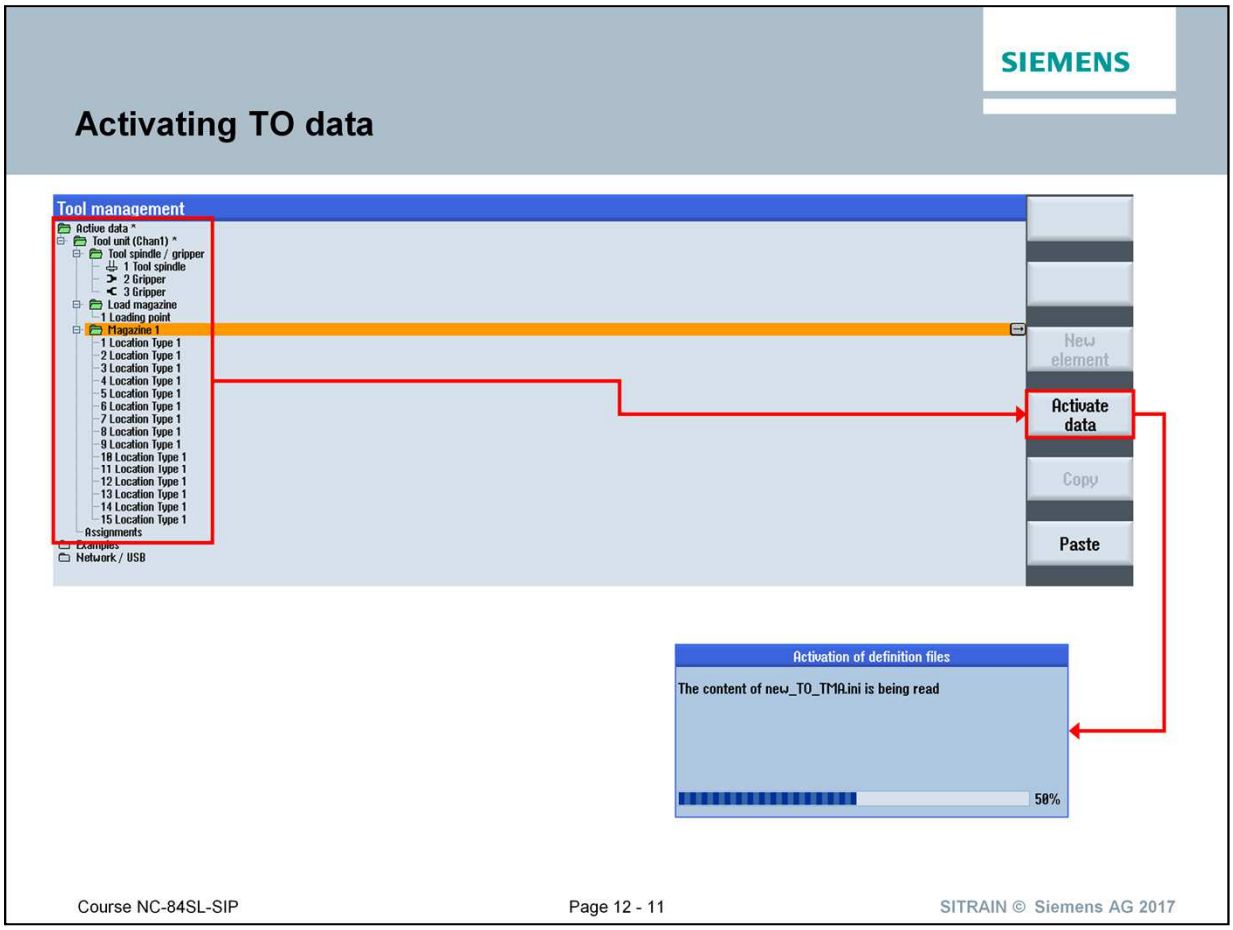
The following elements must be declared or actions performed:

- **TO unit (with channel)**
- **Tool spindle / gripper**
- **Loading magazine**
- **Real magazine**
- **Assignments (with distances)**
- **Activating data**

Backing up on network / USB

Note If you exit this configuration menu without activating the data, all changes will be lost!

The current tool assignment will be lost after activation!



Activating the machine data

The magazine data entered are still not active.

This fact is indicated with the icon " ***** "
in the elements (TO unit, etc.).

To activate, use the **> softkey: "Activate data"**

This reads in a file with the name "**new_TO_TMA.ini**".

Notes

- new_TO_TMA.ini

However, the "**new_TO_TMA.ini"** file used is only visible as a backup on the network or USB (where the filename can be assigned for copying/insertion).

On the NCK, after activation of the data, a corresponding

TO_TMA.INI (under "Magazine data complete")
or **TO.INI** (under "Tool/magazine data complete")

- DB4

The PLC data block DB 4 required for older software versions is superfluous as of NCK-SW 4.7.

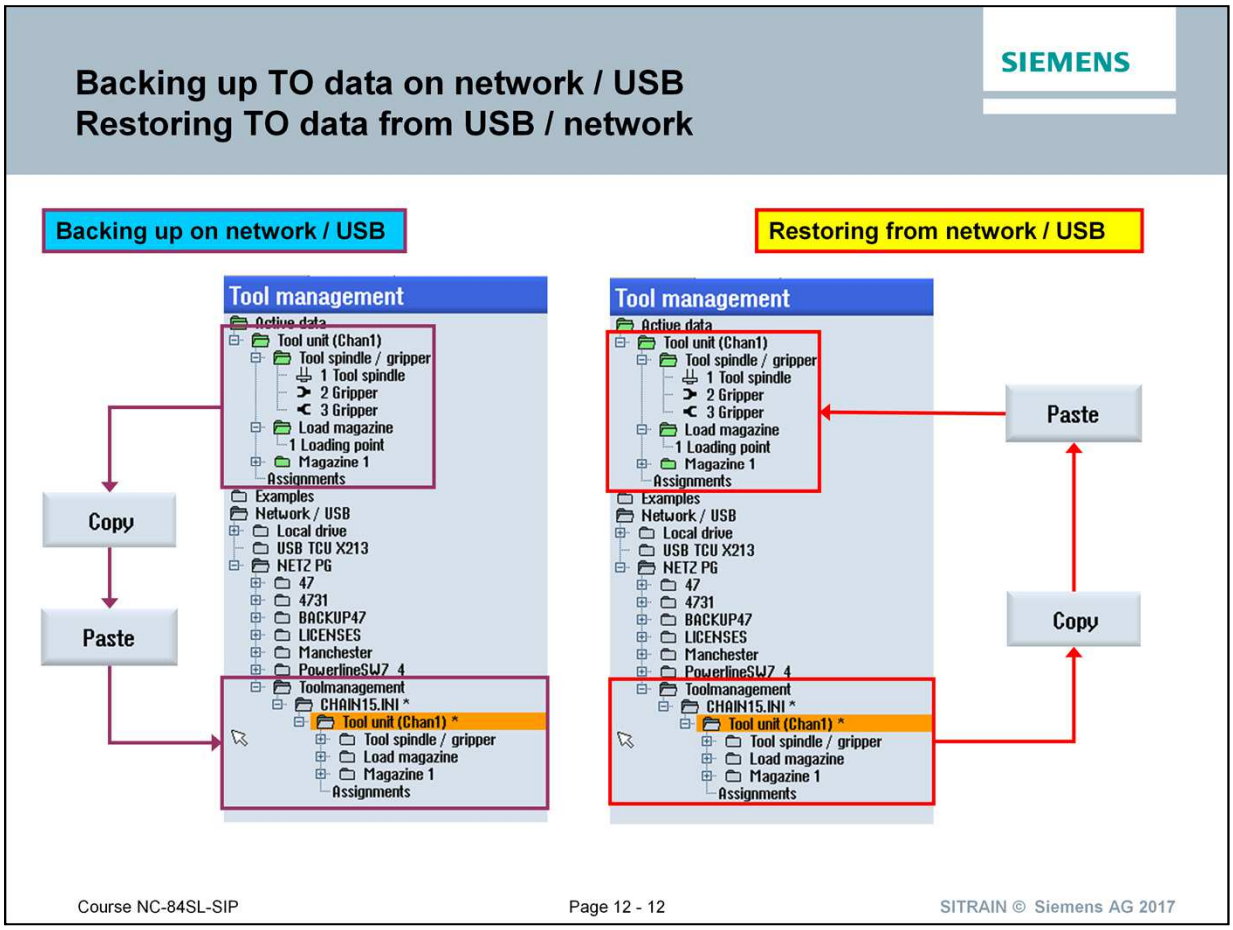
If it exists, DB 4 is nevertheless still evaluated!

For that reason, DB 4 can be removed from the PLC project in any case to avoid accumulating obsolete data!

Warning

Certain data such as hierarchies, wear groups, and the magazine status (tool assignment) will be deleted on activation!

Unfortunately, at the time of activation, the corresponding warning was not output!
The relevant message box was therefore added to this slide as a reminder!



Comments

Whether the TO data are active or not is immaterial in this case!

The created data in this overview look like directories, but are really ini files!

Reactivating TO data from a backup

This can be done by copy and paste.

Magazine configuration ini file

```
N10 METRIC
N20 ;
N30 ;
N40 ; ------------------------------------------------------------
N50 ; Tool magazine configuration: Chain_1   //  Name of the ini
file
N60 ; ------------------------------------------------------------
N70 ;
N80 ; Delete old data
N90 ;
N30 $TC_MAP1[0]=0
N40 $TC_DP1[0,0]=0
;
N130 ; Configuration
N20 CHANDATA(1)
;
N150 $TC_MAMP1="Chain_1"    // Name of tool magazine
Chain_1
N160 $TC_MAMP2=12289        // Search strategy
N170 ;
N180 ; Real magazine
N190 ;
N190 $TC_MAP1[1]=1
N200 $TC_MAP2[1]="Chain_10"
N210 $TC_MAP3[1]=17
N220 $TC_MAP6[1]=1
N230 $TC_MAP7[1]=10
N240 $TC_MAP10[1]=12289
N260 ;
;
; Magazine locations 1 - 10
;
N250 $TC_MPP1[1,1]=1
N260 $TC_MPP2[1,1]=1
N270 $TC_MPP3[1,1]=1
N280 $TC_MPP4[1,1]=2
N290 $TC_MPP5[1,1]=0
N300 $TC_MPP1[1,2]=1
...
...
N700 $TC_MPP1[1,10]=1
N710 $TC_MPP2[1,10]=1
N720 $TC_MPP3[1,10]=1
N730 $TC_MPP4[1,10]=2
N740 $TC_MPP5[1,10]=0

;
; Buffer
;
N750 $TC_MAP1[9998]=7
N760 $TC_MAP2[9998]="Tool buffer"
N770 $TC_MAP3[9998]=17
N780 $TC_MAP6[9998]=1
N790 $TC_MAP7[9998]=3
N800 $TC_MAP10[9998]=0
;
N810 $TC_MPP1[9998,1]=2
N820 $TC_MPP2[9998,1]=0
N830 $TC_MPP3[9998,1]=0
N840 $TC_MPP4[9998,1]=2
N850 $TC_MPP5[9998,1]=1
N860 $TC_MPP1[9998,2]=3
N870 $TC_MPP2[9998,2]=0
N880 $TC_MPP3[9998,2]=0
N890 $TC_MPP4[9998,2]=2
N900 $TC_MPP5[9998,2]=1
N910 $TC_MPP1[9998,3]=3
N920 $TC_MPP2[9998,3]=0
N930 $TC_MPP3[9998,3]=0
N940 $TC_MPP4[9998,3]=2
N950 $TC_MPP5[9998,3]=2
;
; Loading magazine
;
N960 $TC_MAP1[9999]=9
N970 $TC_MAP2[9999]="Loading magazine"
N980 $TC_MAP3[9999]=17
N990 $TC_MAP6[9999]=1
N1000 $TC_MAP7[9999]=2
N1010 $TC_MAP10[9999]=0
;
N1020 $TC_MPP1[9999,1]=7
N1030 $TC_MPP2[9999,1]=0
N1040 $TC_MPP3[9999,1]=0
N1050 $TC_MPP4[9999,1]=2
N1060 $TC_MPP5[9999,1]=1
N1070 $TC_MPP1[9999,2]=7
N1080 $TC_MPP2[9999,2]=0
N1090 $TC_MPP3[9999,2]=0
N1100 $TC_MPP4[9999,2]=2
N1110 $TC_MPP5[9999,2]=2

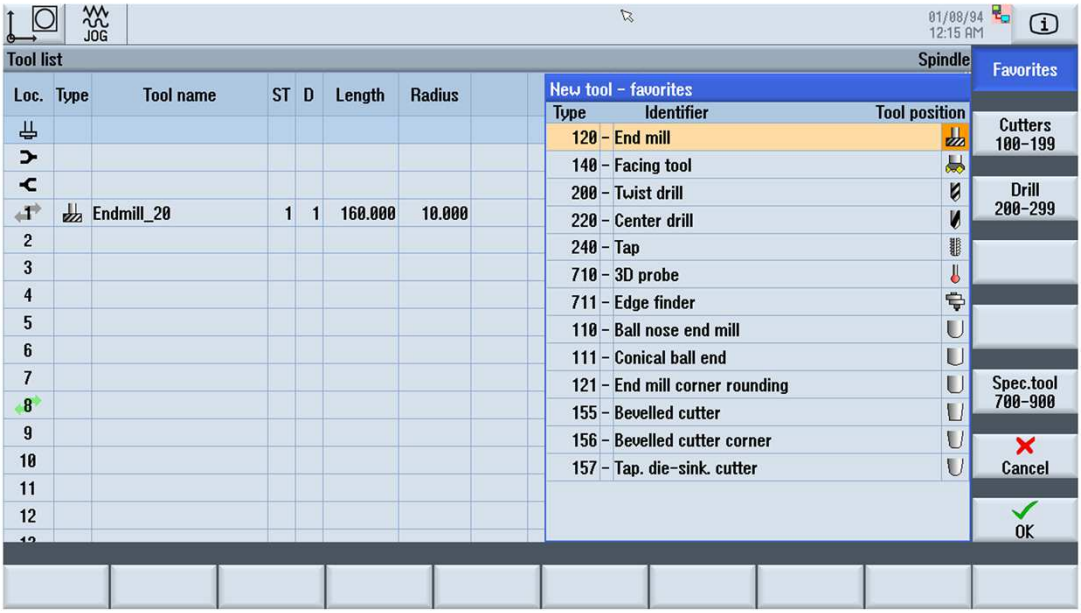
;
N1120 ; spindle assignment
;
N1130 $TC_MLSR[2,1]=0
N1140 $TC_MLSR[3,1]=0
;
N1150 $TC_MDP1[1,1]=0
N1160 $TC_MDP1[1,2]=4
;
N1170 ; distances
;
N1180 $TC_MDP2[1,1]=0
N1190 $TC_MDP2[1,2]=0
N1200 $TC_MDP2[1,3]=0
;
N1210 M17
```


SITRAIN © Siemens AG 2017

Variables of the tool magazine configuration file

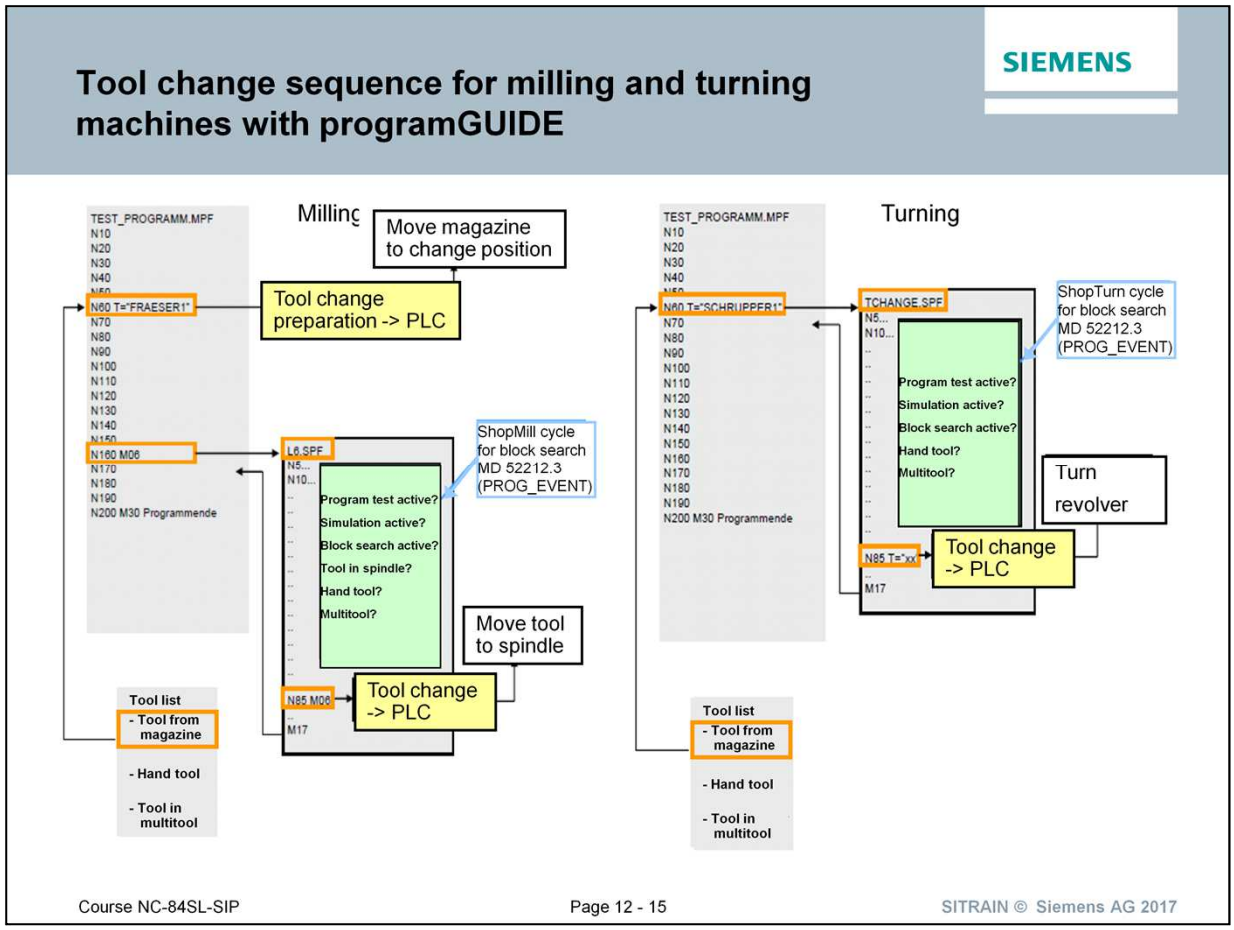
Parameter	Meaning
$TC_MAMP1	Identifier of the configuration magazine
$TC_MAMP2	Type of tool search (bits 0...7) and type of empty location search (bits 8...15) - Bit 0: active tool / min duplo - Bit 12: symmetric actual location - Bit 13: 1:1change
$TC_MAP1	Type of magazine (1= Chain, 3 = Turret, 5 = Box magazine, 7 = Tool buffer magazine, 9 = Loading magazine)
$TC_MAP2	Identifier of the magazine
$TC_MAP3	Status of magazine - Bit 0: Active magazine - Bit 4: Magazine or tool may be moved
$TC_MAP6	Number of tiers (box magazines only)
$TC_MAP7	Number of columns
$TC_MAP10	Current tool search strategies of magazine (see $TC_MAMP2)
$TC_MPP1	Location type (1= Magazine location, 2 = Spindle/toolholder 3 = Gripper, 7 = Loading point)
$TC_MPP2	Location type (0: Every tool fits in this location)
$TC_MPP3	Consider adjacent location (1=on/0=off)
$TC_MPP4	Location state (Bit 1: Free to accept a tool)
$TC_MPP5	Location type index
$TC_MLSR	Assigning magazine locations of the buffer magazine to the spindle
$TC_MDP1	magazine and location of 1st internal magazine (loading magazine, 9999)
$TC_MDP2	magazine and location of 2nd internal magazine buffer magazine, 9998)

Result: Tool magazine and tool view in SINUMERIK Operate



SITRAIN © Siemens AG 2017

Notes

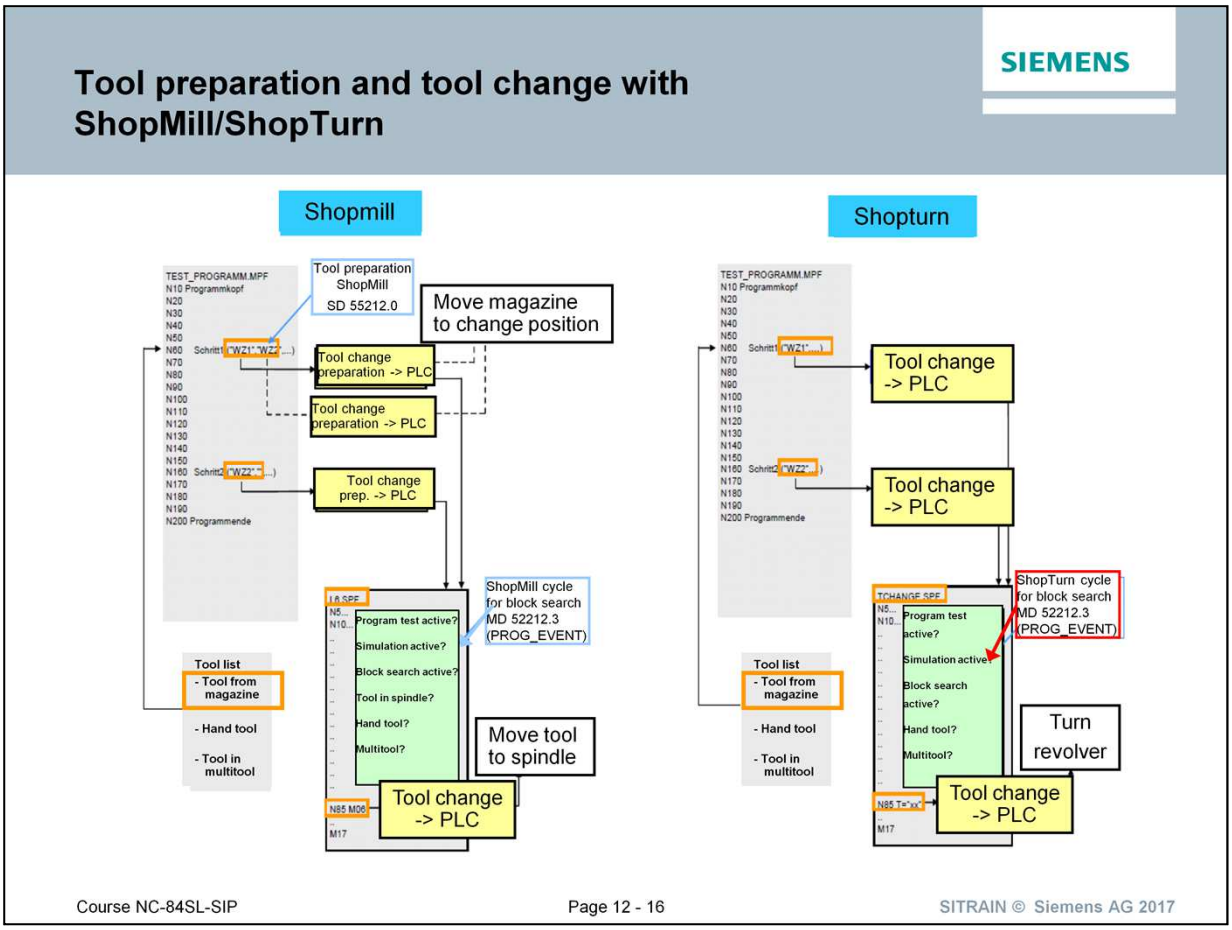


Milling machine

The tool change for milling machines occurs in the following sequence with the "programGUIDE" programming method:
In the NC part program, the preparation of the tool change occurs when the next tool "T="tool name" is called up. This information is used by the PLC to transport the requested tool from storage to the change location. During this time, the workpiece machining can continue using the current tool. The change command M06 is executed in the part program. This command calls up the tool change cycle L6.spf, which was created by the machine manufacturer. This subprogram determines the reason for the call (e.g. tool change or block search, etc.) and moves the spindle to the change location, if applicable. Later in the process, the control routine of the PLC is called up (M206), which then actually changes the tool in the spindle and the next tool. In addition to clamping and unclamping the tools and moving a gripping mechanism, the PLC also handles the reporting back of the current tool position and the change status to the tool management.

Turning machine

In principle, the tool change sequence for a turning machine is identical to that of a milling machine. However, there is generally a tool turret and no preparation of the tool change is performed because there is no transport task. The change cycle is therefore executed directly when the tool is called up in the part program with the "T" command. The task of the PLC is limited to moving the turret and reporting back the position. It is also possible to move the turret with the L6 change cycle.



ShopMill

Unlike "programGUIDE" programming, the cycle-based job shop interfaces have the characteristic that the tool used for a machining process is specified as a parameter in the cycle call of this machining cycle. To extract the tool call for preparing the change from the cycle, bit 0 of the channel setting data $SCS_FUNCTION_MASK_TECH_SET is set to "1". This allows the preparation information to be forwarded to the PLC. For the block search under ShopMill / ShopTurn, bit 3 is also set in MD 52212 $MCS_FUNKTION_MASK_TECH.

ShopTurn

Since turning usually involves working with a turret magazine and there is no tool preparation, the call for a change in ShopTurn is the same as the command for a change. Extracting the change command from the cycle is not required.

Machine data for tool management: Tool change; reset behavior

SIEMENS

- Tool change with preparation for change

; M function replacement M6 -> L6

MD 10715	$MN_M_NO_FCT_CYCLE[0]=6	With M6, calls the tool change cycle L6
MD 10716	$MN_M_NO_FCT_CYCLE_NAME[0]="L6"	
; M206 in L6 for the actual change		
MD 22550	$MC_TOOL_CHANGE_MODE = 1	With "1", prepare tool change; with "0" direct change
MD 22560	$MC_TOOL_CHANGE_M_CODE = 206	M function for tool change

- Reset behavior, block search, etc.

MD 20128	$MC_COLLECT_TOOL_CHANGE = 0	No tool change commands for block search
MD 20130	$MC_CUTTING_EDGE_RESET_VALUE = 1	Tool and cutting edge number remain active (TSM)
MD 20320	$MC_TOOL_TIME_MONITOR_MASK Bit 0 = 1	Time monitoring of the tool in toolholder 1
MD 52274	$MCS_TM_LOAD_STATION=2	Set number of loading station = 2

Course NC-84SL-SIP Page 12 - 17 SITRAIN © Siemens AG 2017

Tool change

MD 22550 $MC_TOOL_CHANGE_MODE = 1 Call tool change with M command for milling machines. For turning machines (turret head), the tool change takes place when the tool is called, e.g. with T="parting tool".

MD 22560 MC_TOOL_CHANGE_M_CODE = 206 is the M function for the tool change call from the PLC within a manufacturer cycle, e.g. L6.SPF .

MD 10715 $$MN_M_NO_FCT_CYCLE = 6 is the M function, which is replaced by the cycle call (name from MD 10716).

MD 10716 $MN_M_NO_FCT_EOP = L6 specifies the name of the manufacturer cycle called, e.g. L6.SPF. This is located in (is stored in) the directory for manufacturer cycles.

Spindle behavior and chuck (turning machine)

MD 10714 $MN_M_NO_FCT_EOP = 32. In the T, S, M screen form, S stands for spindle. When a spindle number is entered and started with NC Start, the spindle runs in jog mode. The "Temp" program called is terminated with M32; in other words, the spindle is active following reset if a value of 2 has been entered in MD 35040 $MA_SPIND_ACTIVE_AFTER_RESET.

MD 52250 $MCS_M_CODE_CHUCK_OPEN[n]. Index 0 is valid for the main spindle, index 1 is valid for the counterspindle. The chuck is opened on the stationary spindle.

MD 52251 $MCS_M_CODE_CHUCK_OPEN_ROT[n]. As MD52250, but on the rotating spindle.

MD 52252 $MCS_M_CODE_CHUCK_CLOSE[n] . The corresponding chucks are closed here.

Other machine data for technological M functions

Spindle behavior and chuck (turning machine)

▪ MD 52250	$MCS_M_CODE_CHUCK_OPEN[n]	Index 0= Main spindle Index 1 counterspindle, stationary spindle
▪ MD 52251	$MCS_M_CODE_CHUCK_OPEN_ROT[n]	Index 0= Main spindle Index 1 counterspindle, rotating spindle
▪ MD 52252	$MCS_M_CODE_CHUCK_CLOSE[n]	Index 0= Main spindle Index 1 counterspindle

Coolant

▪ MD 52229	$MCS_ENABLE_QUICK_M_CODES	Enable fast M functions for coolant
▪ MD 52230	$MCS_M_CODE_ALL_COOLANTS _OFF = 9	M code M9 for all coolants OFF
▪ MD 52231	$MCS_M_CODE_COOLANT_1_ON = 8	M code M8 for coolant 1 ON
▪ MD 52232	$MCS_M_CODE_COOLANT_2_ON = 7	M code M7 for coolant 2 ON
▪ MD 52233	$MCS_M_CODE_COOLANT_1_AND_2 _ON = -1	No M code coolant 1 and 2 ON

SITRAIN © Siemens AG 2017

Notes

Tool change cycle L6.SPF

```
PROC L6 SAVE DISPLOF                    ; Change cycle with saving of the preparatory functions on call
;VERSION: 01.00.14. Apr 23, 2010
;CHANGE : 01.00.08. Oct 27, 2009
DEF INT _WZ_IN_SP,_WZ_VOR               ; Variable definition
;
IF(NOT $P_SEARCH)                                ; If no block search
  _WZ_IN_SP=$TC_MPP6[9998,1]            ; Read tool number from tool in spindle
  GETSELT(_WZ_VOR)                              ; Read tool number from tool preselection
  IF(_WZ_IN_SP<>_WZ_VOR)                ; If tool in spindle is not same as new tool
    ; Spindle positioning blocks, e.g. SPOSA[1]= 0          ; Position spindle
    G0                                  ; Move spindle to change location
    ;                                   ;
    G75 Z=0
    WAITS(1)                            ; wait for spindle
  ENDIF
ELSE
ENDIF
;
M206                                    ; Information/call of PLC – start motion sequence of the changing mechanism, e.g. gripper
;
;SETPIECE(1)                            ; If the number of workpieces finished is monitored with the tool
                                        ; (column "Tool Wear", "Column TC = C") counter of the tool adapter
                                        ; Decrement number by "1"
M17
```

Course NC-84SL-SIP Page 12 - 19 SITRAIN © Siemens AG 2017

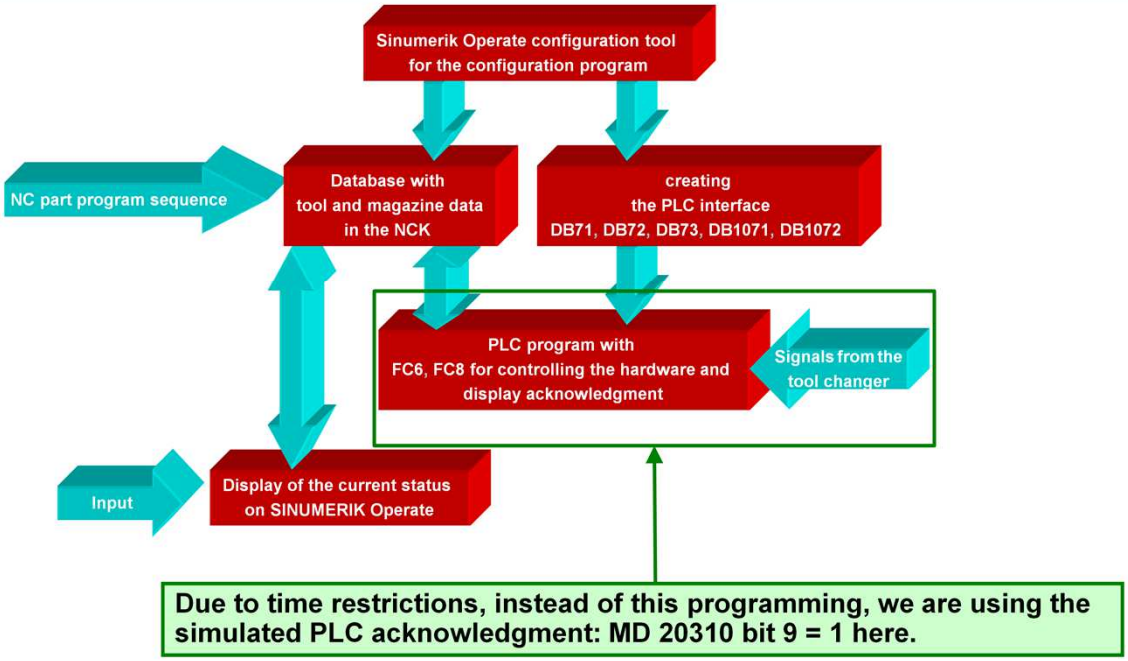
The figure above shows an example of a change cycle. This is only a basic outline and is far from complete, but it is executable.
The function shown compares whether or not the tool in the spindle is the same as the new tool. If the answer is yes, the spindle is moved to the change location (by whatever means) and the spindle is oriented. Then the PLC is called and the mechanical change operation is performed. After acknowledgment of the change sequence, the cycle is continued and ultimately changes back to the part program.

Note

To optimize the change sequence, it is also possible to control motion tasks, e.g. of the mechanical system, by means of synchronized actions. However, in this case, the system load will be higher during the tool change because the interpolator is running. Please consider this when optimizing the machine.

Note for PLC programming of the individual basic functions

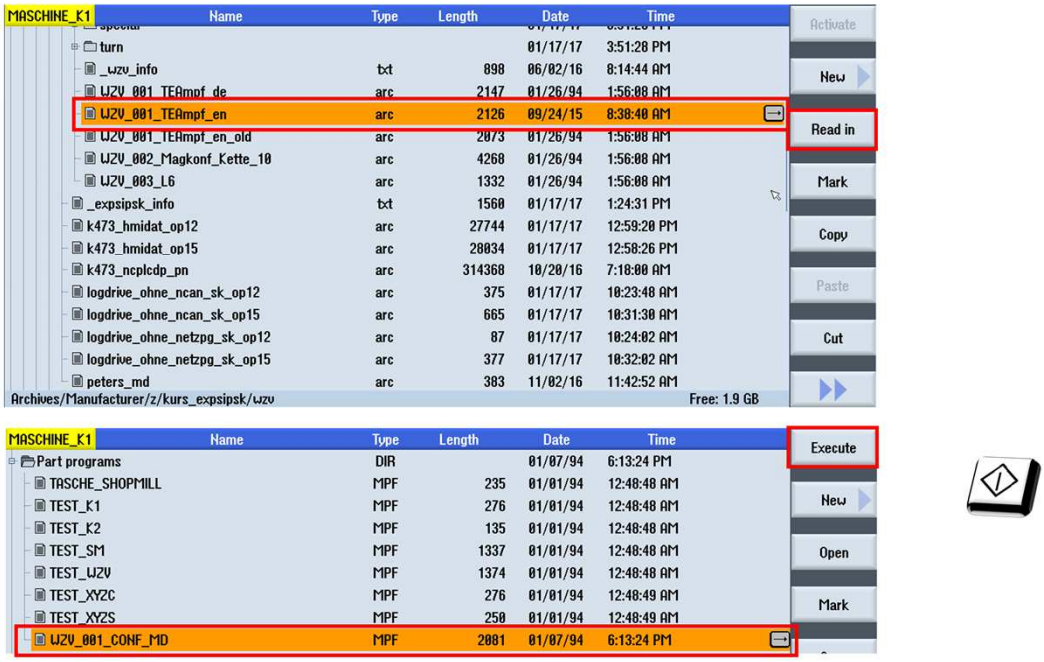
SIEMENS



 SITRAIN © Siemens AG 2017

Notes

Exercise 1: Activate machine data for tool management



 SITRAIN © Siemens AG 2017

Objective

To activate machine data for tool management.

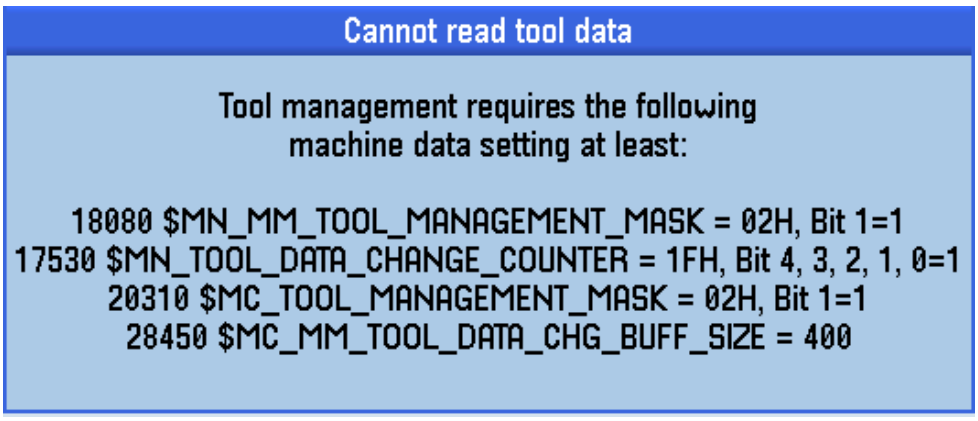
To save time, we will just read in the machine data using a part program in this case.

Steps

1. Check OA parameters
2. Read in archive "WZV_001_TEAmpf_de.arc"
   - Directory: "Archive/Hersteller/z/kurs_expsipsk/wzv /"
3. Select and start the "WZV_001_CONF_MD.MPF" part program
4. Reset (po)
5. Test

Sequence

1. Check OA parameters The following error message should be displayed:



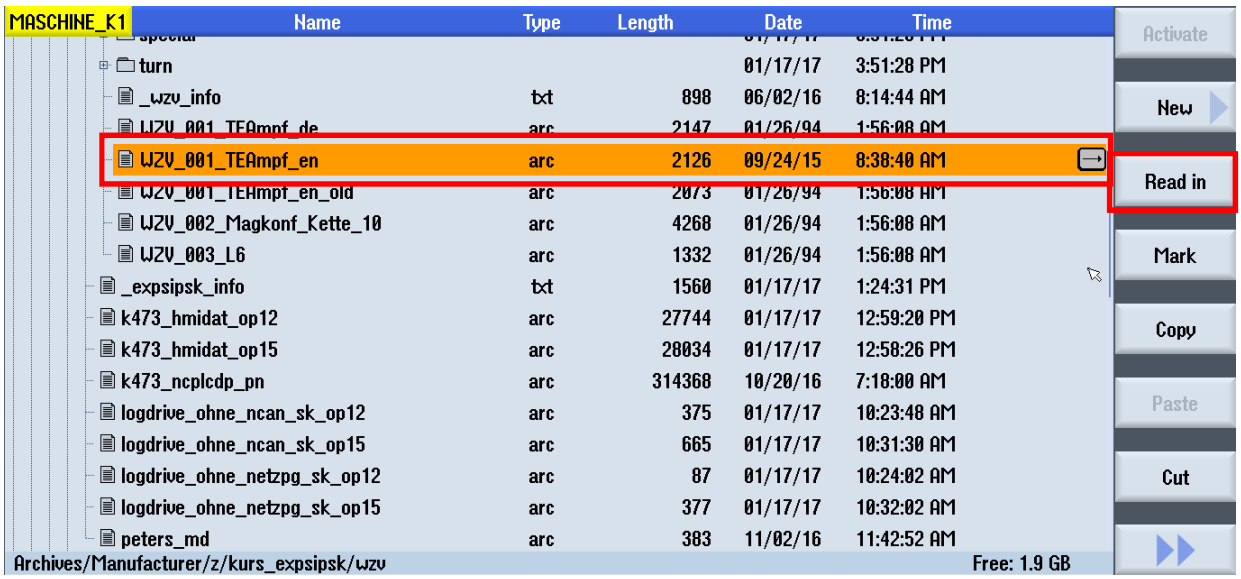
2. Read in archive "WZV_001_TEAmpf_en.arc"

Read in the following archive:

"WZV_001_TEAmpf_en.arc"

You will find it in the following directory:

"Archive / Hersteller / z / kurs_expsipsk / wzv /"

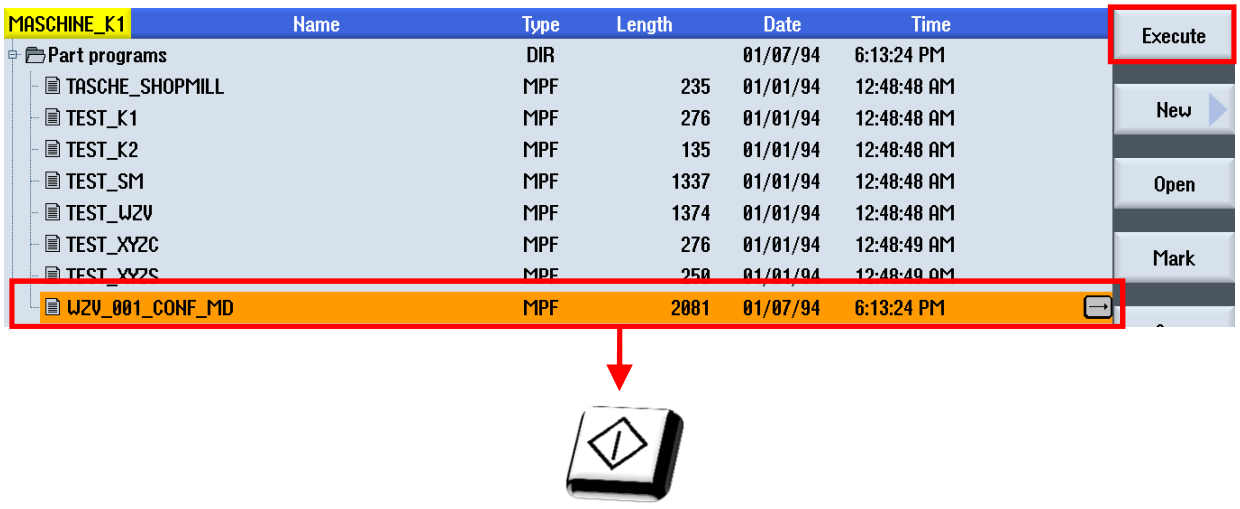


3. Select and start the "WZV_001_CONF_MD.MPF" part program

Select the following part program for execution:

"WZV_001_CONF_MD.MPF"

and start it in AUTOMATIC operating mode
(you can do this by setting MD 20700=0: NC start without reference point)



Note

In older software versions V< 4.7, the NC data must be backed up and read in again at this point (alarm 4400). These steps are no longer necessary in SW 4.7! Instead:

4. Reset (po)

With

> Softkey: Reset (po)

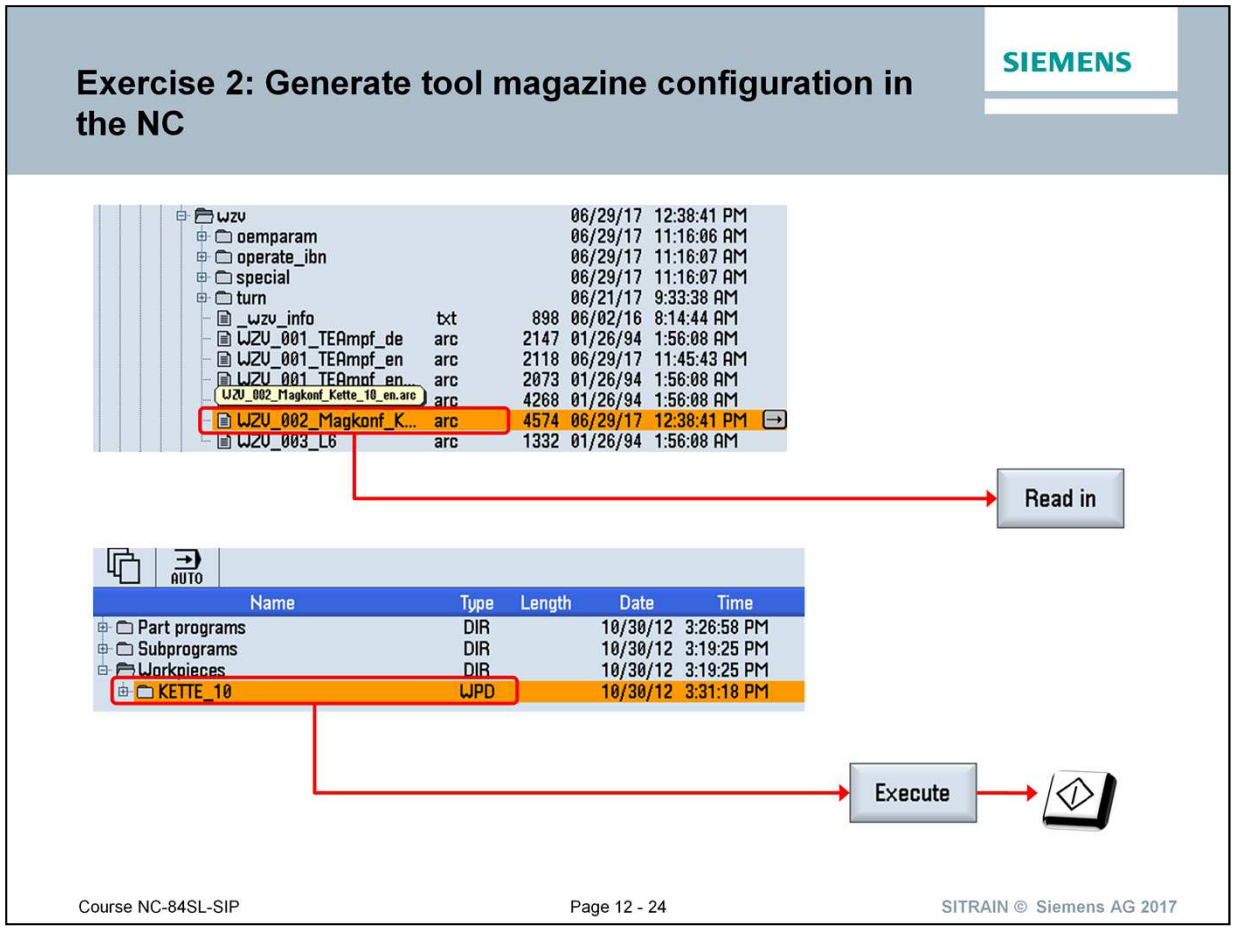
reconfigure the memory

5. Test

Parameters OA:

The message "Initialization error in magazine data" is displayed:





Objective

Generate tool magazine configuration in the NC.

To save time, we will just set up the machine configuration using a part program in this case.

Steps

1. Read in archive "WZV_002_Magkonf_Kette_10.arc"
- Directory: "Archive / Hersteller / z / kurs_expsipsk / wzv /"
2. Select and start workpiece "KETTE_10.WPD"
3. Test: OA: Parameter: Tool magazine is displayed
4. Test 2: OA: Parameter: A tool can be created and loaded

Note

The magazine configuration you have generated has the following characteristics:

- Chain magazine with 10 locations
- 1 spindle with dual gripper
  -> 3 buffer locations
- 2 load locations:
  No. 1: Manual load location
  No. 2 Standard load location

Sequence:

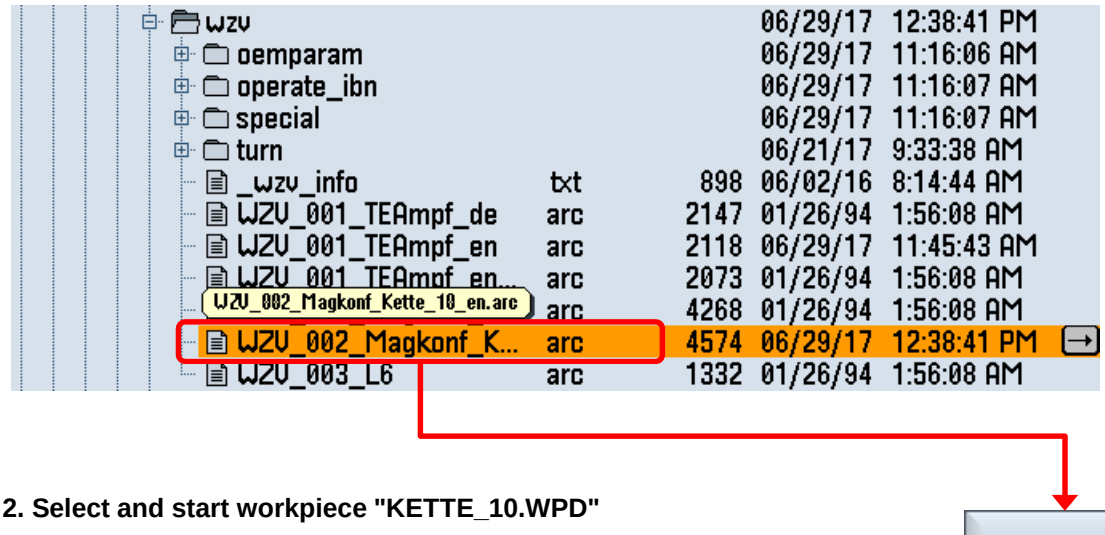
1. Read in archive "WZV_002_Magkonf_Kette_10_en.arc"

Read in the following archive:

"WZV_002_Magkonf_Kette_10_en.arc"

You will find it in the following directory:

"Archive/Hersteller / z / kurs_expsipsk / wzv /"

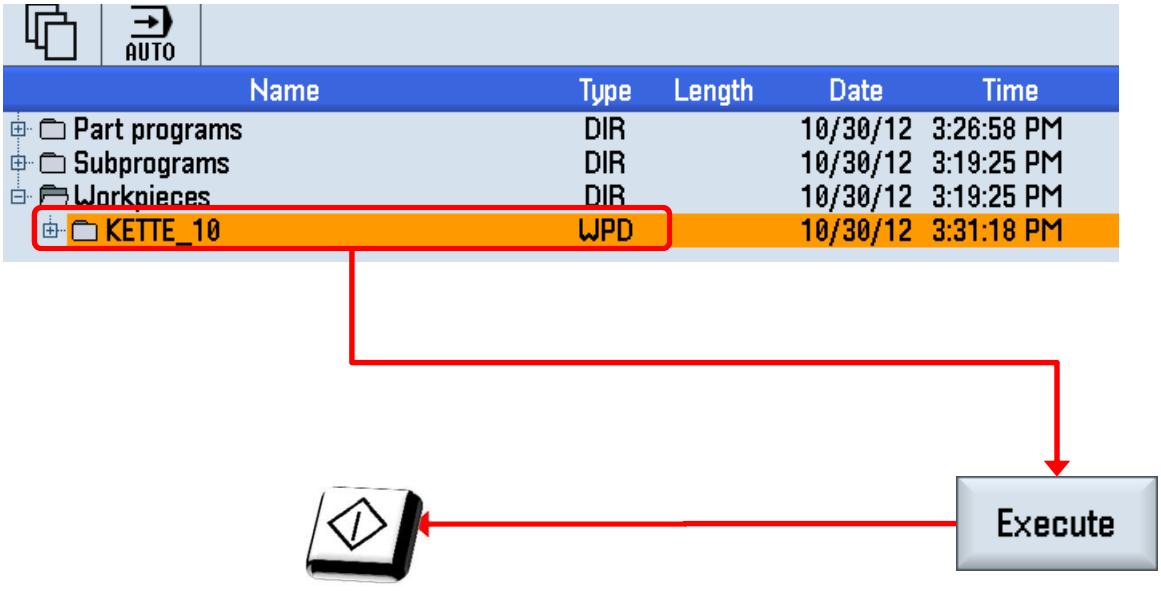


2. Select and start workpiece "KETTE_10.WPD"

Select the following workpiece for execution:

"KETTE_10.WPD"

and start it in AUTOMATIC operating mode



3. Test: OA: Parameter: Tool magazine is displayed

Check under

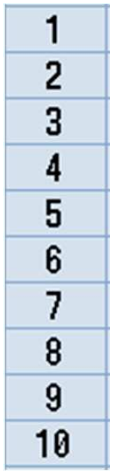
- **OA: Parameter:**

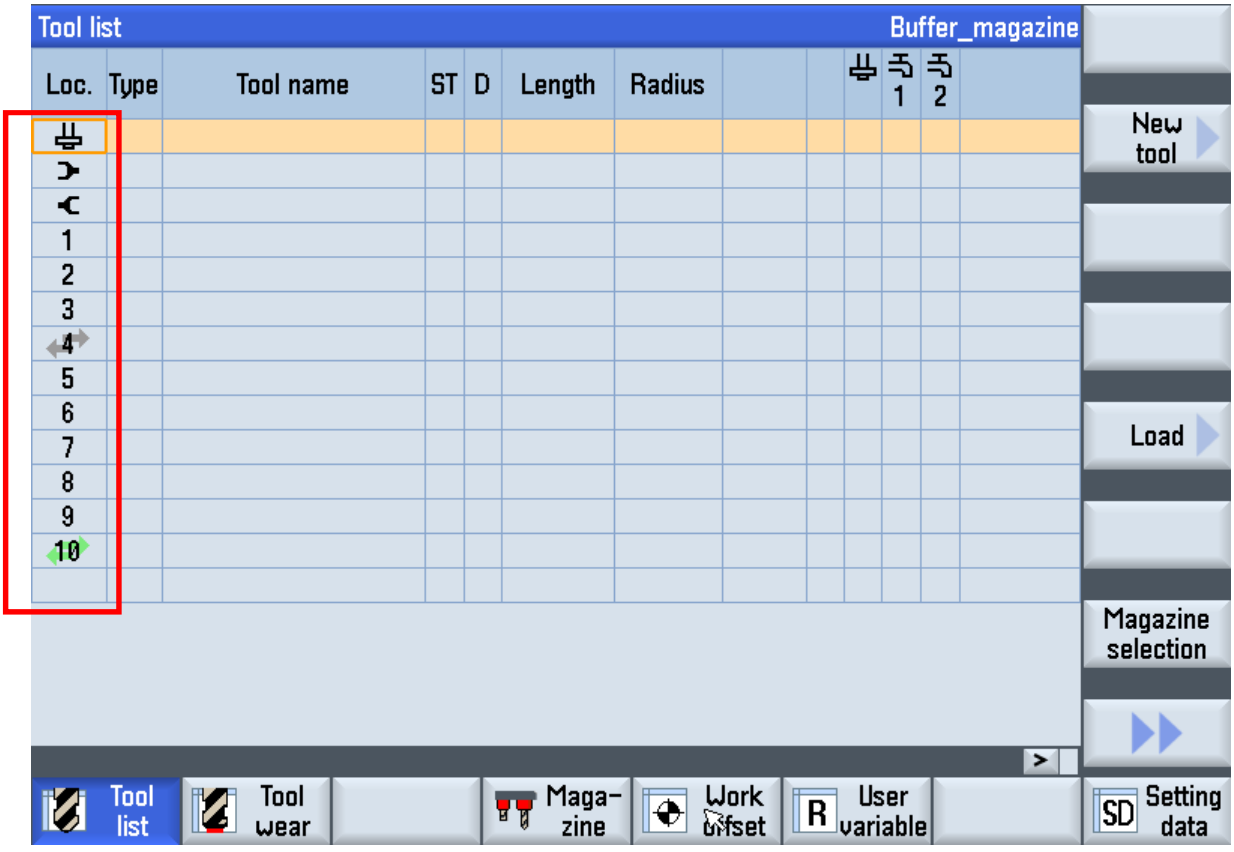
You should be ableto see

- the spindle
- the two gripper locations, and



- the magazine locations:





4. Test 2: OA: Parameter: A tool can be created and loaded

Check under:

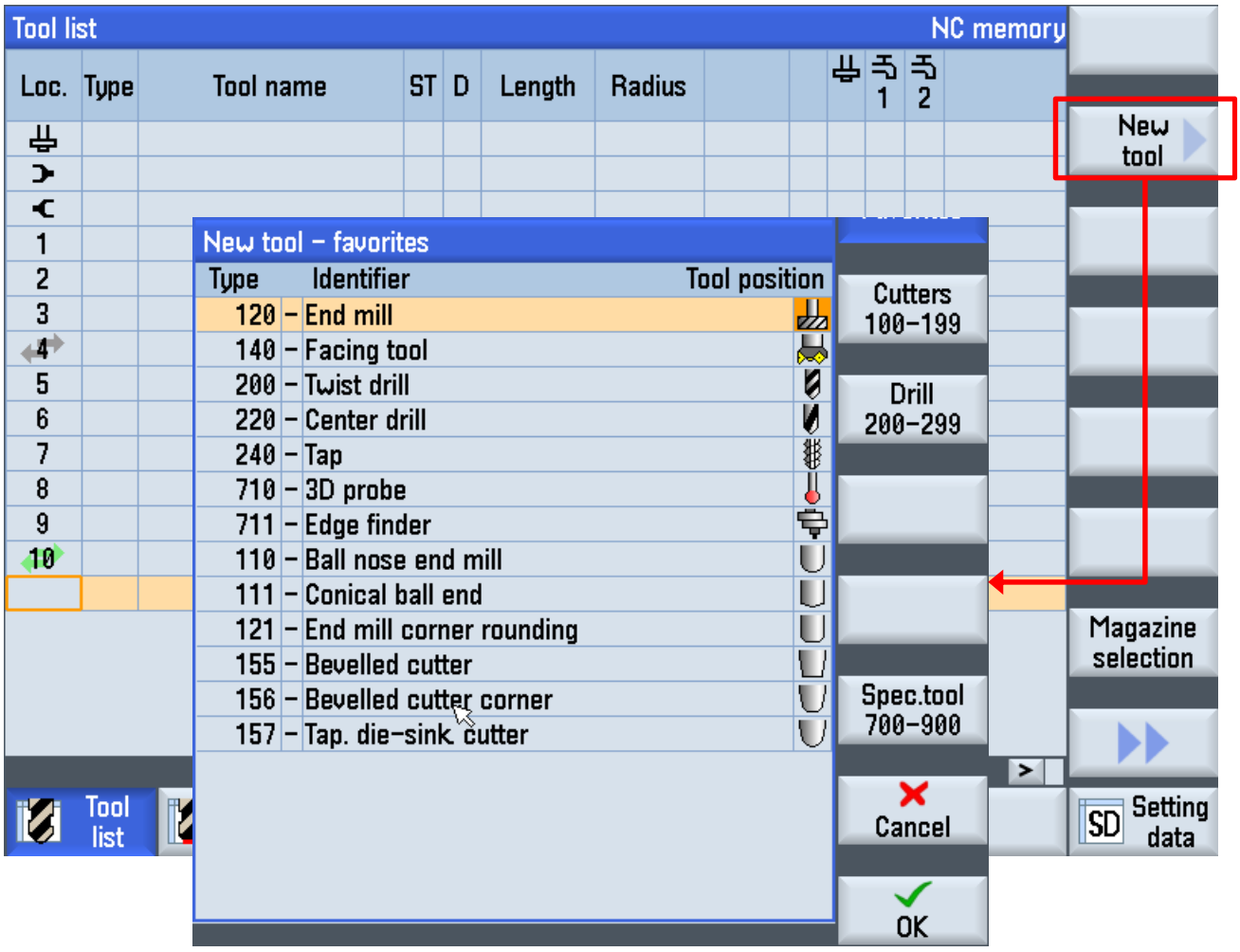
- **OA: Parameter**

A tool can be created and loaded:

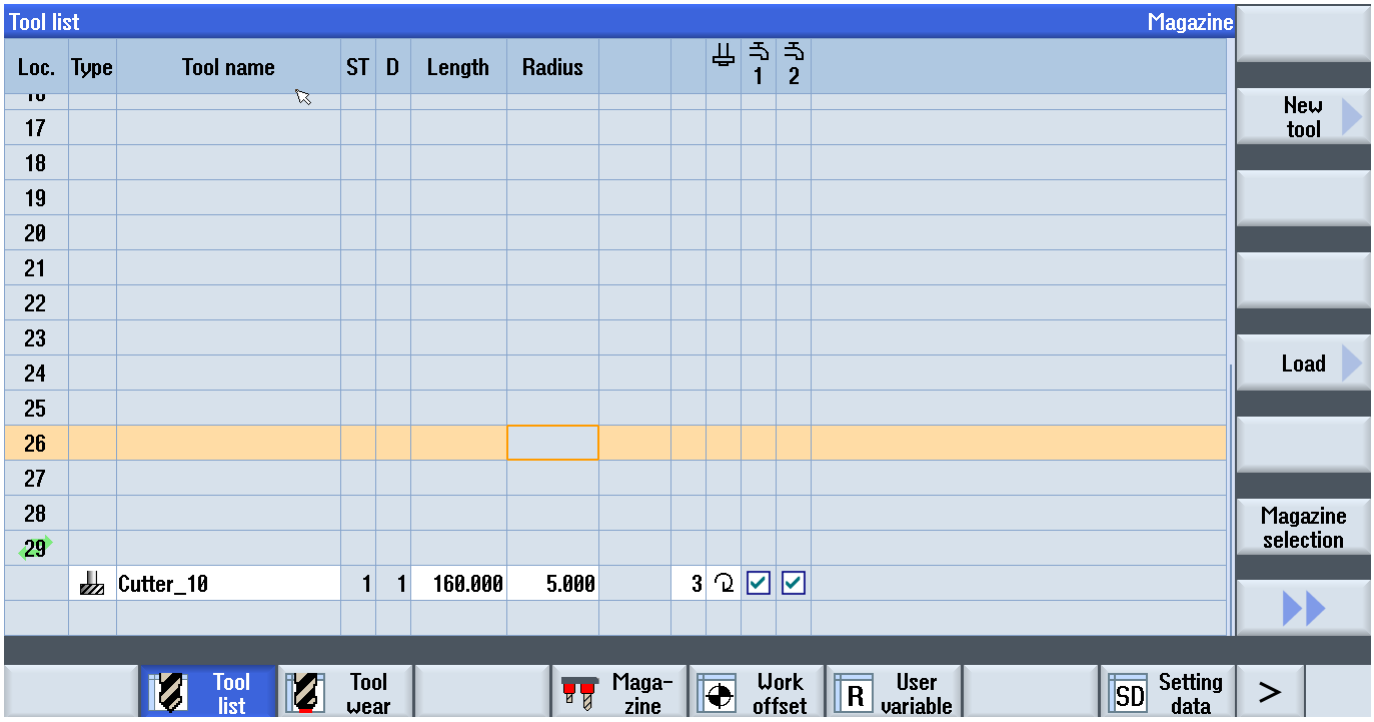
Softkey: New tool

Select, e.g., End mill,

Name: EndmillL_10



Softkey: 'Load'

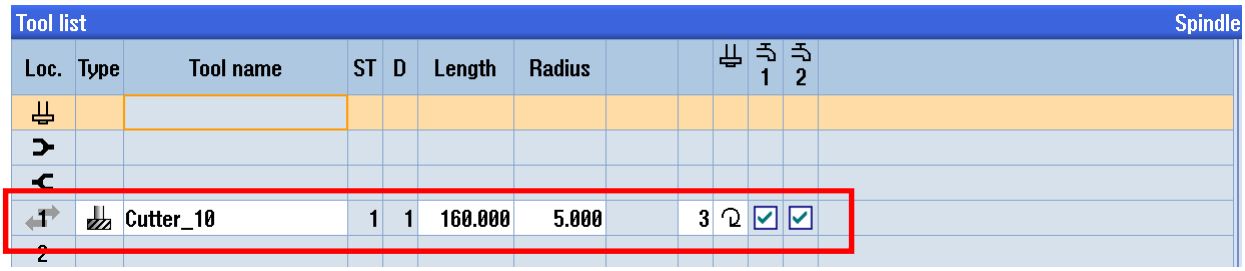




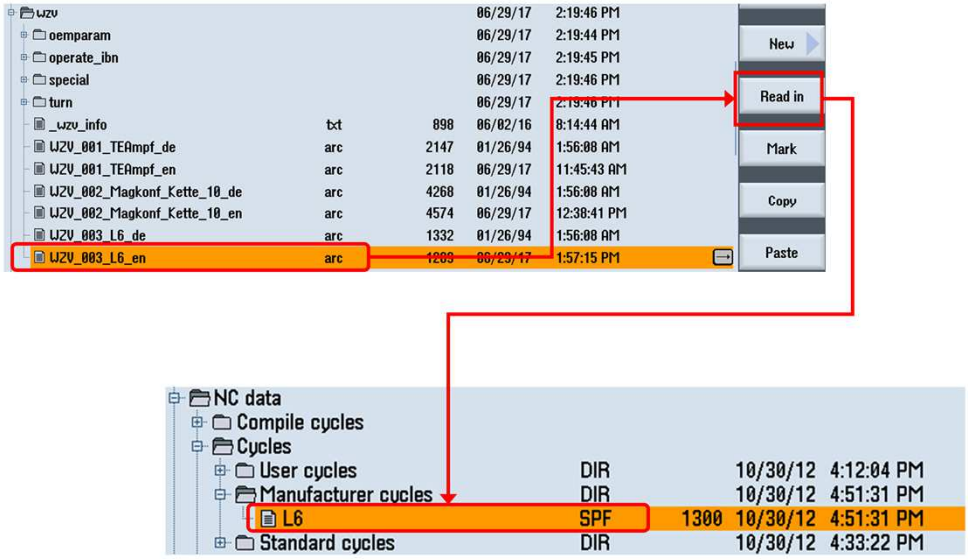
Select e.g. location 1:



Result:



Exercise 3: Activate tool change cycle L6 and test with T, S, M



Course NC-84SL-SIP Page 12 - 29 SITRAIN © Siemens AG 2017

Objective

Activate tool-change cycle L6.
(corresponding machine data has already been set in the first exercise file).

To save time, we will just activate
L6 using the archive here.

Steps

1. Read in archive "WZV_004_L6.arc"
   - Directory: "Archive / Hersteller / z / kurs_expsipsk / wzv /"
2. Check: L6 in "Manufacturer cycles" directory
3. Test: Tool change with T,S,M

Sequence:

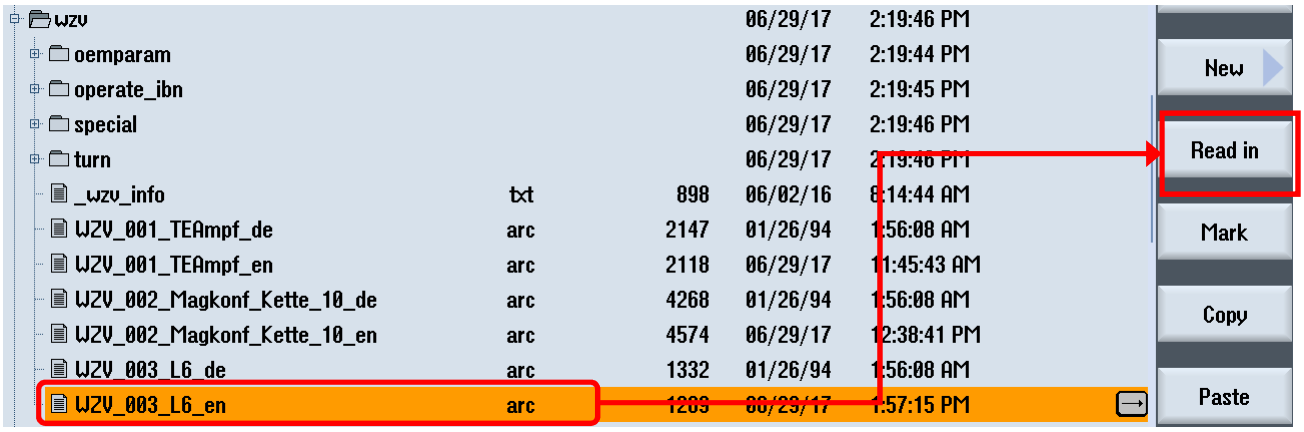
1. Read in archive "WZV_004_L6.arc"

Read in the following archive:

"WZV_004_L6.arc"

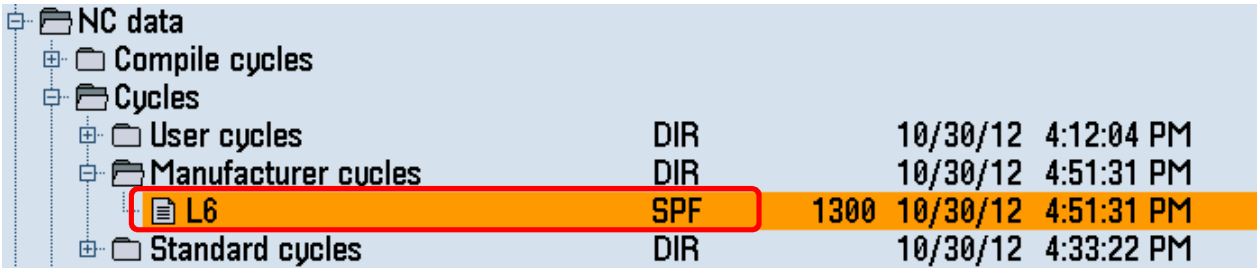
You will find it in the following directory

"Archive / Hersteller / z / kurs_expsipsk / wzv /"



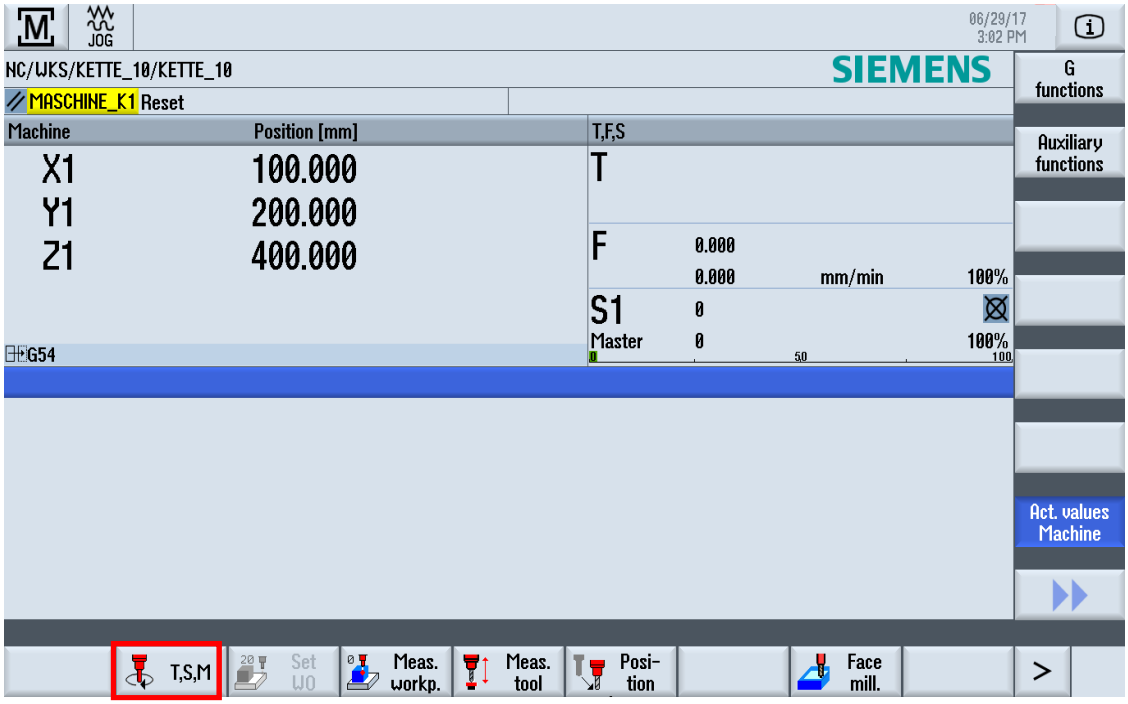
2. Check: L6 in "Manufacturer cycles" directory

Check that the "Manufacturer cycles" directory now contains an L6 cycle



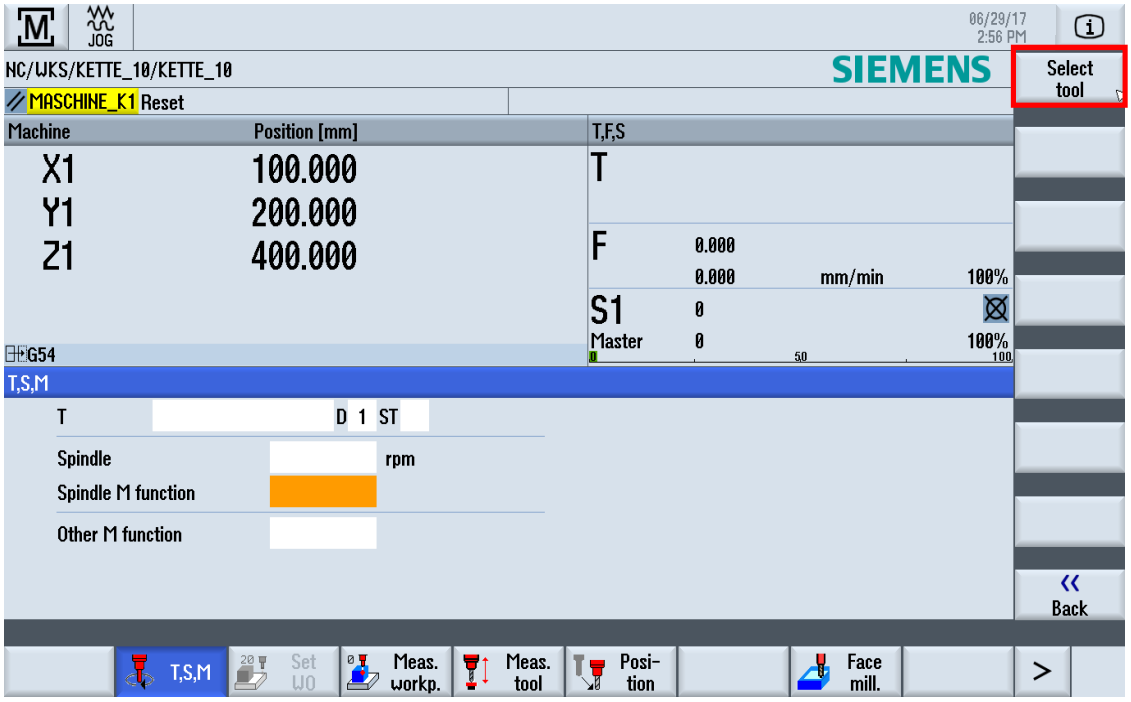
3. Test: Tool change with T,S,M

Check that a tool change with the
"T,S,M" function is possible in **JOG** mode:



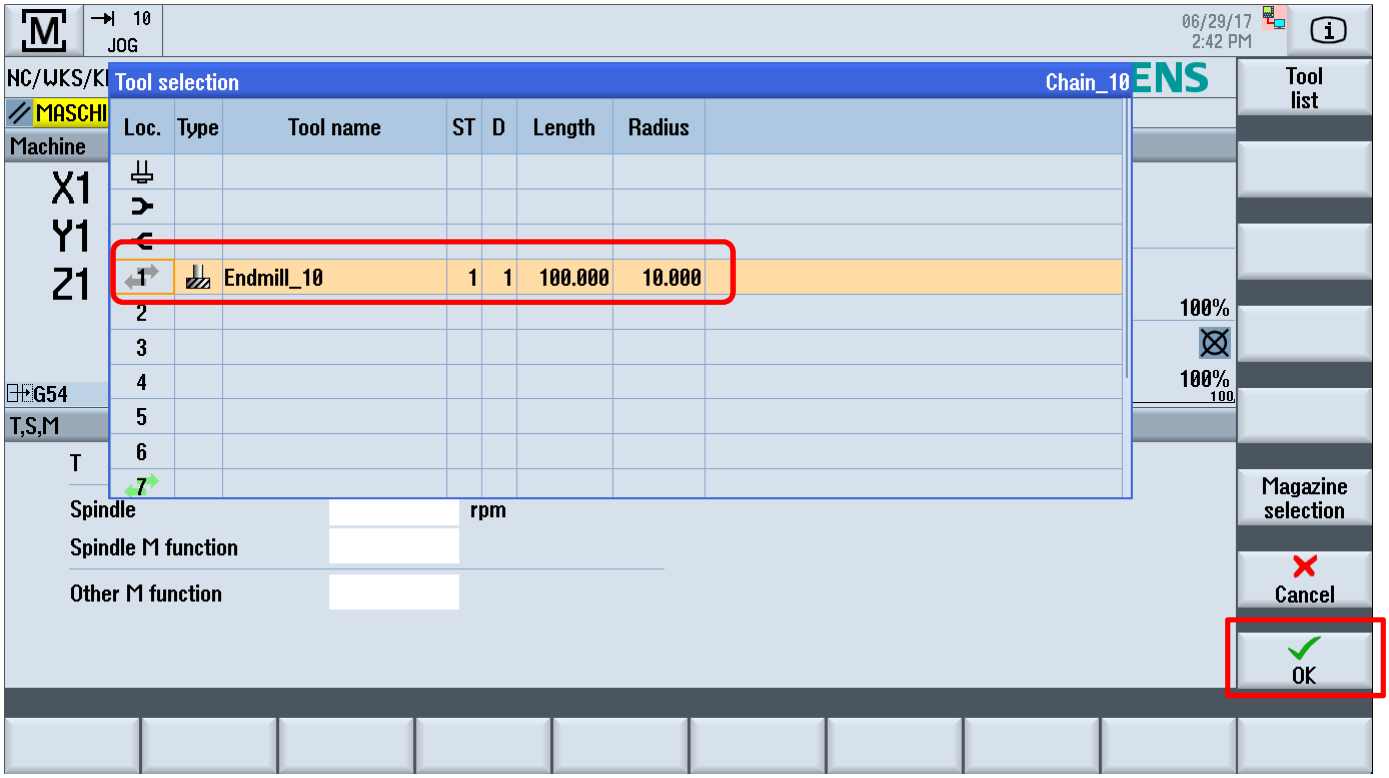
To access the screen form for changing the tool, select:

> Softkey: 'Select tool'



Select the tool created previously with:

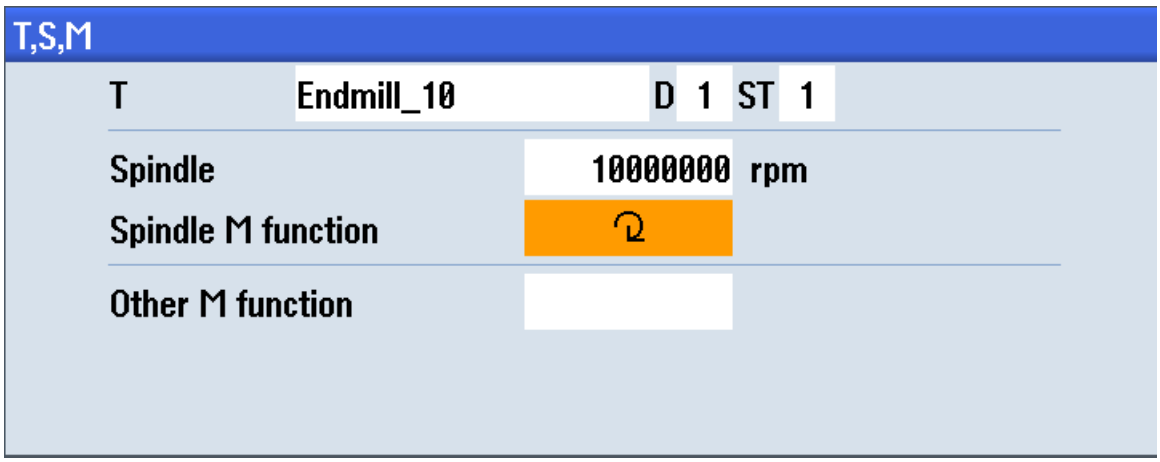
Softkey: ‘OK’



For the spindle, select:

1000 rpm

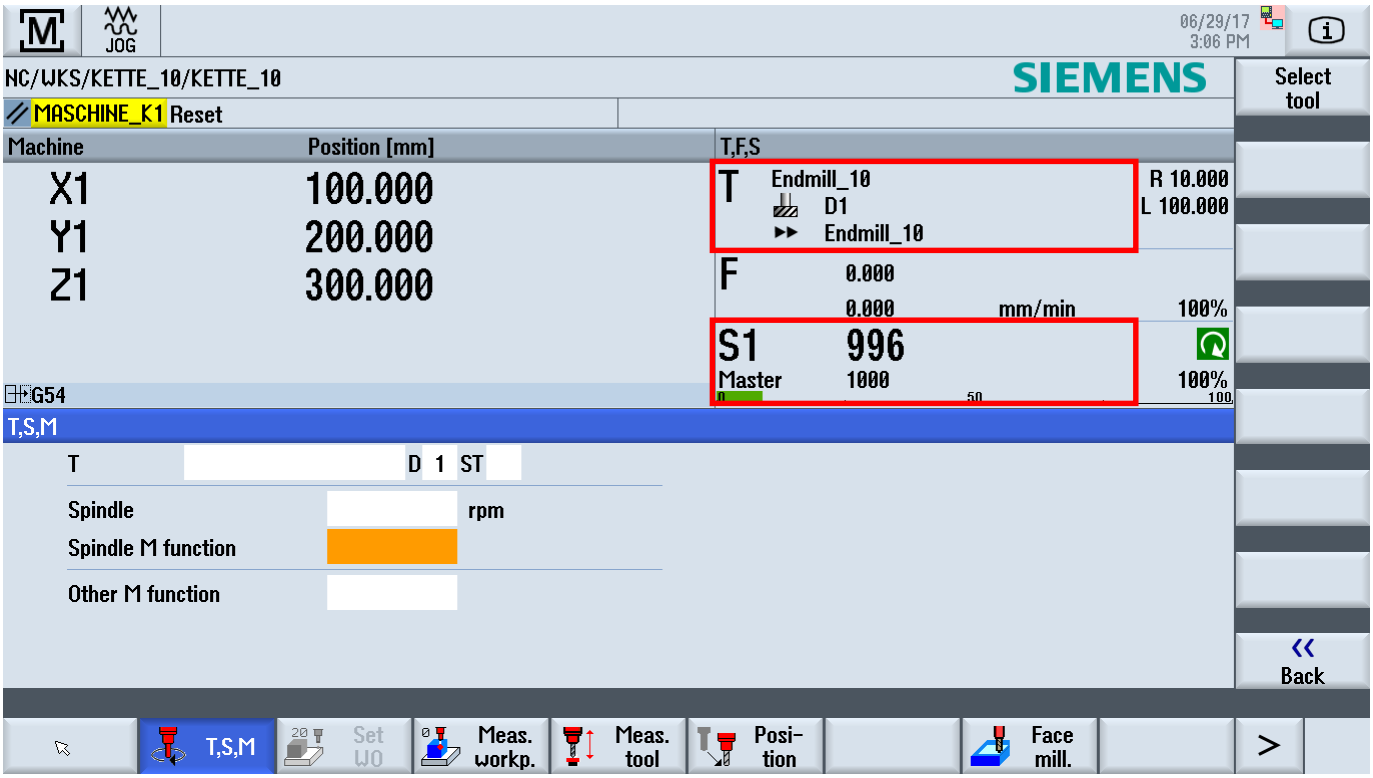
SELECT key -> CW rotation: 



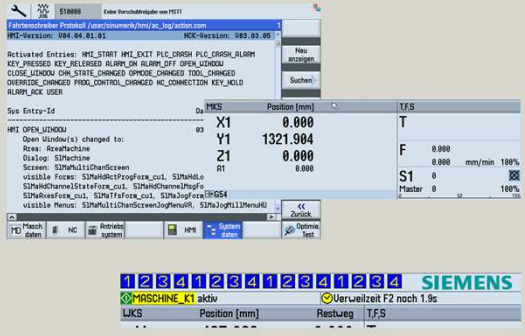
Press NC-START:



-> The tool should be displayed in the "T,F,S" window
and the spindle should turn at 1,000 rpm:



SIEMENS



Chapter 13

Customizing Sinumerik Operate

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Tools for SINUMERIK Operate on PCU 3
Linux service shell: 4
Directory structure of SINUMERIK Operate 5
Customizing softkeys 6
Setting the font size of the actual value display 9
Deactivating the Program Test 10
Activating the “Teach in" function 11
Templates 12
Selected initialization files 13
slfsfileattributes.ini 14
slmagcodeconfig.ini 17
slmahdconfig.ini 18
actlog.ini 21
slpmconfig.ini 23
slsuconfig.ini 24
Own logo in the Machine OA 25
3D simulation (option) 26
Simultaneous recording 27
Machine data for simulation 28
Raw part definition for simulation/simultaneous recording 29
Configuring simulation and simultaneous recording 30
Coordinate system orientation MD 52000 31
EUNA: Electronic logbook 32
Exercises 33

Learning objectives

- You will be familiar with the configuration options for the PCU HMI
- You will be able to make softkeys available based on access rights
- You will be familiar with selected display settings in SINUMERIK Operate
- You will be familiar with a selection of SINUMERIK Operate initialization files



 SITRAIN © Siemens AG 2017

Notes

Tools for SINUMERIK Operate on PCU

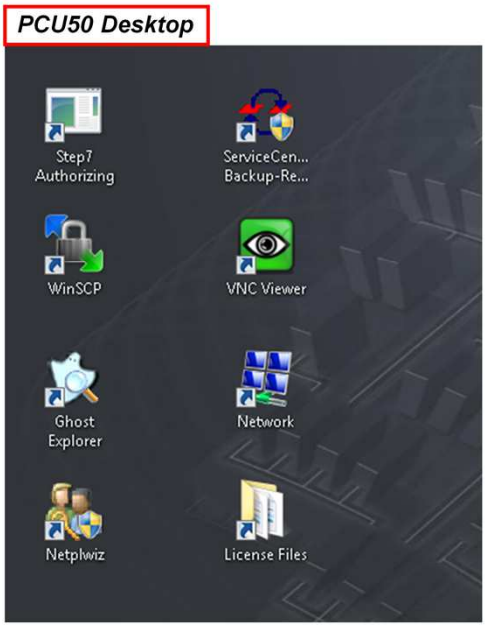
SIEMENS

ServiceCenter Backup-Restore	Configuring / Start of Ghost Backup/Restore
Ghost Explorer	Extract / Change Ghost Backup file

Is treated in the chapter Data Backup PCU50t.

WinSCP	Linux file handling (NCU); Service Shell
VNC Viewer / VNC Scanner	Remote operation PCU/NCU
Netplwiz	Automatic logon

Is treated in the chapter Networking Tools



Course NC-84SL-SIP Page 13 - 3 SITRAIN © Siemens AG 2017

Tools The most important tools on the PCU and their functions are:

ServiceCenter backup/restore

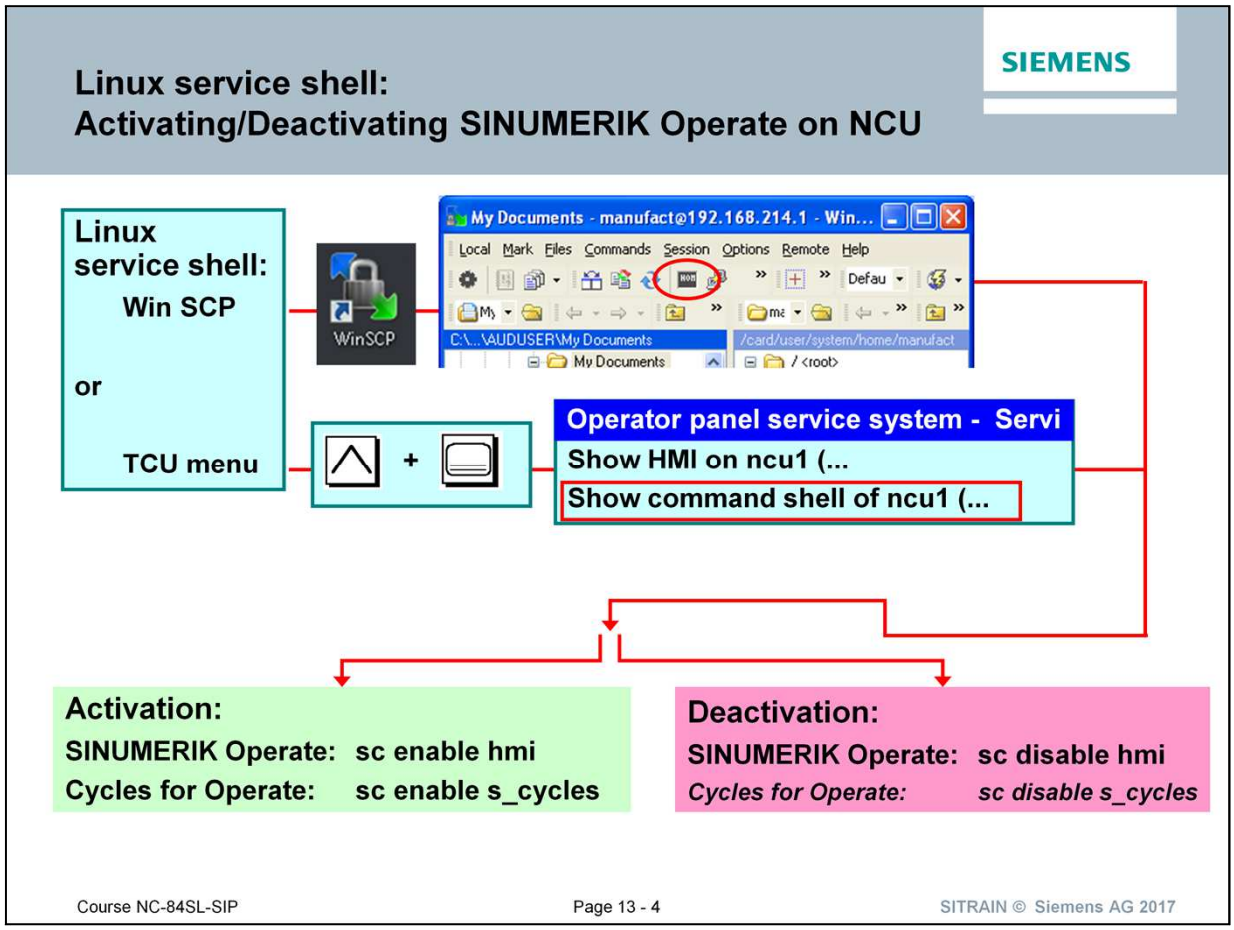
The Ghost backup/restore is configured (network settings) or started here.

Ghost Explorer Used to extract and also edit the individual files from a Ghost backup file.

WinSCP For Linux data handling of the Ext3FS system on the CF card of the NCU; Call the Linux ServiceShell (e.g. service commands sc, etc.).

VNC Viewer Permits remote monitoring/remote control of PCU and NCU.

Netplcwiz Used to configure the automatic logon



SINUMERIK Operate on NCU

SINUMERIK Operate is active by default on the NCU.

Deactivating

This is necessary if a PCU 50 with an active user interface is connected.
To avoid the two user interfaces (PCU and NCU) "clashing", SINUMERIK Operate must be deactivated on the NCU.
This is done by issuing the following command in the Linux service shell:

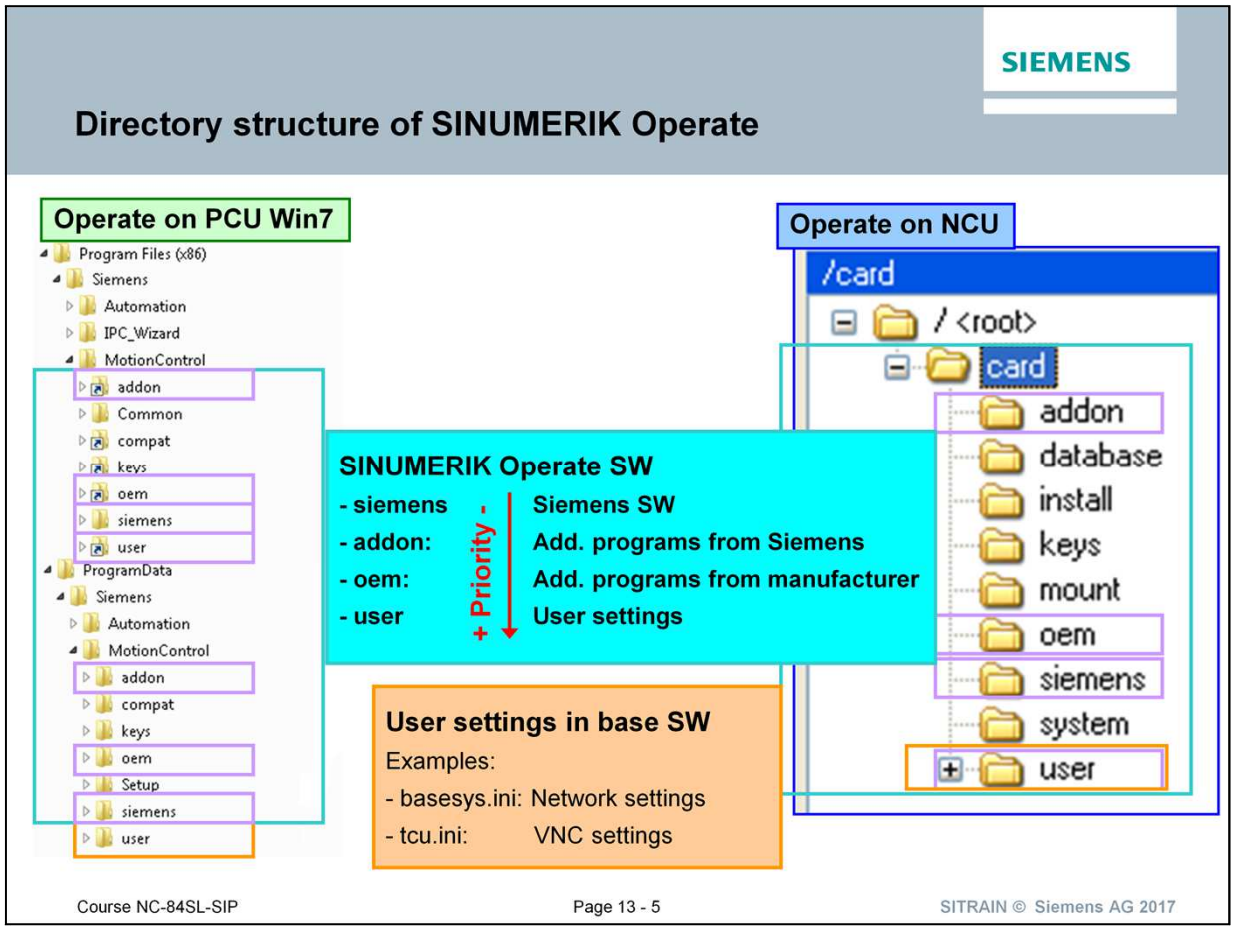
sc disable hmi

Activating

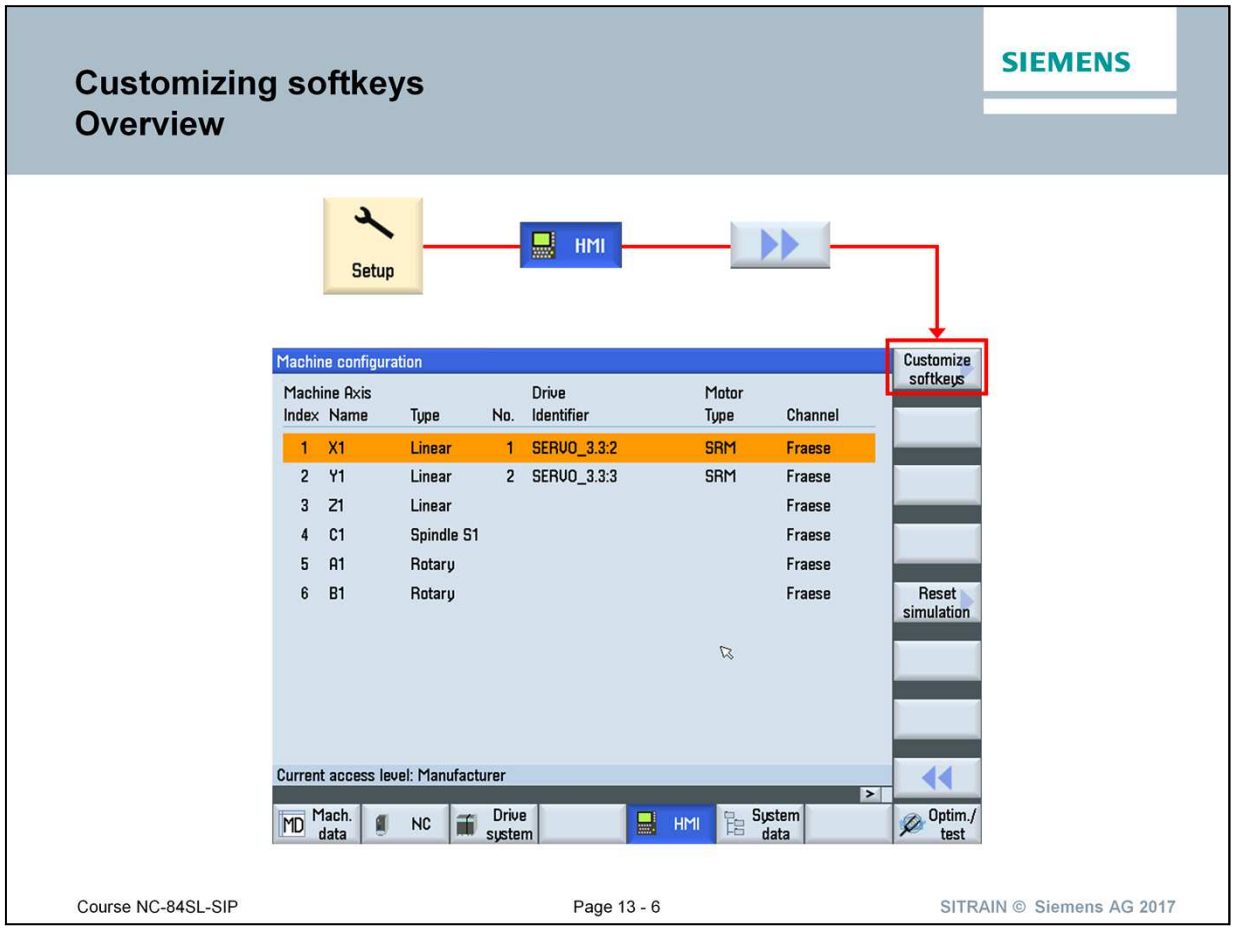
The following commands are required to activate the SINUMERIK Operate user interface on the NCU
including standard cycles:

sc enable hmi

sc enable s_cycles



Directory structure	The add_on, oem, and user directories are not deleted or overwritten when an update is performed. The directories contain the following contents:
siemens	The directory contains the SINUMERIK Operate I Software. In the case of the NCU, it also contains the control software (NCK, SINAMICS, PLC, CP) In this case, the actual HMI software is located in subdirectory: /card/siemens/sinumerik/hmi
add_on	Directory for additional Siemens products.
oem	Directory for OEM applications.
user	User directory in which customer settings different from standard are stored.
Priority	The priority of entries in ini files is: **user > oem > add_on > siemens**
user_base (PCU)	User settings (base software) for Operate on PCU.
or user (NCU)	User settings (base software) for Operate on NCU.
Examples:	• C:\Program Files(x86)\Siemens\MotionControl\user\system\etc\ (Operate on PCU) • /card/user/system/etc/ (Operate on NCU) - **basesys.ini** Network settings - **tcu.ini** Settings for VNC (remote access) • C:\ProgramFiles(x86)\Siemens\MotionControl\ user\common\tcu\TCU1\common\tcu (Operate on PCU) • /card/user/common/tcu/TCU1/common/tcu/ (Operate auf NCU) **- config.ini** Default setting VNC Server



In the extension of the “Setup" > "HMI" operating area, you will find the softkey to access the area for customizing access rights to softkeys.

Access hierarchy

A new access level can only be allocated for those softkeys that are also visible and can be used in the actual access level.

Example:
If a softkey is only accessible with the "Manufacturer" access level, the access level cannot be changed by an operator with access level "Key-operated switch setting 3".

Note

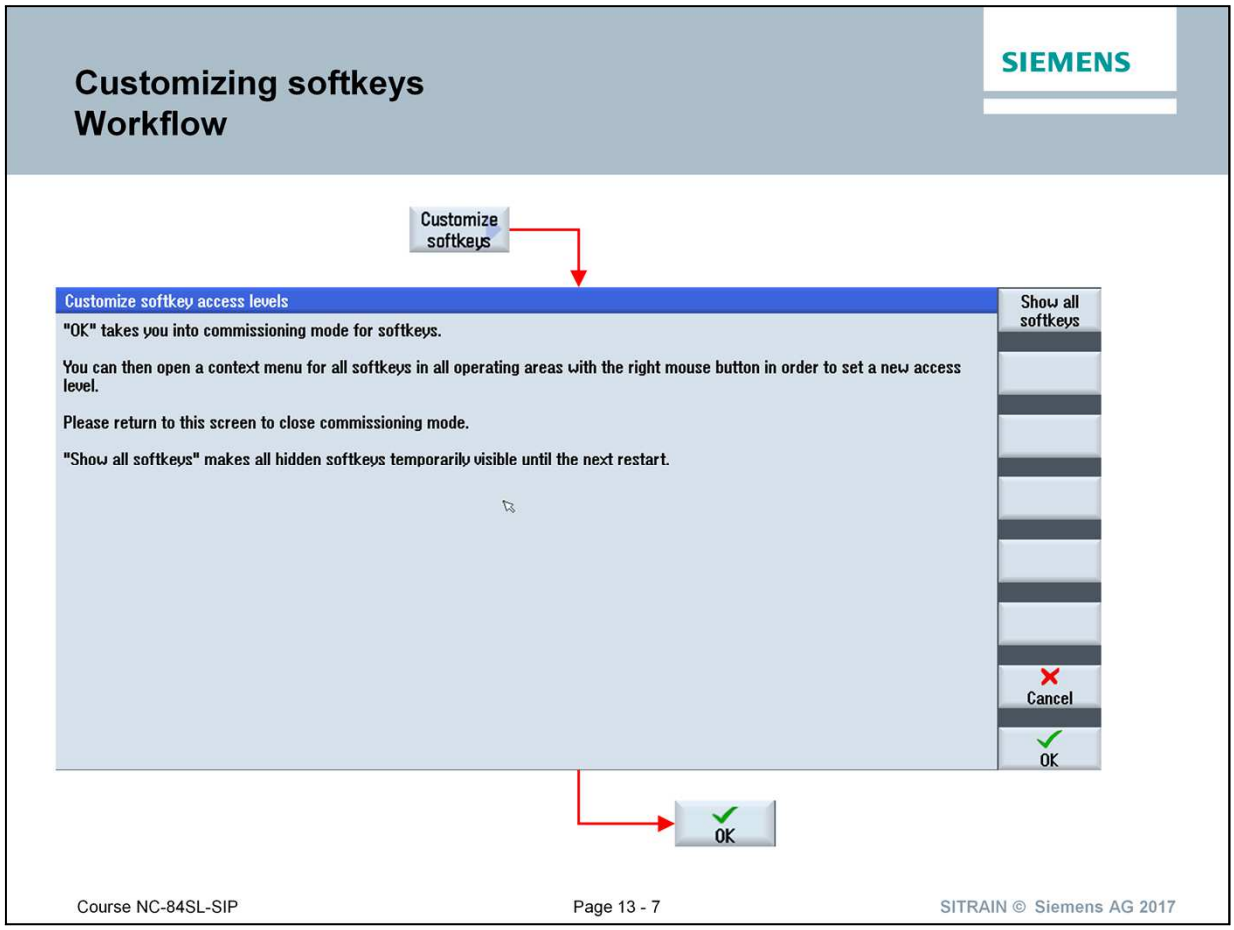
To ensure that the system remains stable and operational, access rights to certain softkeys cannot be customized.

The following softkeys cannot be hidden and the access level for them cannot be changed:

- All "OK" softkeys
- All "Cancel" softkeys
- All "Accept" softkeys
- All "<<" (previous) and ">>" (next) softkeys

The access level for softkeys in the cycle dialog boxes can also be changed; this does not apply to softkeys in ‘Run My Screens’ dialog boxes.

Changes to softkeys remain effective even after a software upgrade possibly involving softkeys being moved.



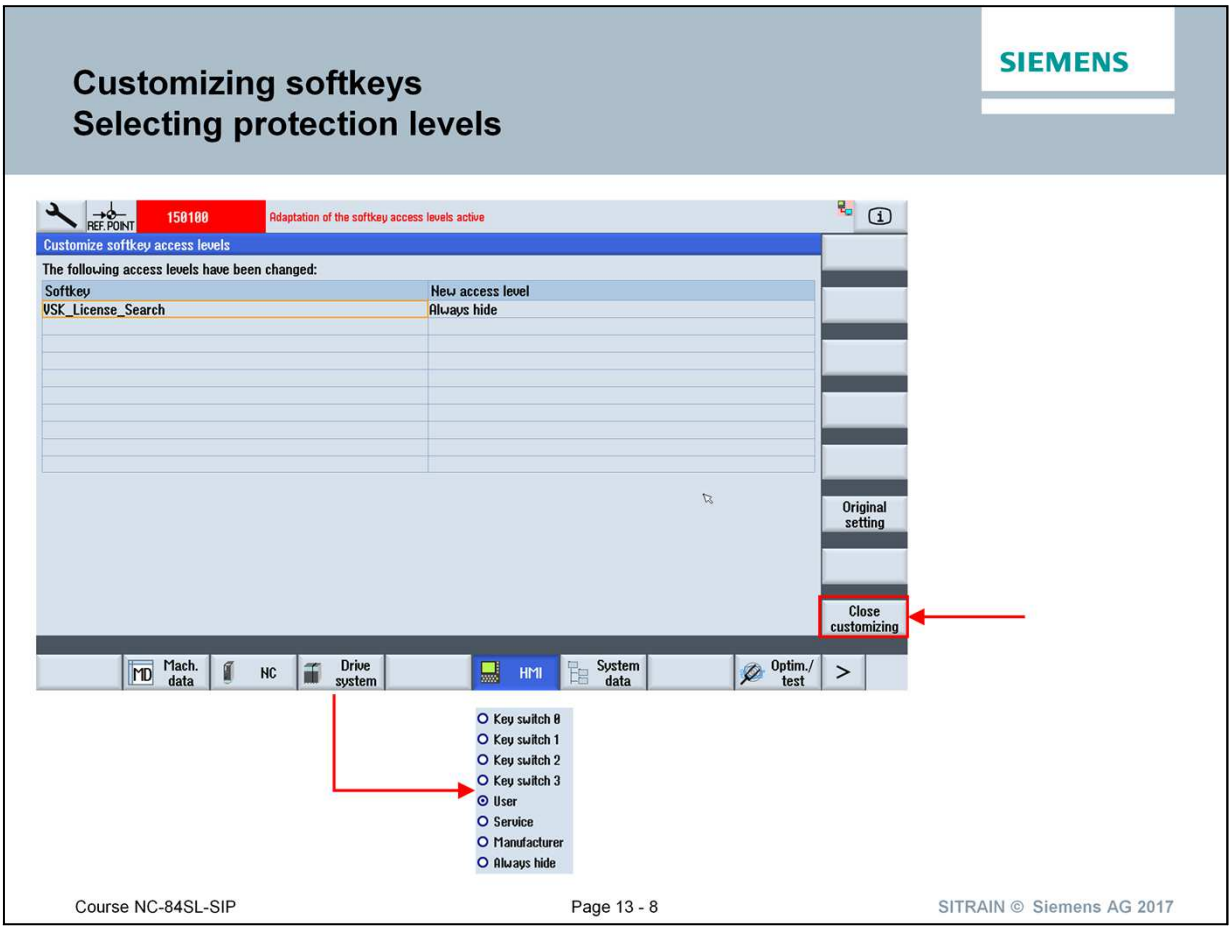
When you open the "Customize softkeys" menu, a brief description of how changes can be made is displayed on the screen. Click "OK" to start customization mode.

Note

Access to the "Customize softkeys" softkey is protected by a general MD:

MD 51073 $MNS_ACCESS_SET_SOFTKEY_ACCESS.

This machine data is preassigned access level 3, i.e. the function is available for users working with the "Service" password and higher.



The softkeys can be customized using the operating screen shown above. All softkeys, even those that are hidden, are displayed in the customization phase. An external mouse is required in order to make changes. All modified softkeys are listed in the table; their original access settings can easily be restored here. At the end of the customization phase, the softkeys are shown or hidden according to the operator protection level that has been assigned to them.

When the changes have been accepted, a configuration file called **"slsoftkeyaccess.xml"** is created and saved in the file system under the following path:

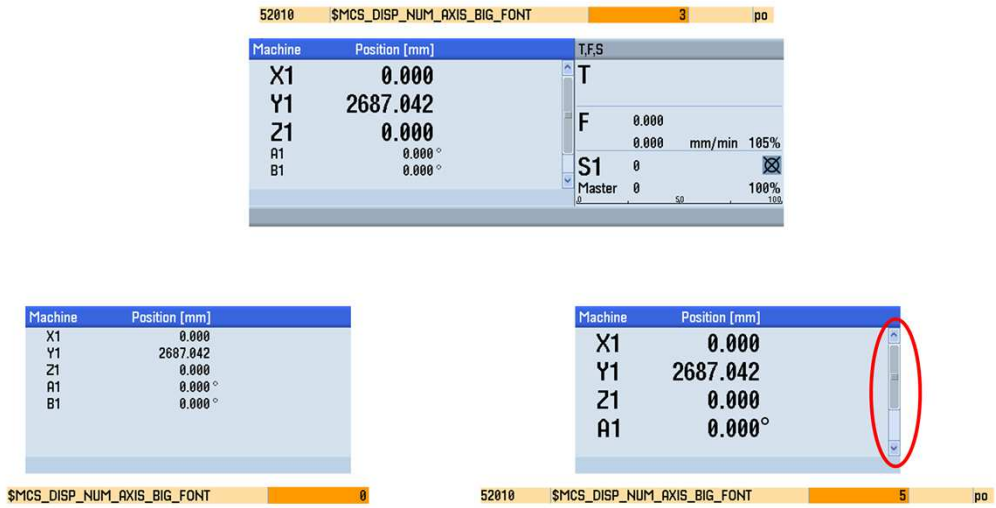
../user/sinumerik/hmi/cfg.

Files can also be saved on the /oem path; these files are evaluated when the system boots. The following priority applies: oem < user.

NOTICE

To avoid errors in the xml syntax and for the softkey identifier, the "slsoftkeyaccess.xml" file may only be edited via the operating software as described above, and not directly in the file system under "System data".

Setting the font size of the actual value display



Course NC-84SL-SIP Page 13 - 9 SITRAIN © Siemens AG 2017

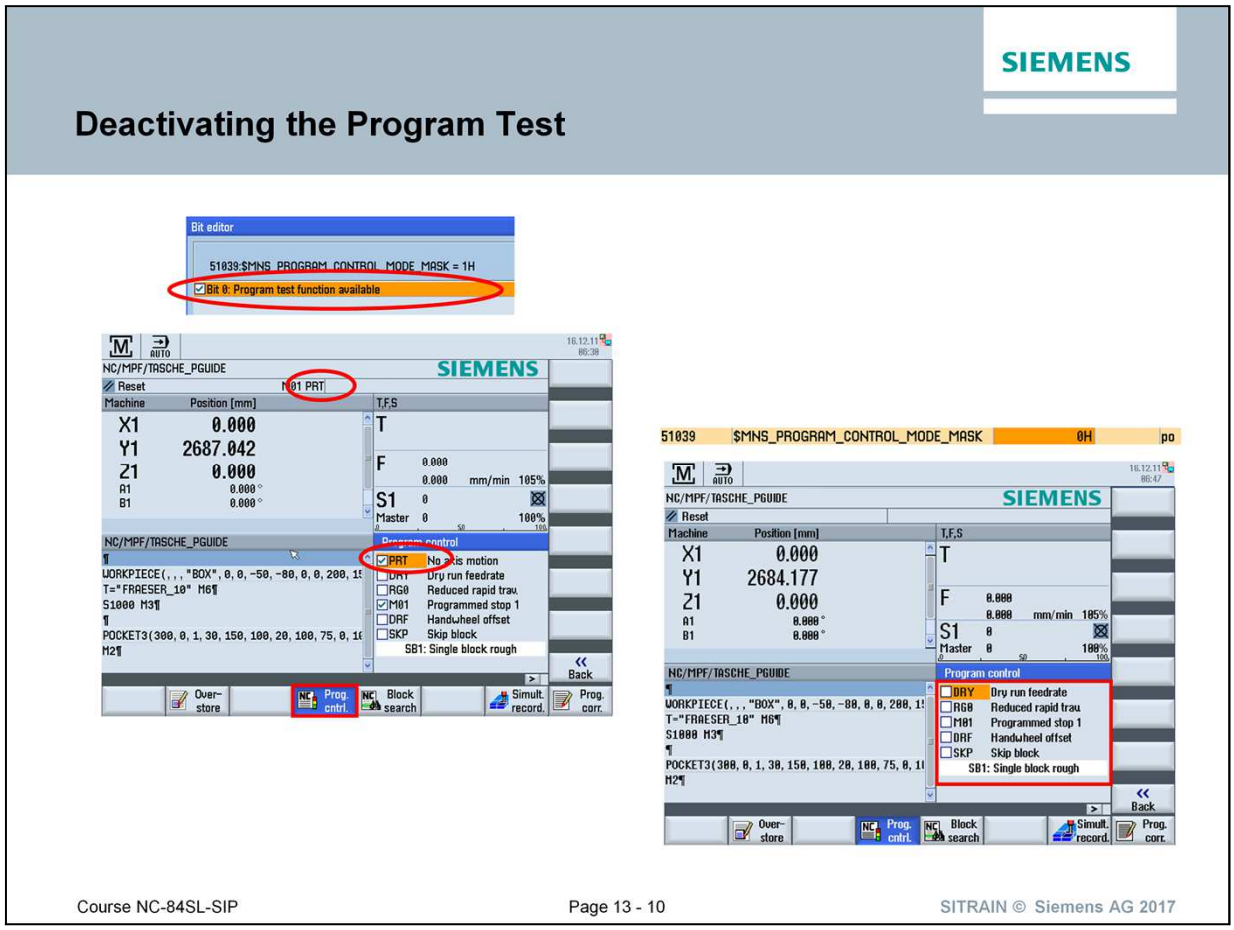
Setting the font

The actual values for the machine coordinate system or workpiece coordinate system can be displayed in two different font sizes in the "Machine" operating area.

Using a channel machine data item, you can set how many axes will be displayed in a larger font.

MD 52010 $MCS_DISP_NUM_AXIS_BIG_FONT = 3

Number of actual values in large font (default)



There are several program test options to test and run in a part program, one of these being the program test. This function is active as standard. You can select the program test in the "Machine" operating area, in the "Program control" window and using the "PRT" check box.

Deactivating the Program Test

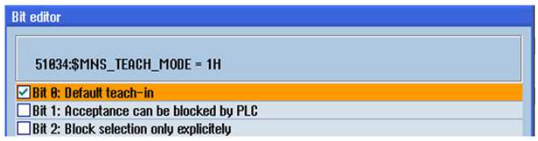
You can remove the Program Test from the user interface using the following configuration machine data. "PRT" is no longer displayed in the status line.

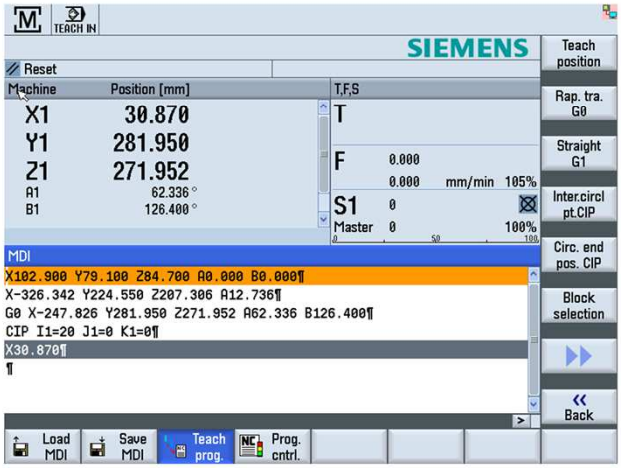
MD 51039 $MNS_PROGRAM_CONTROL_MODE_MASK Bit 0 = 0
-> Program Test function not available

Note

After you have removed the Program Test function, an HMI restart is required. Make sure that the program test is not active when it is removed because otherwise there is no way of shutting it down.

Activating the “Teach in" function





 SITRAIN © Siemens AG 2017

Activate Teach in

The teach in function is activated in part via the general configuration machine data.

Settings

MD 51034 $MNS_TEACH_MODE

Bit 0 = 1 The "Teach Prog." softkey is displayed.
With the softkey "Teach position", traverse commands and positions can be entered or approached in jog mode and accepted in the MDI window.

Bit 1 = 1 Inclusion of the taught block can be blocked by the PLC.
DB19.DBX13.0 = 0 block is added
DB19.DBX13.0 = 1 block is not added

Access to the function still depends on access rights.

MD 51045 $MNS_ACCESS_TEACH_IN = 7	TEACH IN protection level.
MD 51050 $MNS_ACCESS_WRITE_PROGRAM = 7	Write part program protection level.

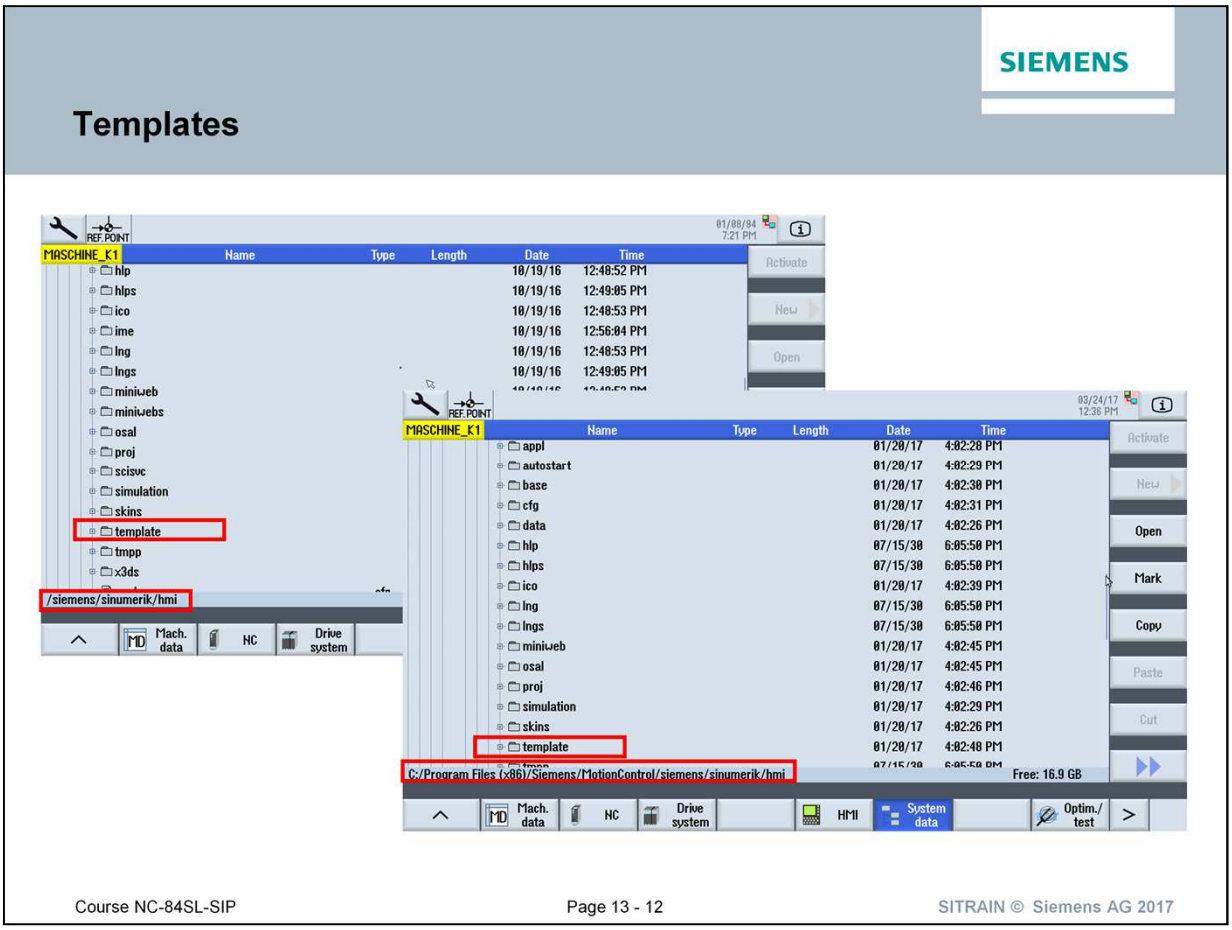
Activate Teach in

1. Set the machine data and initiate a reset.
2. Switch to "AUTO" or "MDA" mode.
3. Select TEACHI IN submode on the machine control panel.
4. Press the "Teach prog." softkey.

Teach in not active

If the "Prog. teach in" softkey is not active, the reasons can be as follows:

- The program is in stop, reset, or interrupt mode.
- Not all of the axes that require referencing have been referenced.



The templates for customizing how SINUMERIK Operate behaves are in the following locations:

PCU: C:\Program Files (x86)\Siemens\MotionControl \sinumerik\hmi\template\cfg

NCU: /card/siemens/sinumerik/hmi/template/cfg

The files are also accessible in **Setup** → **System data** in the directory HMI data/Templates/Examples/Configuration files.

Storage

A copy of the files containing the necessary entries is saved by the machine manufacturer in the following locations:

PCU : C:\ProgramData\Siemens\MotionControl\oem\sinumerik\hmi\cfg

NCU : card/oem/sinumerik/hmi/cfg

If the user customizes the settings further, the modified files are stored in the following locations:

PCU : C:\ProgramData\Siemens\MotionControl\user\sinumerik\hmi\cfg

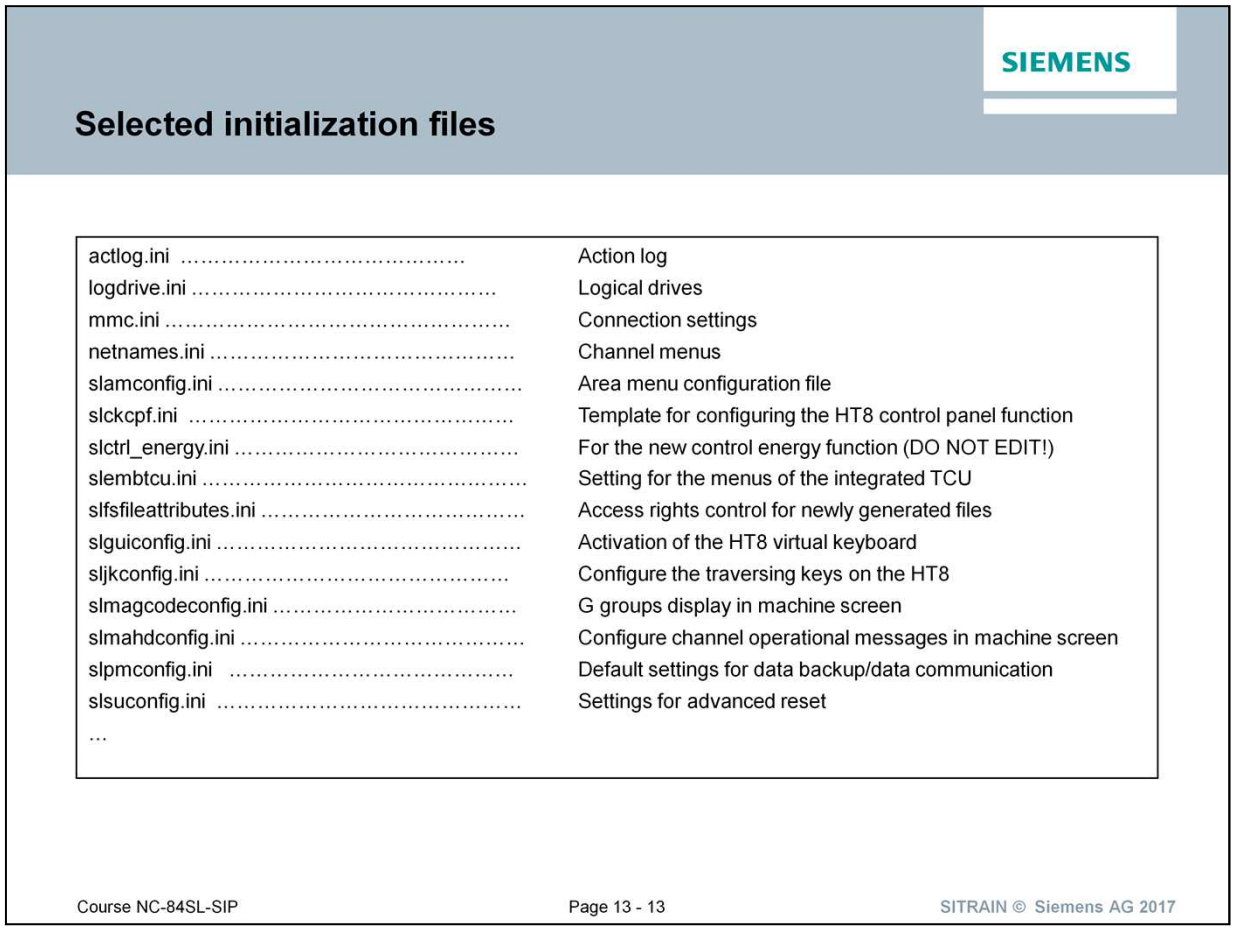
NCU : /card/user/sinumerik/hmi/cfg

Activation

To activate preassigned entries, remove the semicolon that precedes them. A semicolon is followed by a comment up to the end of the line.

Note

As soon as the file is in the user-specific directory, the entries of this file have priority over the Siemens file. If an entry is missing in a user-specific file, the corresponding entry from the Siemens file is used instead.



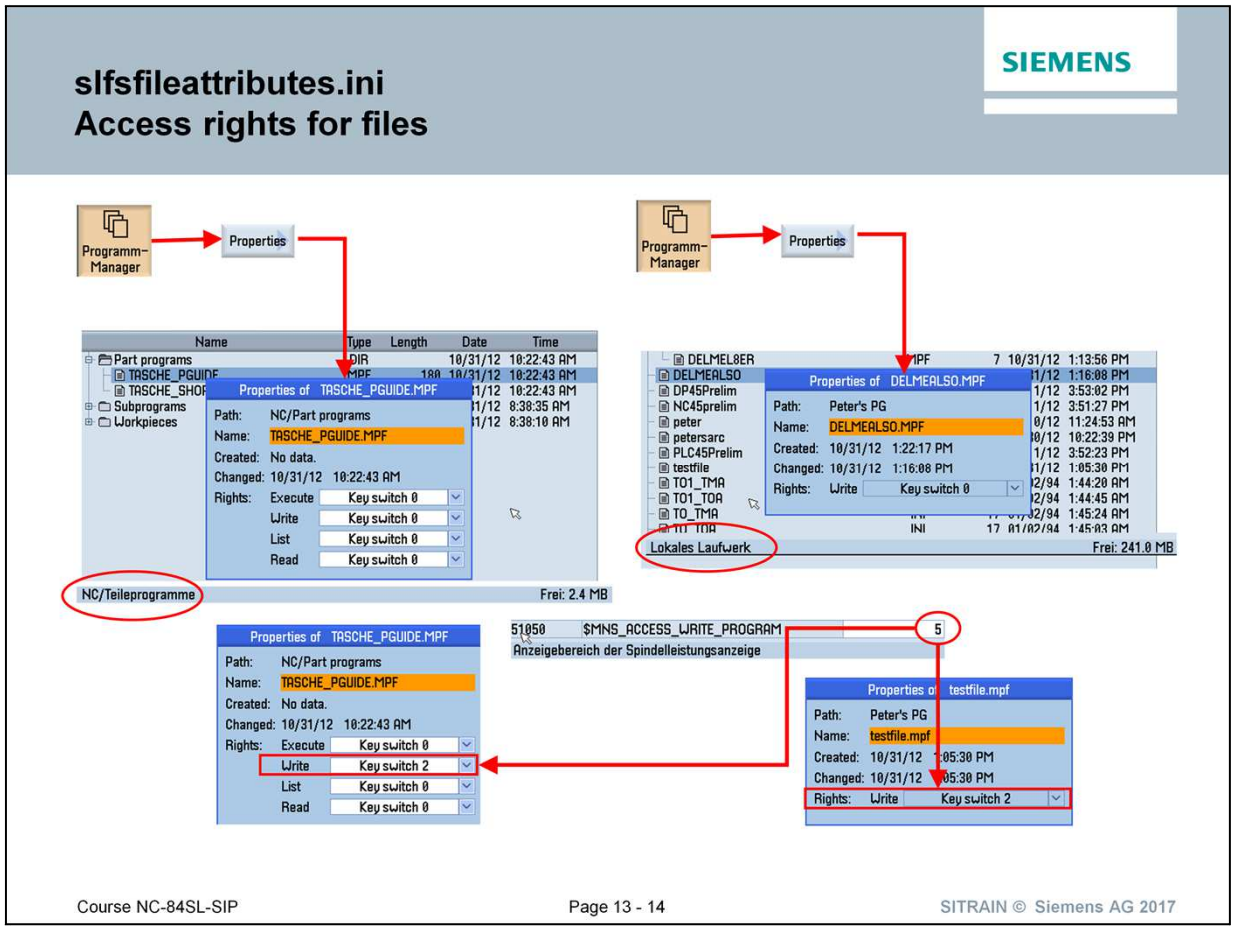
In the screenshot above you can see a selection of initialization files from the directory:

siemens/sinumerik/hmi/template/cfg

Not all of the files can or should be edited. For example, the file slctrl_energy.ini cannot be edited, otherwise there is a risk that
a SENTRON PAC will not be able to evaluate data correctly. Other initialization files such as netnames.ini, on the other hand, can be edited.

Note

Please remember that on Linux systems file name notations must be observed exactly. They are case-sensitive, for example.



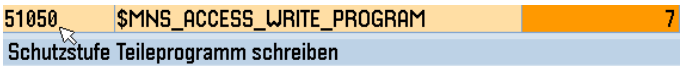
Access rights for execution, writing, listing, and reading are displayed in the "Properties" window.

- Execute: is used for the selection for execution.
- Write: controls the changing and deletion of a file or a directory.

For NC files, you have the option to set the access rights from key-operated switch 0 to the current access level, to be set separately for each file. If an access level is higher than the current access level, it cannot be changed. For external files (e.g. on a local drive), the access rights are displayed to you only if settings have been executed for these files by the machine manufacturer. They cannot be changed via the "Properties" window.

Configuration file slfsfileattributes.ini

The access rights defined in the configuration file **slfsfileattributes.ini** and in **MD 51050**



are relevant in the NC when creating new files. They can subsequently be modified individually in the Program Manager with the Properties function. The access rights are an individual attribute of each file in the NC. The access rights defined in the configuration file **slfsfileattributes.ini** and MD 51050 become valid for external data on the CF card of the NCU or on the hard disk of the PCU/PC immediately for all the specified directories and file types; they cannot be changed by means of operator intervention. The access rights can only be set globally for external data.

slfsfileattributes.ini Structure

SIEMENS

```
[ACCESSMASKS]
 ; settings for nck-filesystem
;\mpf.dir\*.mpf = 6577          ; mask for mpf-files in partprogram directory
;\spf.dir\*.spf = 6577          ; mask for spf-files in subprogram directory
;\cus.dir\*.spf = 6577          ; mask for spf-files (cycles) in user cycles directory
;\cma.dir\*.spf = 7377          ; mask for spf-files (cycles) in manufacturer cycles directory
;\wks.dir\*.wpd = 7777          ; mask for .wpd-directories (workpiece-directories) in workpieces directory
;\wks.dir\*.wpd\*.mpf = 6577    ; mask for mpf-files in all workpiece directories
;\wks.dir\*.wpd\*.spf = 6577    ; mask for spf-files in all workpiece directories
;\wks.dir\xyz.wpd\*.* = 6507    ; default mask for other files in all workpiece directories
;\wks.dir\xyz.wpd\*.mpf = 6577  ; mask for mpf-files in the workpiece directory xyz

; settings for local drive; paths relative to the directory 'data', i.e /user/sinumerik/data
;prog\mpf.dir = 7507            ; mask for the partprogram directory
;prog\spf.dir = 7507            ; mask for the subprogram directory
;prog\wks.dir = 7607            ; mask for . workpieces-directory
;prog\wks.dir\*.wpd = 7607      ; mask for .wpd-directories (workpiece-directories)
;archive = 7507                 ; mask for the archive directory
;*.arc = 7707                   ; mask for the archive-files
;*.ard = 7707                   ; mask for easy archive files 828D
;*.mpf = 6577                   ; mask for mpf-files
;*.spf = 6577                   ; mask for spf-files
;*.*   = 6507                   ; default mask for other files
```

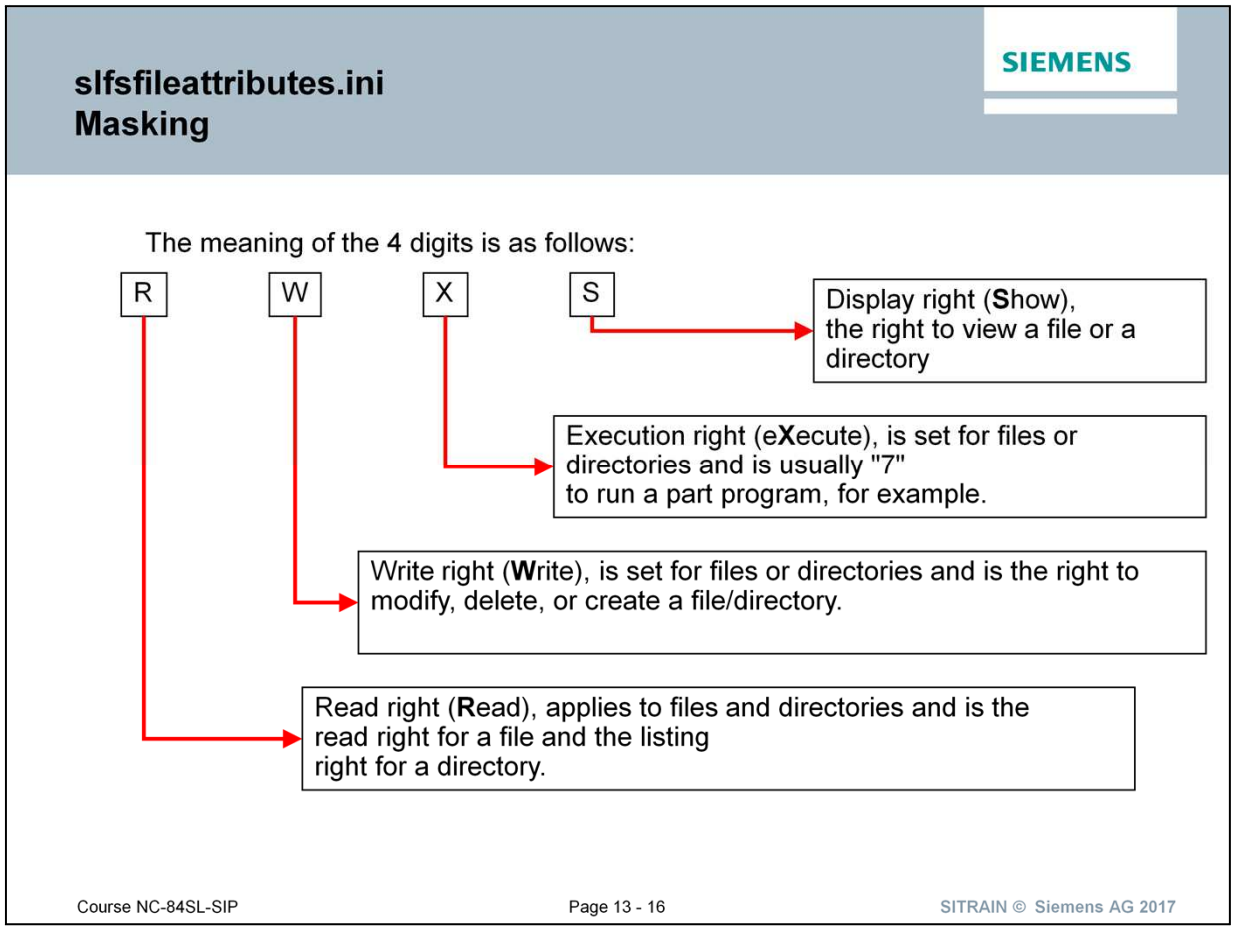
Course NC-84SL-SIP Page 13 - 15 SITRAIN © Siemens AG 2017

Syntax

"\" or "/" are separators for directories.
For names, upper and lower case are not important.

Search strategy for files

Explicit data of paths and file names have higher priority than placeholders.



Access level

is a 4-digit string containing values between 0 and 7, where

0 = system access level (SIEMENS)

:

7 = access level key-operated switch "0" (operator).

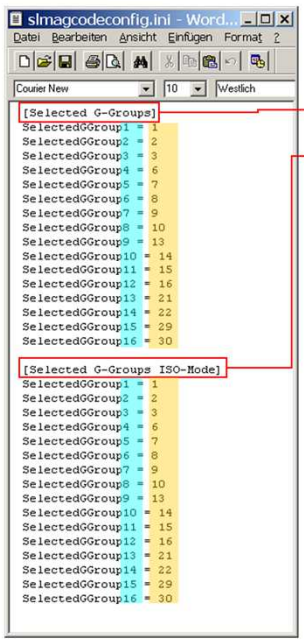
Note

The current value of the NCK machine data 51050 $MNS_ACCESS_WRITE_PROGRAM is 'mixed' with the value in the slfsfileattributes.ini file (the lower value of the access level applies).

NCK machine data	slfsfileattributes.ini	Result
$MNS_ACCESS_WRITE_PROGRAM=4	*.mpf = 6577	*.mpf = 6477
$MNS_ACCESS_WRITE_PROGRAM=6	*.mpf = 6577	*.mpf = 6577

The result is always the more restricted access right.

slmagcodeconfig.ini
Configuring the display of the G code groups



Section	Meaning
Selected G groups	This area is intended for operation of the controller in Siemens mode.
Selected G groups ISO mode	This area is intended for operation of the controller in ISO mode.

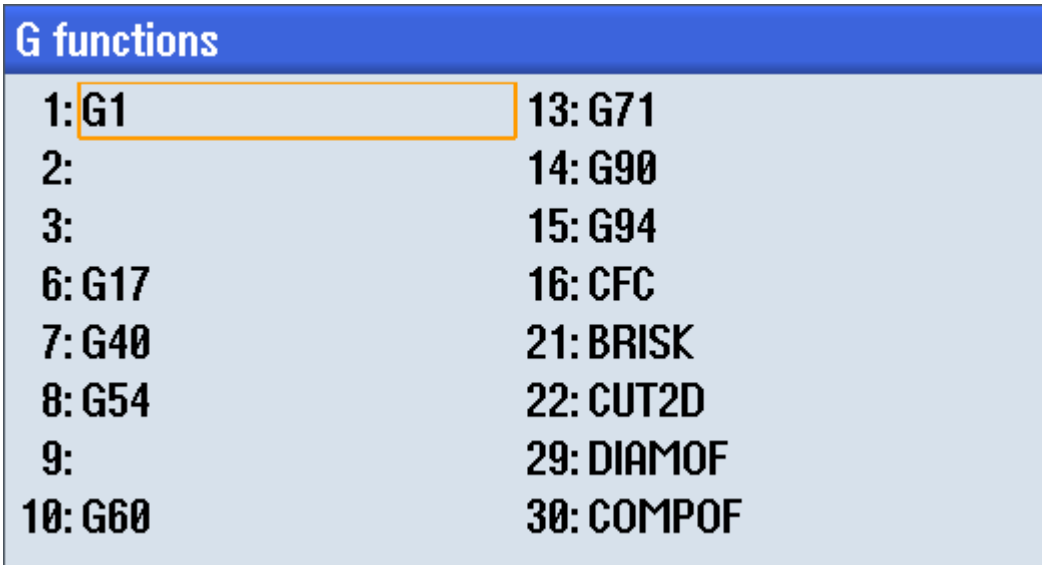
SelectedGGroupN = M	
N	Position where G-code group M will be displayed. If you want to leave a position empty, omit the number (M), e.g. "SelectedGGroup1 = "
M	The number of the G-code group to be displayed at position N

Course NC-84SL-SIP Page 13 - 17 SITRAIN © Siemens AG 2017

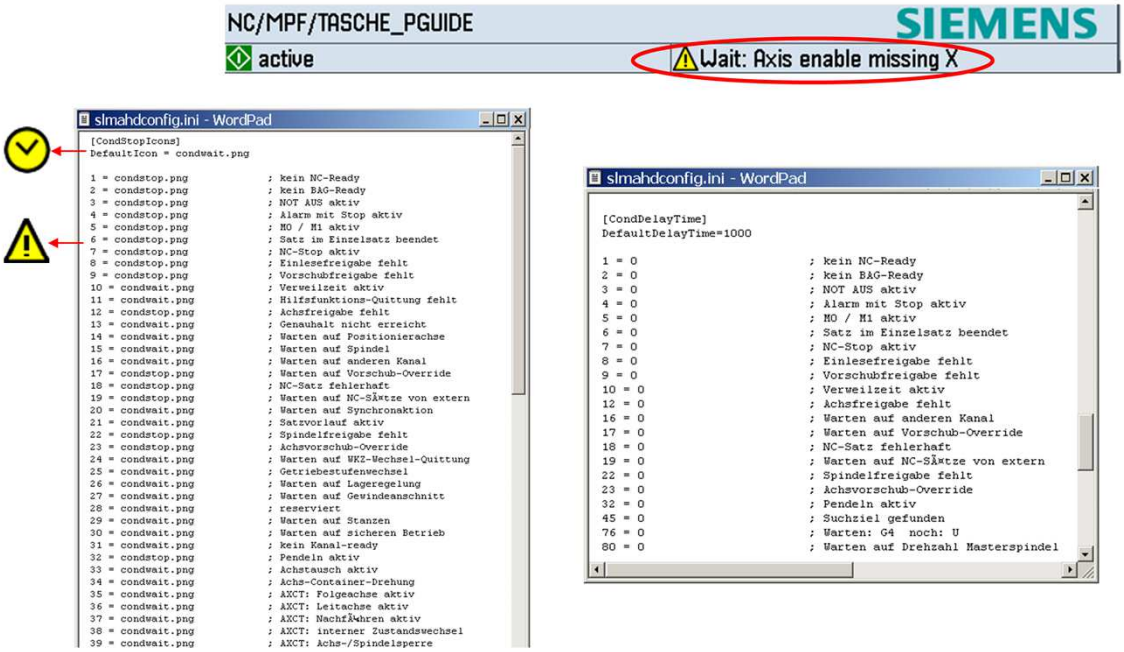
The "G functions" pane is visible in the "Machine" operating area. Up to a maximum of 16 G code groups are displayed in this pane. Configure the **"slmagcodeconfig.ini"** file accordingly to determine which G code groups should be displayed, and where.

Procedure

1. Copy the file from the system into the OEM or user directory.
2. Open the file and enter the number of the G code in question at the required position.



slmahdconfig.ini
Configuring the channel operational message



SITRAIN © Siemens AG 2017

The channel operational messages are displayed in the third line of the header in the "Machine" operating area. There are two types of channel operational messages:

1. Channel operational messages that require an operator action to continue a program, e.g. feed enable missing, M01 active. These channel operational messages are identified by exclamation marks.
2. Channel operational messages that do not require an operator action to continue a program. A stop condition is active for a length of time (>1 s) and is interrupted by the program processing, e.g. dwell time active, waiting for tool change. This channel operational message is identified by a clock icon and disappears after a certain time.

File

The settings for this are specified in the "**slmahdconfig.ini**" file.

Procedure

Copy the file from the system into the OEM or user directory and edit it.

Delay time

A delay time can be displayed in the [ConditionDelayTime] section for the various displays. If a delay time is not set, the default time applies.

slmahdconfig.ini
User status display

SIEMENS

```
[TPM]
TPM=0
TPMGreen=1
TPMYellow=2
TPMRed=3
TPMPLCData=DB59.DBB800

; The section UserIcons enable you to show your own icons (up to 16)
; in the machine header instead of the actual program name.
; The icons can be displayed dynamically by settings a plc double word.
[UserIcons]
USER_ICON_BASE=DB19.DBB80     ; set the start byte of the plc double word that displays the icons (DB19.DBD80)
UI_0 = icon1.png,1            ; show icon1.png on position 1 if bit 0 (DB19.DBX83.0) is set
UI_1 = icon2.png,2            ; show icon2.png on position 2 if bit 1 (DB19.DBX83.1) is set
                              ; this overwrites the icon from the first line
;...
;UI_31 = icon3.png,1          ; show icon3.png on position 1 if bit 31 (DB19.DBX80.7) is set

; The sections PosXY enable you to show channel status icons in the machine header
; instead of the actual program name. They compete for a free position with the icons defined
; in the UserIcons section. Each icon is double wide and takes position XY and XY+1.
; There are four types of channel status icons
;   ChanStat                    shows the channel status of the given channel
;   SpindStat                   shows the spindle status of the given spindle
;   ChanStatFeedStop            shows the channel and feed stop status of the given channel
;   ChanStatFeedStopSpindleStop shows the channel, feed stop and spindle status of the given channel
;                               the spindles are configured by an additional Spindles-Entry
; e.g.
;[Pos4]                          ;show on positionn 4 and 5 the
;ChanStatFeedStopSpindleStop=2   ;channel status of channel 2 overlayed by feed stop and
;Spindles=1,2                    ;spindle state of spindles S1 and S2
;
```


SITRAIN © Siemens AG 2017

Function

User icons can be displayed in the Machine operating area in the second line in the header. In this case, the program name is displayed in the right-hand field of the third line in the header. Active hold conditions hide the program name.

The display of the user icons is controlled via PLC bits. The icons are provided in the form of PNG files and stored by specific resolution in the file system under:

/user/sinumerik/hmi/ico/ico640, etc.

Up to 16 positions are available in the header to display the user symbols.

Superimposition

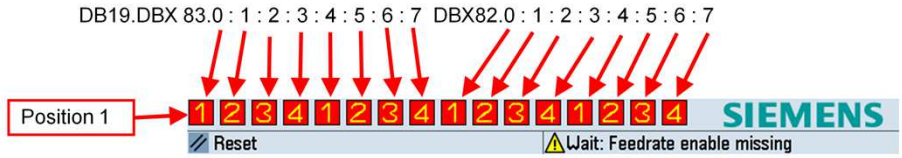
If the selection is not unique (bits for multiple identifiers, whose image is to be displayed in the same position):
The image with the higher identifier number is superimposed on the image with the lower identifier number for the same position. Non-unique selection can also be used with less than 16 active selection bits. The PLC can deliberately use this superimposition to overlay less important displays (with lower identifier numbers) with important displays (high identifier numbers).

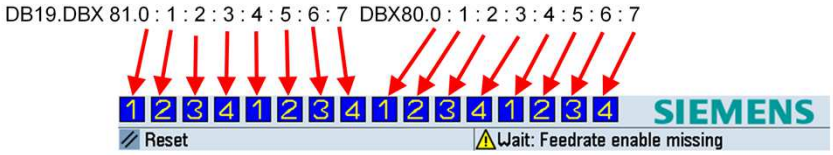
Example

A user icon with number UI_3 is hidden by a user icon UI_17 if both are displayed in the same position and both activation bits are set to state "1".

slmahdconfig.ini
User status display control by PLC

SIEMENS



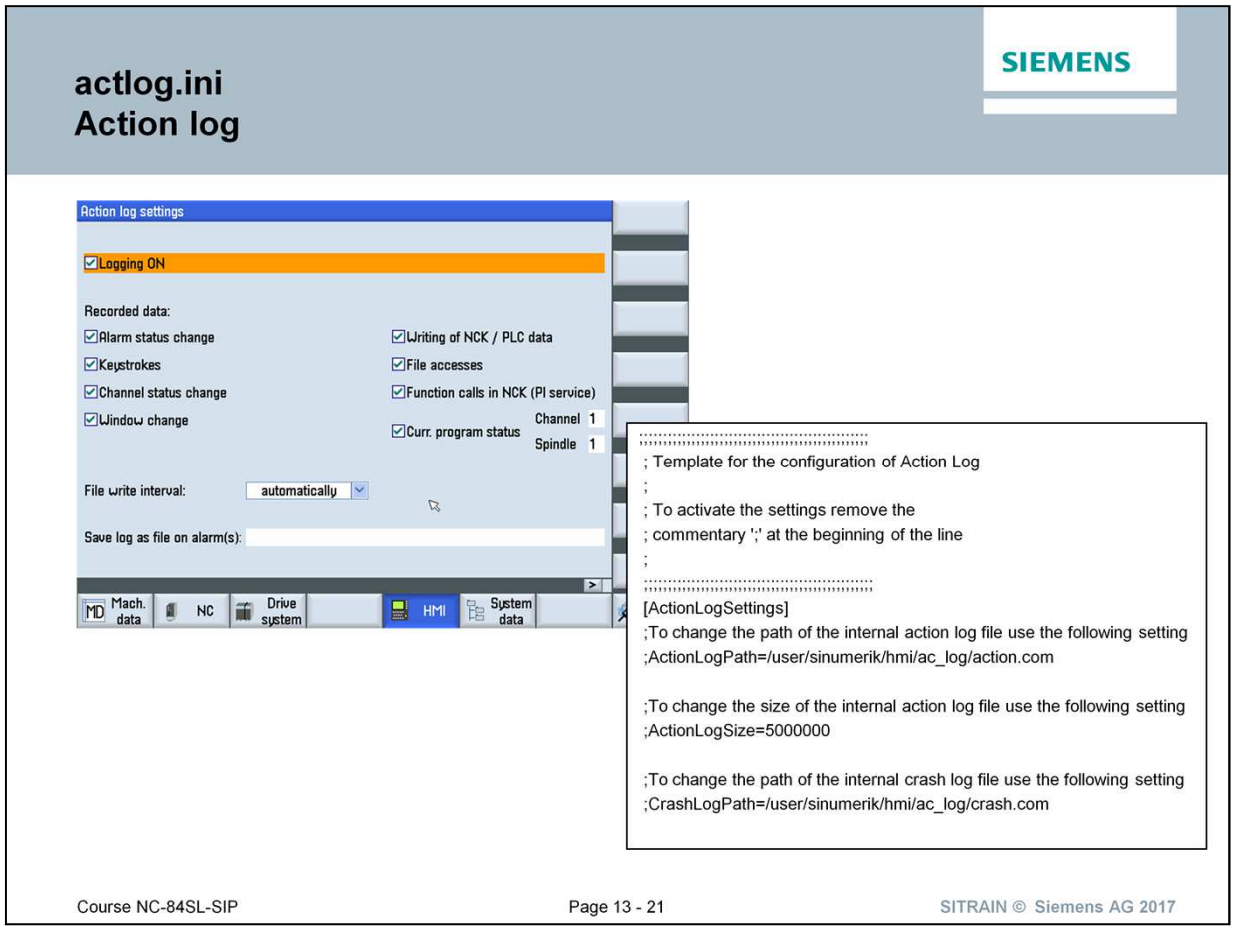


Higher priority bytes DBB80 and DBB81 overwrite DBB83 and DBB82 icons, since bytes 80 and 81 address user icons 16 to 31.

 SITRAIN © Siemens AG 2017

```
[UserIcons]
USER_ICON_BASE=DB19.DBB80  ; set the start byte of the plc double word that displays the icons
UI_0 = icon1.png,1              ; show icon1.png on position 1 if bit 0 (DB19.DBX83.0) is set
UI_1 = icon2.png,2              ; show icon2.png on position 2 if bit 1 (DB19.DBX83.1) is set
…
UI_6 = icon3.png,7              ; show icon3.png on position 7 if bit 6 (DB19.DBX83.6) is set
UI_7 = icon4.png,8              ; show icon4.png on position 8 if bit 7 (DB19.DBX83.7) is set
…
UI_8 = icon1.png,9              ; show icon1.png on position 9 if bit 8 (DB19.DBX82.0) is set
UI_9 = icon2.png,10             ; show icon2.png on position 10 if bit 9 (DB19.DBX82.1) is set
…
UI_14 = icon3.png,15            ; show icon3.png on position 15 if bit 14 (DB19.DBX82.6) is set
UI_15 = icon3.png,16            ; show icon4.png on position 16 if bit 15 (DB19.DBX82.7) is set
…
…
UI_16 = icon1b.png,1            ; show icon1b.png on position 1 if bit 16 (DB19.DBX81.0) is set
…
UI_23 = icon4b.png,8            ; show icon4b.png on position 8 if bit 23 (DB19.DBX81.7) is set
…

UI_24 = icon1b.png,9            ; show icon1b.png on position 9 if bit 24 (DB19.DBX80.0) is set
…
UI_31 = icon4b.png,16           ; show icon4b.png on position 16 if bit 31 (DB19.DBX80.7) is set
```



You will find the action log settings in the "Setup" > "HMI" > "Diagnostics" > "Action log" OA.

Action log

In the "Action Log Settings" window, you set whether the operating sequences will be logged. Here, you select which control actions will be logged, so that sequences can be traced at a later date. When the action log is active, you can display recorded traces in the system data using the following files:

- "action.com", action log
- "crash.com", crashlog backup

Setting

You define the file size and storage path of the log file of the action log in the "actlog.ini" configuration file.

Procedure

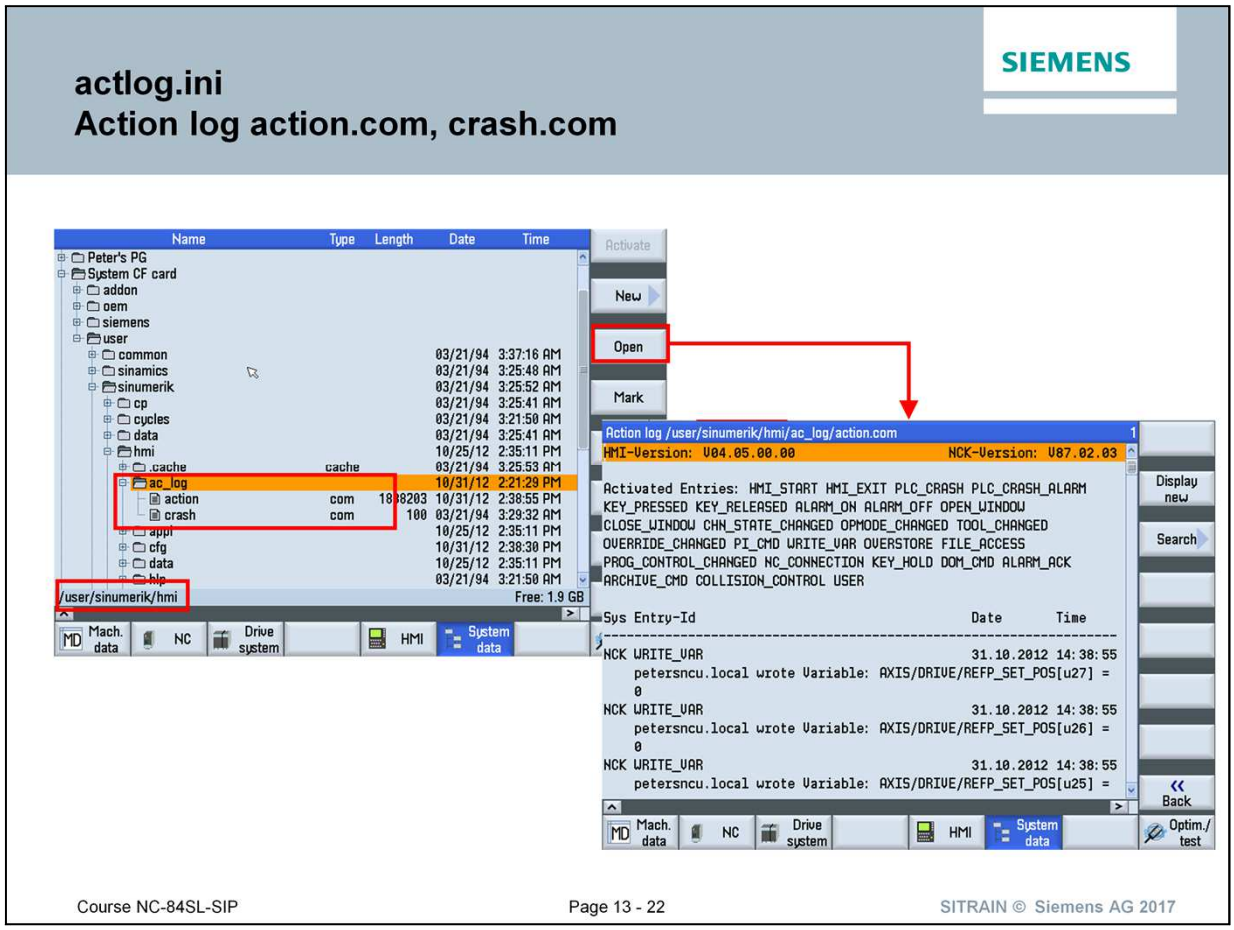
1. Copy the file from the system into the OEM or user directory.
2. Open the file and after "; ActionLogSize=", enter the file size (in bytes).

Crashlog file

"actual_crashlog.com" is a backup file of the actual log when a critical event occurs (for example, EMERGENCY OFF alarm). The file is then no longer stored in the ring buffer of the action log and can no longer be overwritten with new entries.
The entries in the crashlog file are not overwritten until after a further critical event.

At the beginning, the file has no entries and is only written to after the following events:

- Interface signal DB19.DBX0.6 "Save action log" changes from 0 to 1
- The alarm entered in the "Save log as file for alarm(s)" entry field is output.



Action log Display

You can find the log file of the action log like this:
OA: Setup > **softkey: System data >** directory: **HMI data/Logs/Action log**

(This is in the directory:

/card/user/sinumerik/hmi/log/action_log for Operate on NCU

or

C:\ProgramData\Siemens\MotionControl\user\sinumerik\hmi\log\action_log for Operate on PCU)

-> File **actual_actionlog.com**

Readable copy

When **softkey: Save log** is pressed, 2 files are created on the specified path from the file that is open in the editor:

action.com binary file (copy of file ../hmi/ac_log/action.com)

action.log text version of this file

The latter file can be edited with a text editor.

Crash log Display

The crash log can be opened as follows:

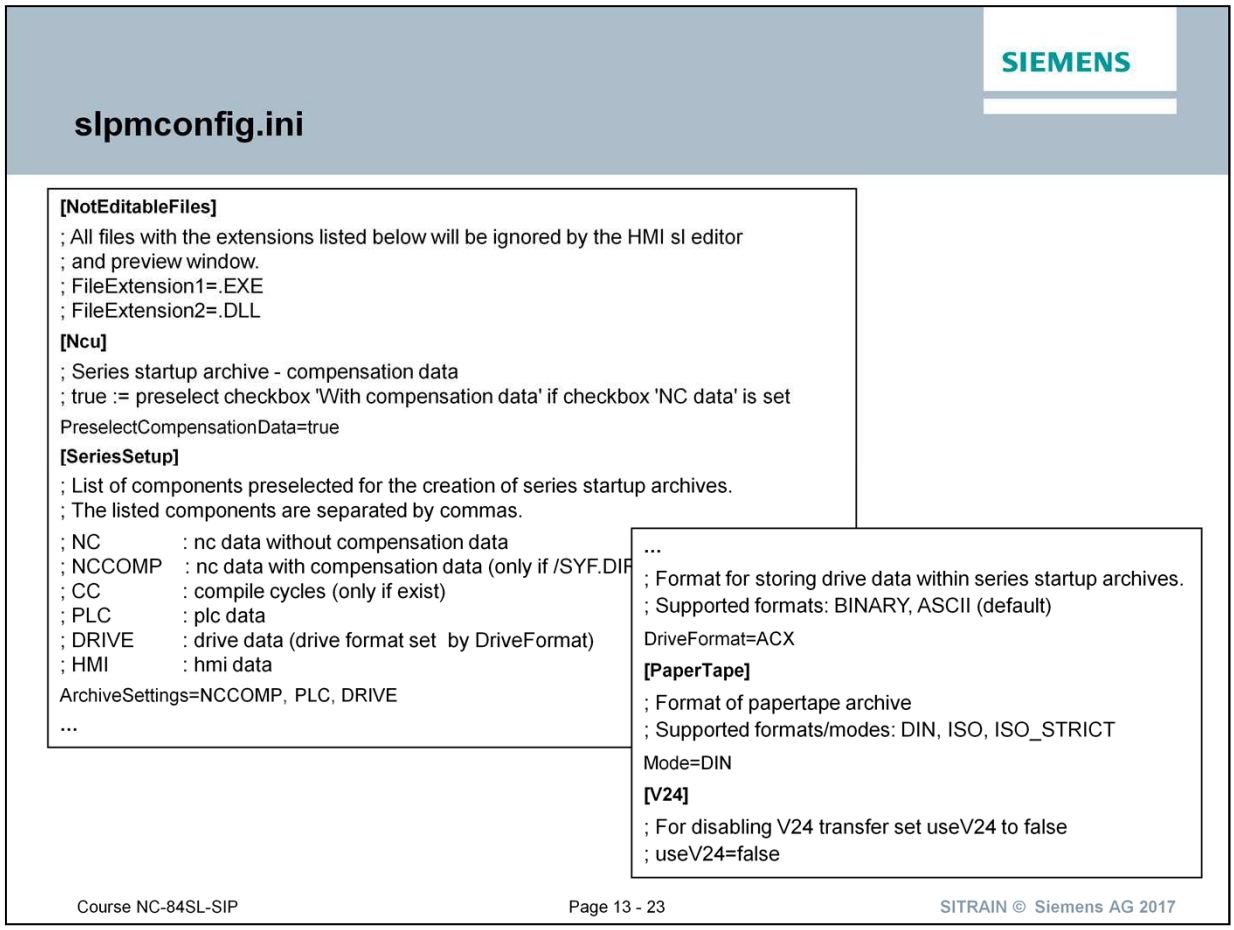
-> file: **actual_crashlog.com**

Readable copy

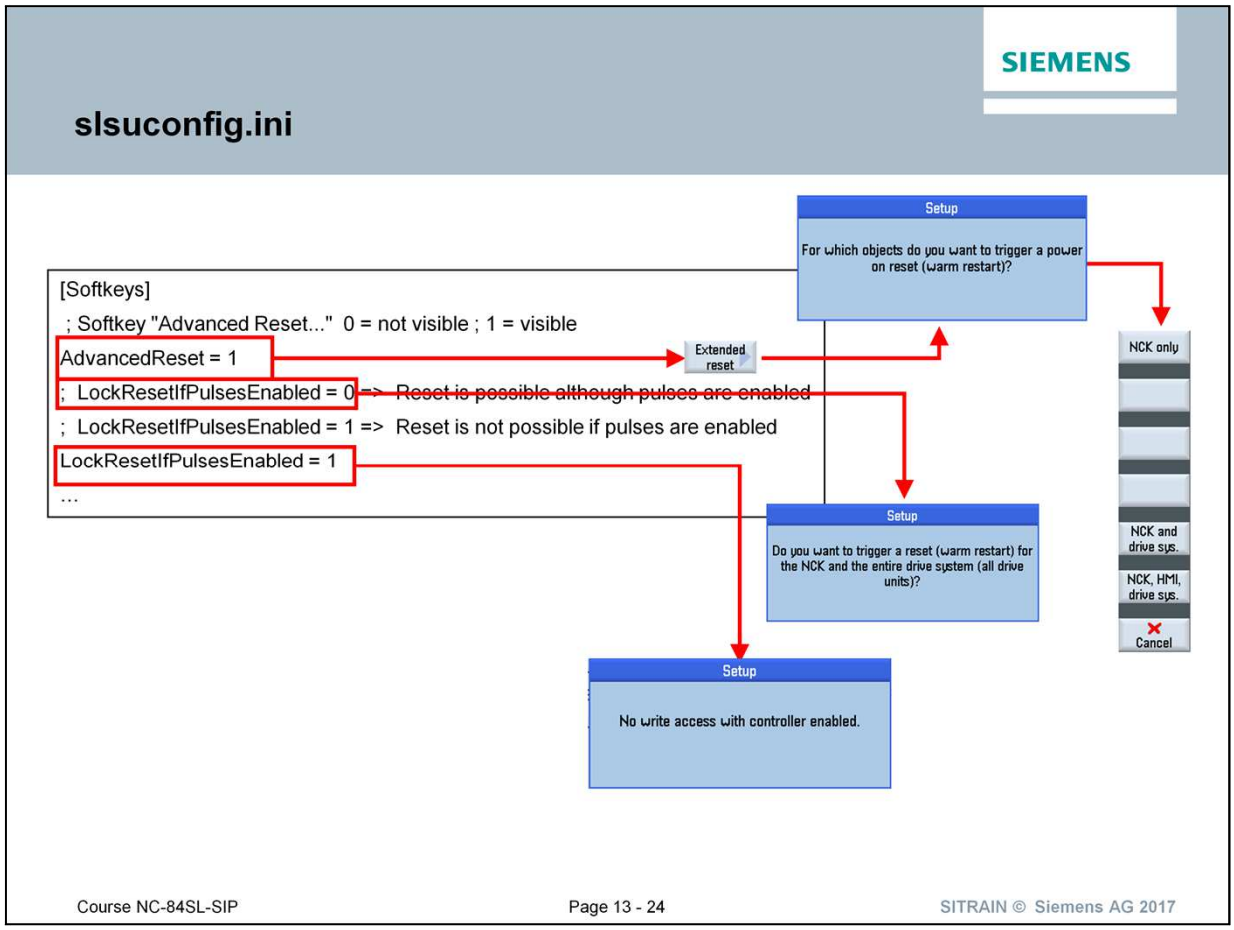
When **softkey: Save log** is pressed, 2 files are created on the specified path from the file that is open in the editor:

crash.com binary file (copy of file ../hmi/ac_log/crash.com)

crash.log text version of this file



[NotEditableFiles]	Enter all file types that cannot be edited by the editor in this section.
[Ncu]	Set this parameter to specify whether or not the setting for NCU data backup has activated the preassignment for the compensation data.
[SeriesSetup]	In this section you specify which data backup elements are preselected. You can also preassign the data format for the drive data backup.
[PaperTape]	The preassignment for the "tape" data format is specified here.
[V24]	Activate or deactivate the V24 interface



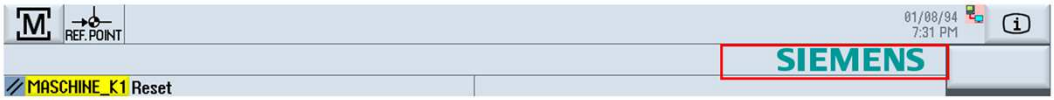
Hide the "Extended reset" softkey

The "Extended reset" softkey is defined in the "slsuconfig.ini" file. If the softkey is not to be displayed, proceed as follows:

1. Copy the file from the system into the OEM or user directory.
2. Open the file and in the section
   [Softkeys] and enter
   AdvancedReset = 0

Own logo in the Machine OA

Folder	Bitmap - Size
/lco640	122 x 19
/lco800	160 x 24
/lco1024	199 x 30



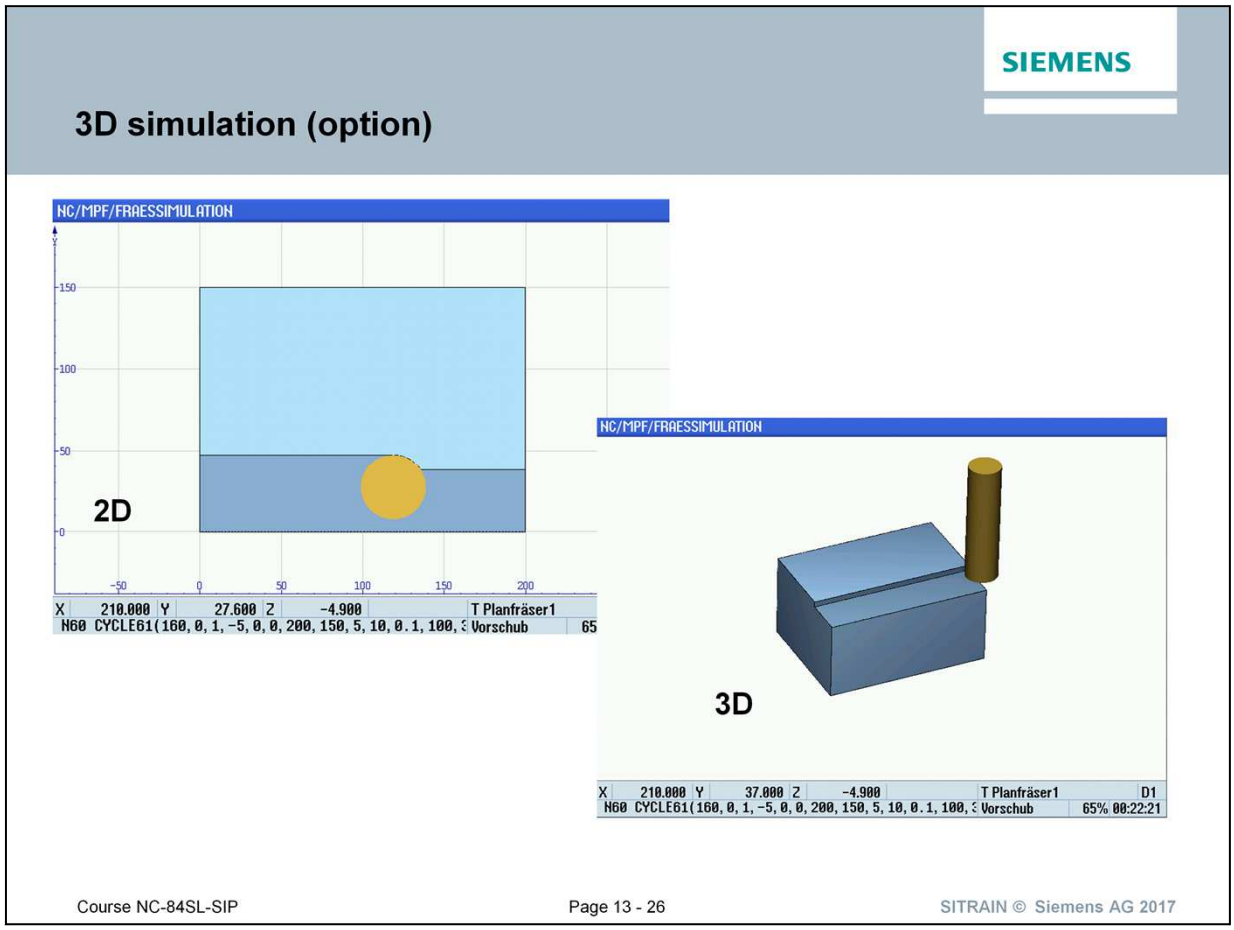
Course NC-84SL-SIP Page 13 - 25 SITRAIN © Siemens AG 2017

The SIEMENS logo is visible in the header of the "Machine" operating area. You can replace the SIEMENS logo with your own logo. Procedure

1. Name your own logo "logo.png".
2. Save your own logo, depending on the bitmap size, in one of the directories: /oem/sinumerik/hmi/ico or /user/sinumerik/hmi/ico

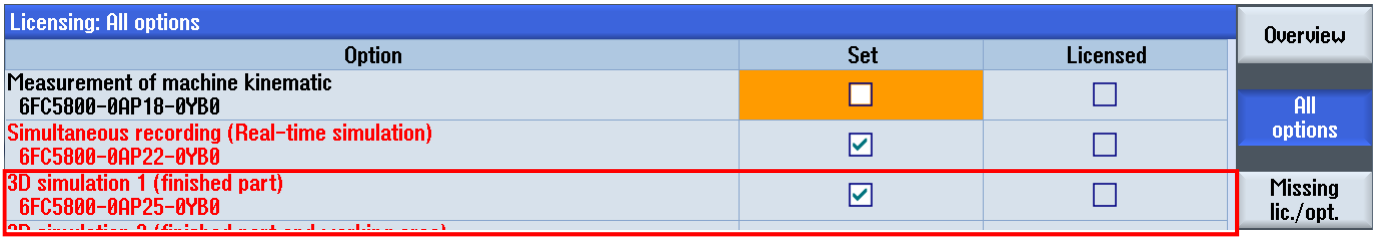
Note

You will find templates for logo.png in the various resolutions on the path card/siemens/sinumerik/hmi/ico.

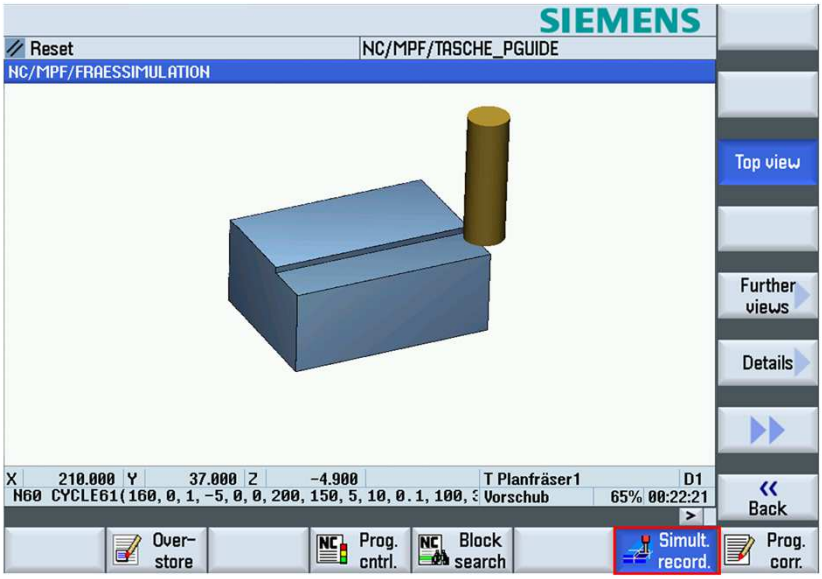


Simulation enables you to display real machining of a workpiece to eliminate any remaining errors. 3D simulation goes a step further, making the simulation more vivid.

Note For simulation, option "3D Simulation 1" P25 is necessary.



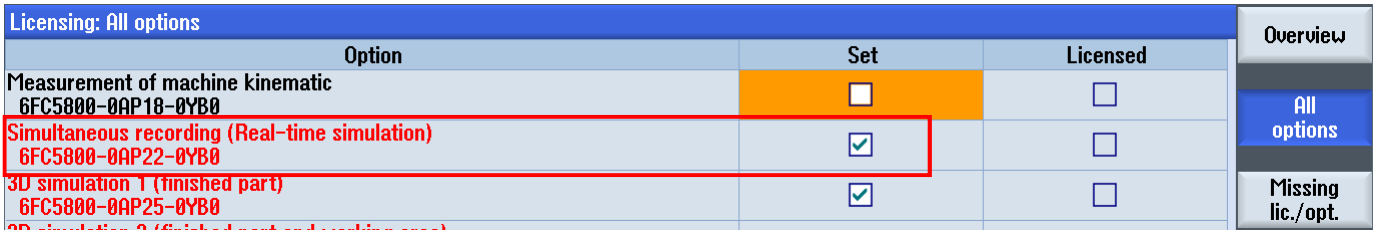
Simultaneous recording



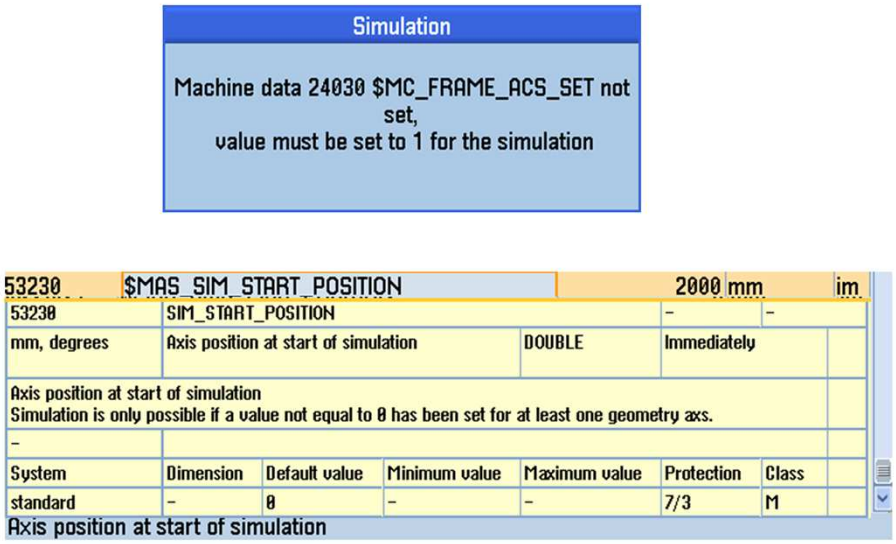
The simultaneous recording function shows the state of the workpiece as it is machined on the machine during runtime. It is comparable with the simulation function in that it works with the same display method. However, the machining step that is being displayed is actually being performed simultaneously in real time. So the machining time is real, for example, whereas with simulation, non-productive time (tool changes, acceleration ramps, etc.) are not taken into account.

Note

The "Simultaneous recording (real-time simulation)" P22 option is required for simultaneous recording.



Machine data for simulation



Course NC-84SL-SIP Page 13 - 28 SITRAIN © Siemens AG 2017

For simulation, set some of the machine data shown on the following pages. Please also remember that movements from program parts that are not immediately obvious, for example (e.g. from a tool change) are also included in the display.

MD 24030 Machine data

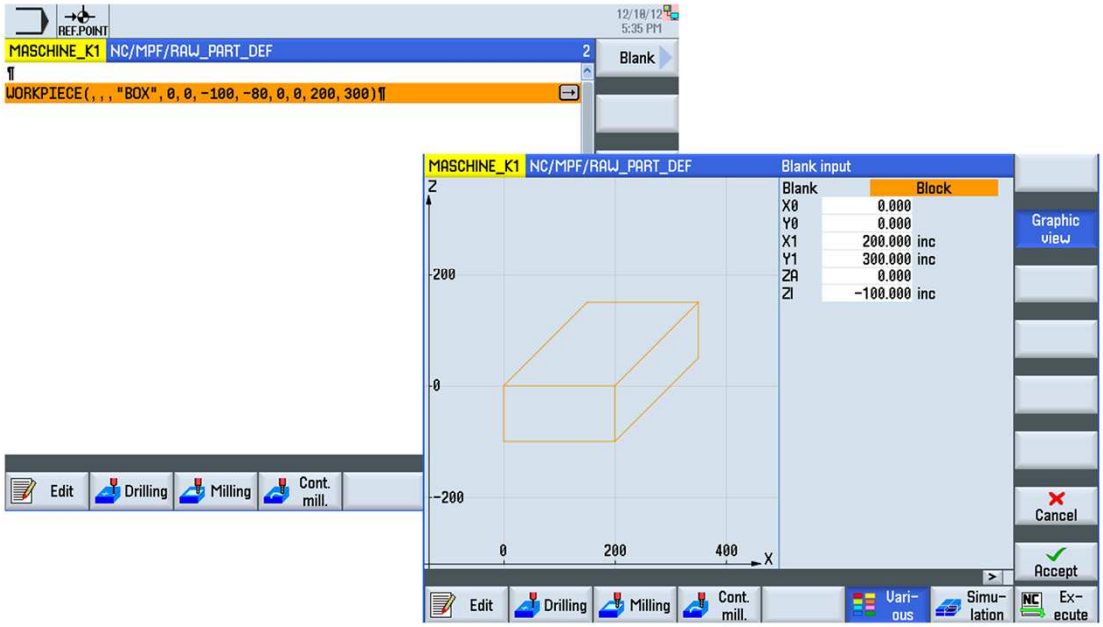
MD 24030 $MC_FRAME_ACS_SET

must be set to "1" to use the simulation.

MD 53230 You set the position of the axes at the start of simulation in

MD 53230 $MAS_SIM_START_POSITION.

Raw part definition for simulation/simultaneous recording



 SITRAIN © Siemens AG 2017

Raw part definition

The blank dimensions that are entered in the program editor are used for the workpiece. The blank is clamped with reference to the coordinate system that was valid when the blank was defined.

This means that before the blank is defined in G code programs, the required output conditions must be established, e.g. by selecting a suitable zero offset.

The blank is set up in the programming settings under "Miscellaneous" > "Blank" and added to the part program by selecting "Accept".

Programming a blank (example)

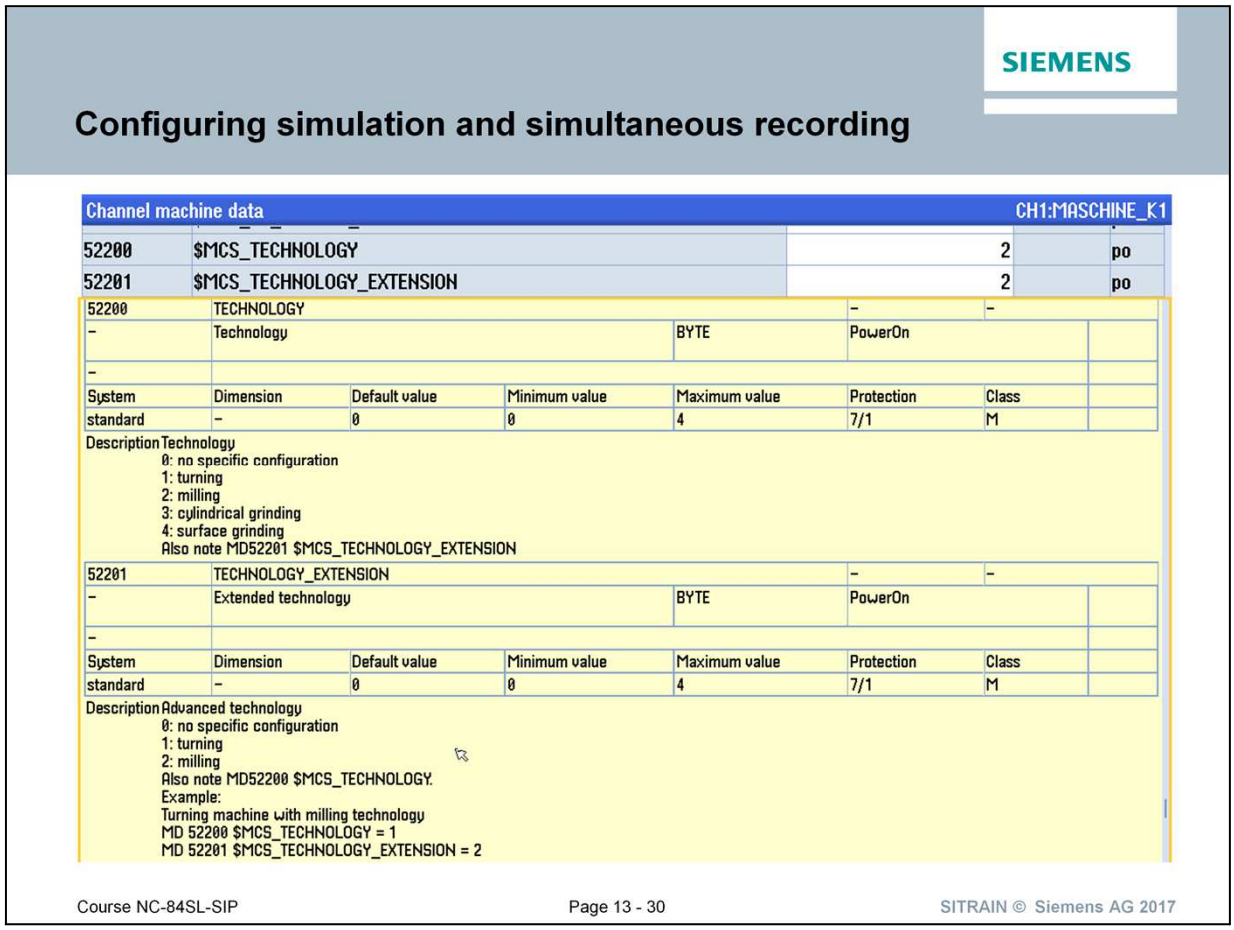
```
G54 G17 G90
CYCLE800(0,"TISCH", 100000,57,0,0,0,0,0,0,0,0,-1,100,1) ;Swivel
WORKPIECE(,,,"Box",112,0,-50,-80,00,155,100)
T="NC-ANBOHRER_D16
```

Note

If the starting points for the axes are too close to the blank in the simulation (see MD 53230 on the previous page), the selected tool might collide directly with the blank. Therefore, a sufficient clearance should be maintained.

ShopMill / ShopTurn

With this type of programming, the blank is defined in the program header!



Displaying machining operations

Turning technology

- Conventional turning with two geometry axes
- Three spindles: Main spindle, counterspindle, tool spindle
- Counterspindle slide, tailstock as NC axis
- B axis: Aligning turning tools in the tool spindle

Milling on turning machines

- Milling with geometry axes: TRANSMIT, TRACYL, TRAANG

Milling technology

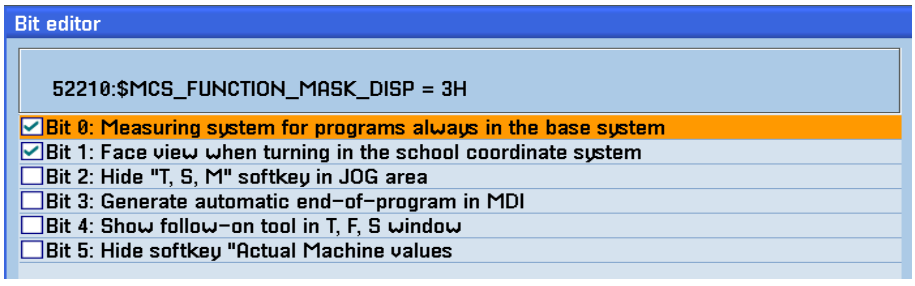
- Milling with 5 axes: Swivel/TRAORI
- Swivel head change

Configuring the simulation

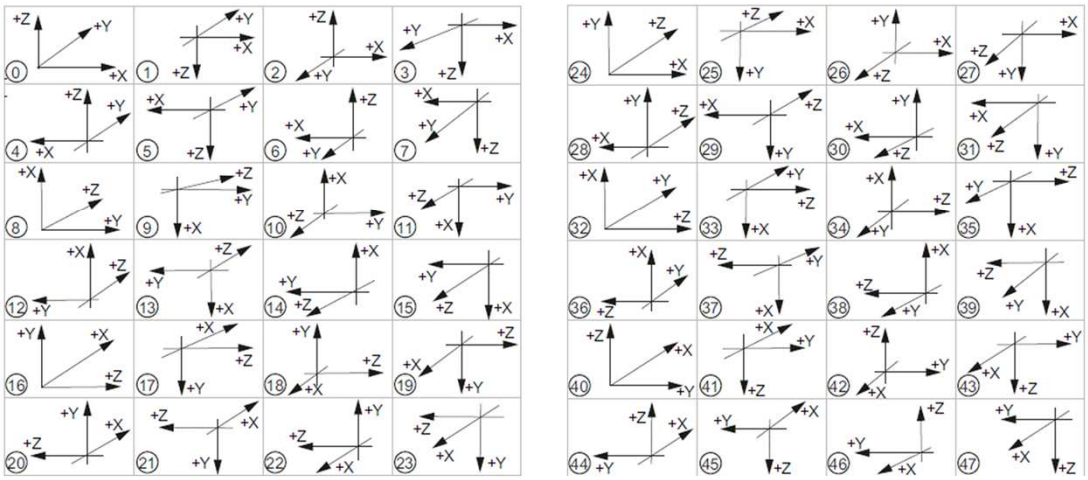
Set the turning or milling technology and the corresponding coordinate system in the channel using the channel-specific machine data specified above.

MD 52000

This MD is used to adapt the coordinate system of the user interface to that of the machine. In the user interface, all help screens, sequence graphics, simulations, and input fields with circle direction indication change automatically according to the selected position.
Also note MD 52210 $MCS_FUNCTION_MASK_DISP, bit 1.



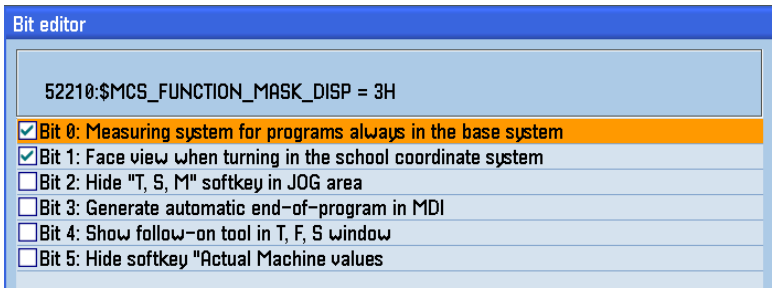
Coordinate system orientation MD 52000



Course NC-84SL-SIP Page 13 - 31 SITRAIN © Siemens AG 2017

MD 52000

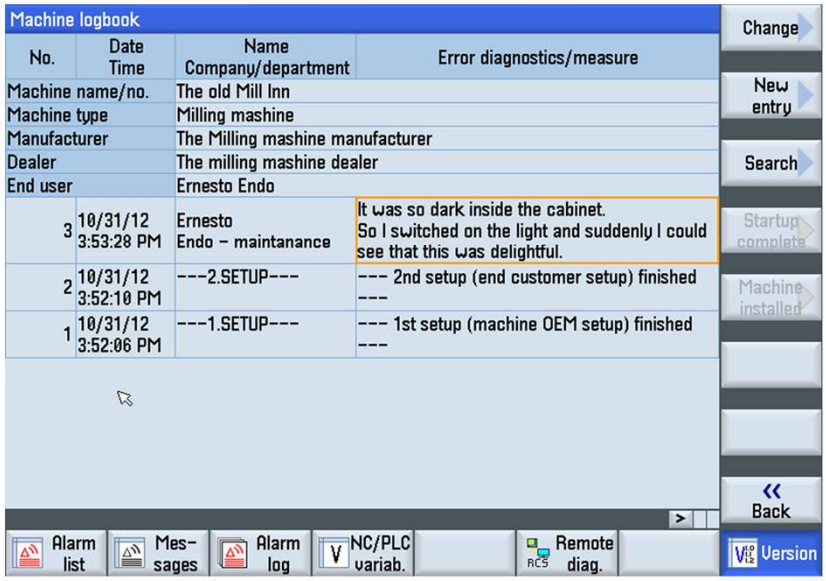
This MD is used to adapt the coordinate system of the user interface to that of the machine. In the user interface, all help screens, sequence graphics, simulations, and input fields with circle direction indication change automatically according to the selected position.
Also note MD 52210 $MCS_FUNCTION_MASK_DISP, bit 1.



NOTE

There are other machine data which may modify the display on the machine.
Among other things consider the MD from MD 52206 ff.

EUNA: Electronic logbook



 SITRAIN © Siemens AG 2017

What is EUNA? The **E**nd **U**ser **N**otification **A**greement (**EUNA**) contains information about the final destination of the machine
and its history (service work performed, replacement of modules, etc.).

What is EUNA for? One of the aims of the EUNA is to optimize spare part stocking for the regional Siemens service centers.

What does EUNA contain?

- Manufacturer information on the OEM
- Dealer information for machine delivery and installation
- End user information for machine delivery and installation
- First commissioning - End of commissioning phase with manufacturer
- Second commissioning - Machine installed - End of commissioning phase with end customer
- All service visits
- Control components

Where can I find the EUNA data?

The result of the data import or data entry in the

Version -> Logbook menu is a file called **'<Mach.name>…config.xml'**.

You will find it in the following directory:

PCU: **System hard disk**
C:\ProgramData\Siemens\sinumerik\user\sinumerik\hmi\data\version

NCU: **System CF card\user\sinumerik\hmi\data\version**

How is the data transmitted to Siemens?

The address of the online registration website is:

www.siemens.com/SINUMERIK/register

Exercises



SITRAIN © Siemens AG 2017

Notes

Exercise 1: Softkey "settings" with password only

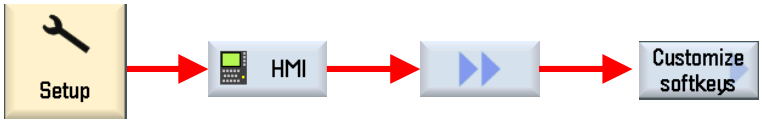
Task



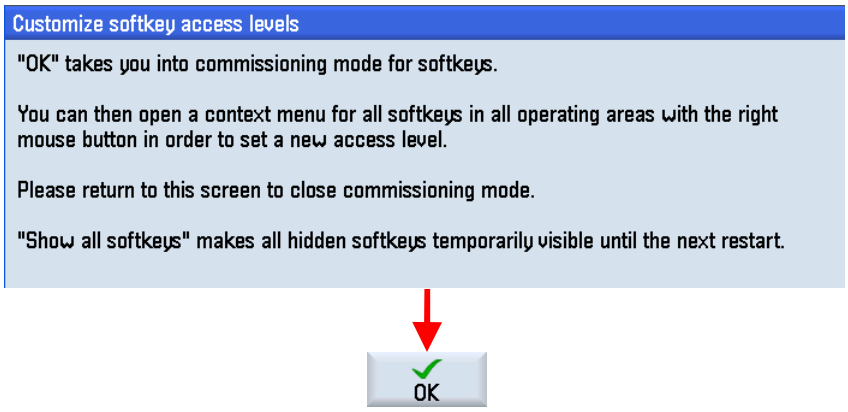
The "Settings" softkey is located in the extension of the Machine operating area. Usually, this can be accessed by all users at and above access level "7", i.e. everybody. This is to be changed so that the softkey can only be accessed at and above level "3". This requires the "CUSTOMER" password or a password with a higher value.

How it is done

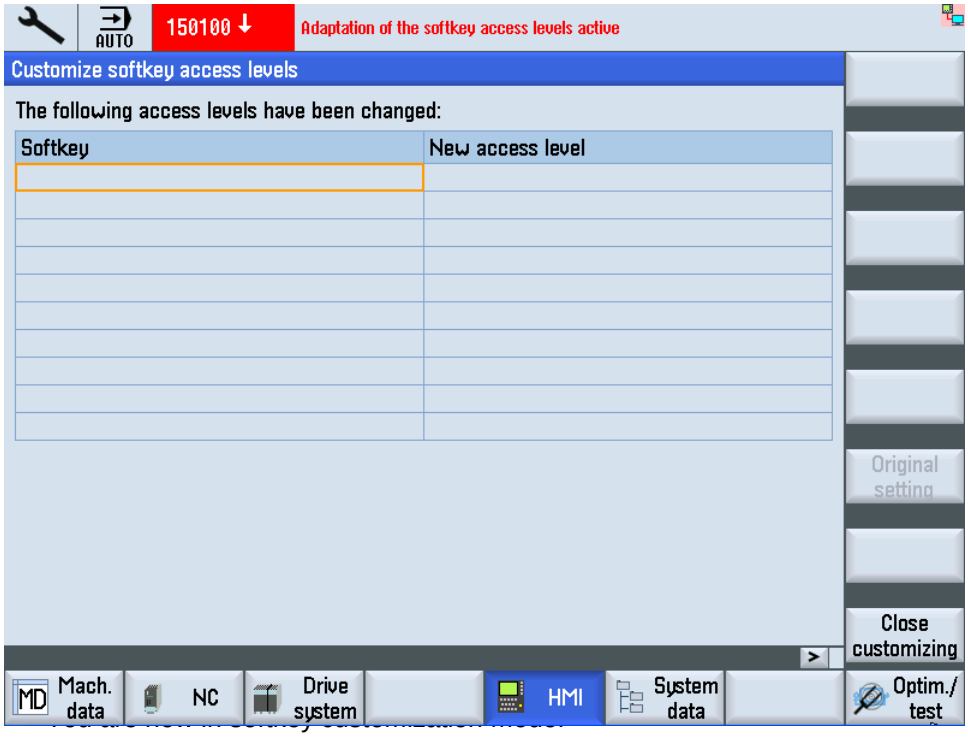
Switch to the Setup operating area. You will find the softkey for customizing the softkeys under "HMI" in the extension.



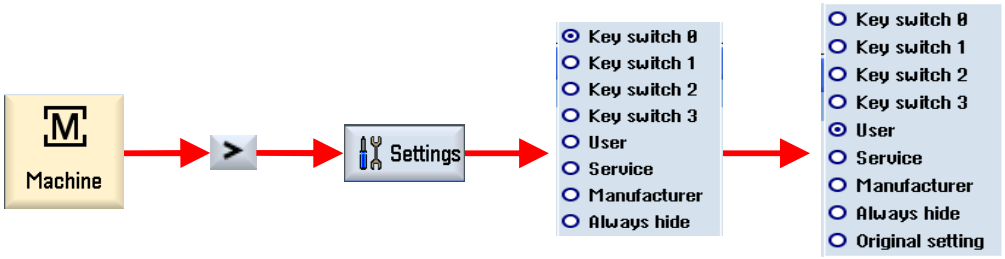
Press the softkey to call up the function for customizing the softkeys.



Select "OK" to start the function once you have read the text displayed on the screen carefully. You all see message no. 150100; this tells you that softkey customization is active.

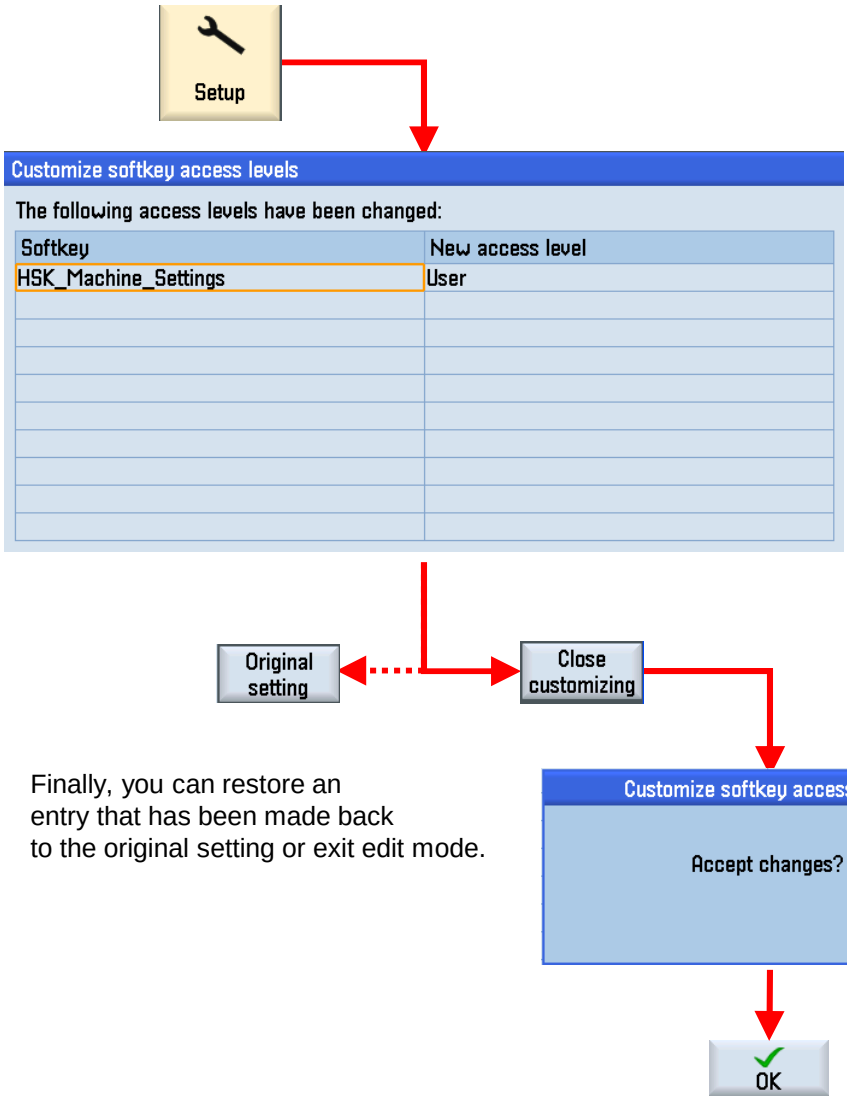


Next, go to the Machine operating area and right-click the Settings softkey. In the window that is displayed, select the required access level for this softkey.



Note that "Original setting" is also displayed for selection here.

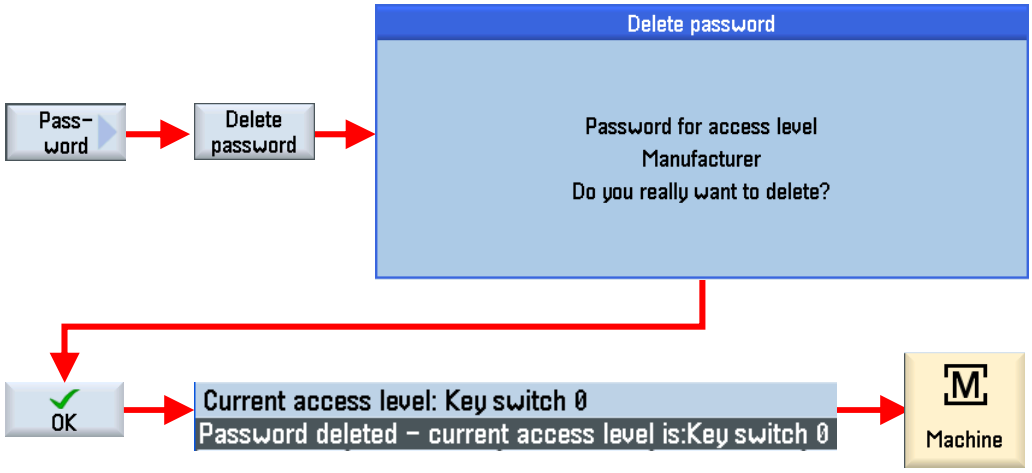
Complete the reconfiguration by switching back to the configuration menu.



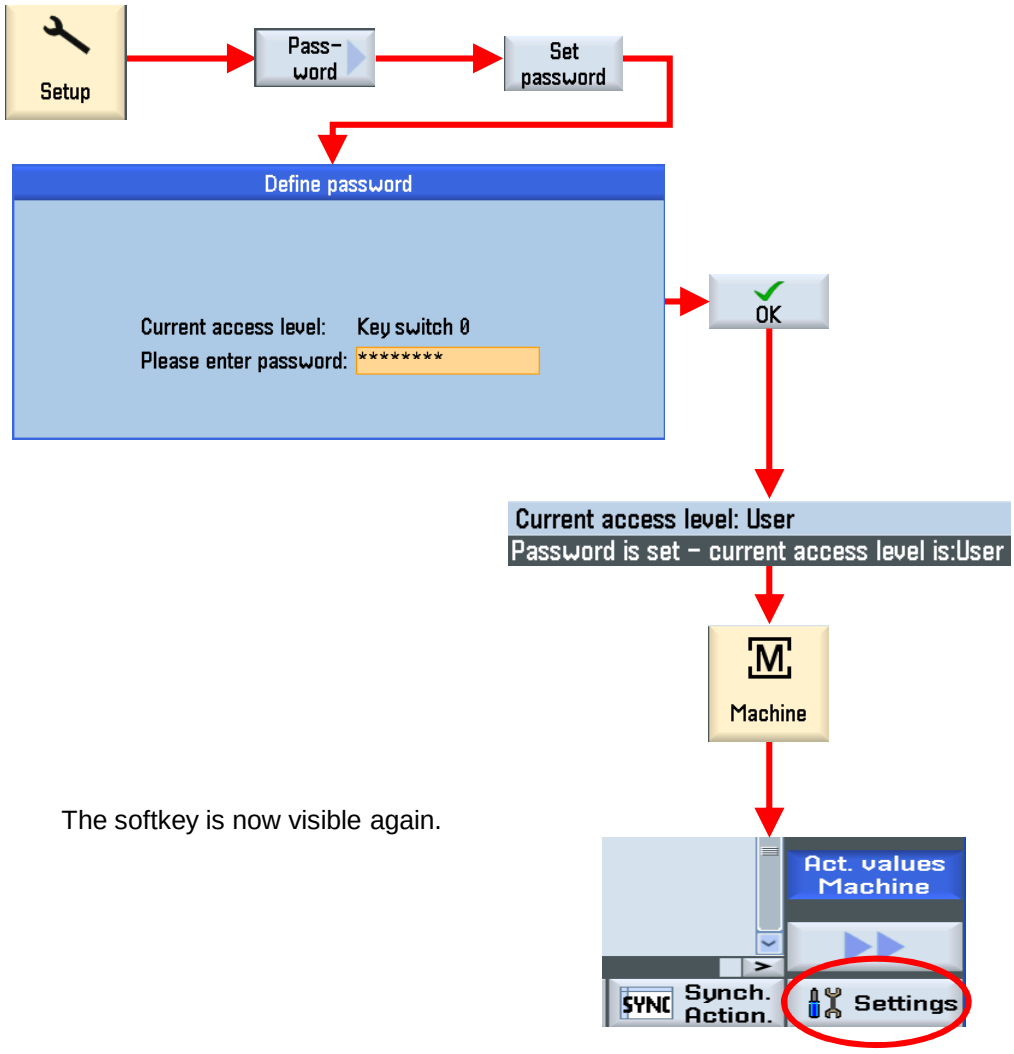
Finally, you can restore an entry that has been made back to the original setting or exit edit mode.

Function test

Delete the current password. Check whether the softkey is displayed.
Then set the "CUSTOMER" password and check whether the softkey appears again now.



The softkey should no longer be visible. Next, set the password for the user.



The softkey is now visible again.

Exercise 2: Set axis display, teach in, and program test.

Task

In this exercise, you will work with a number of items of machine data that affect display and operation.

1. Only axes X and Y are to be displayed in large font on the screen.
2. The "teach in" function is to be activated.
3. The program test function is not to be available.

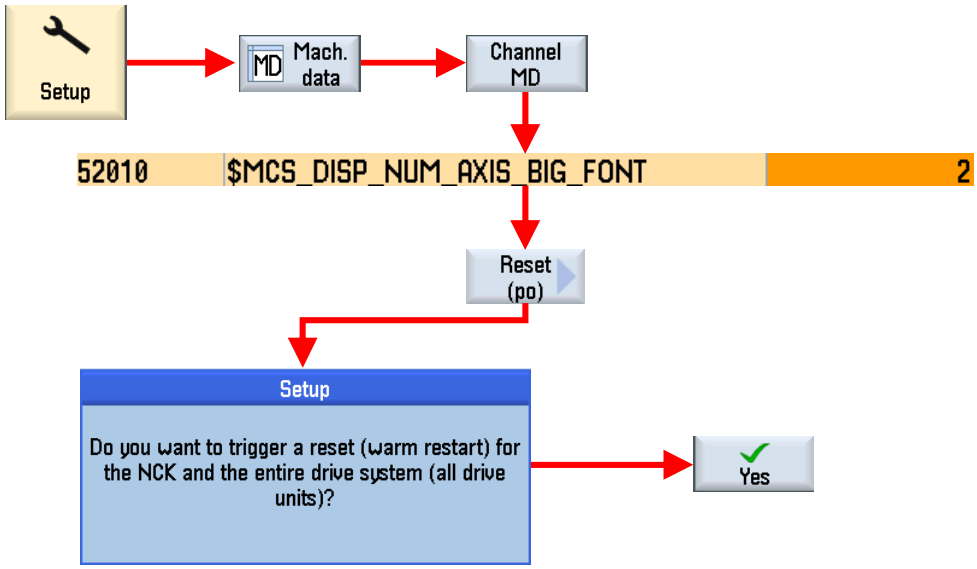
How it is done

Switch to the Setup operating area.

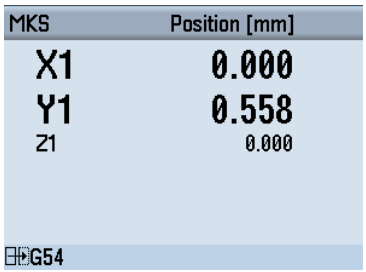
1. Set large-font display for axes X and Y for channel 1. All others should be displayed in small font.

MD 52010 $MCS_DISP_NUM_AXIS_BIG_FONT = 2

Number of actual values in large font (default)



Function test:
The number of axes displayed in large font matches the value set in the MD.



2. The "teach-in" function is to be activated.
To do this, set the following machine data.

Softkey

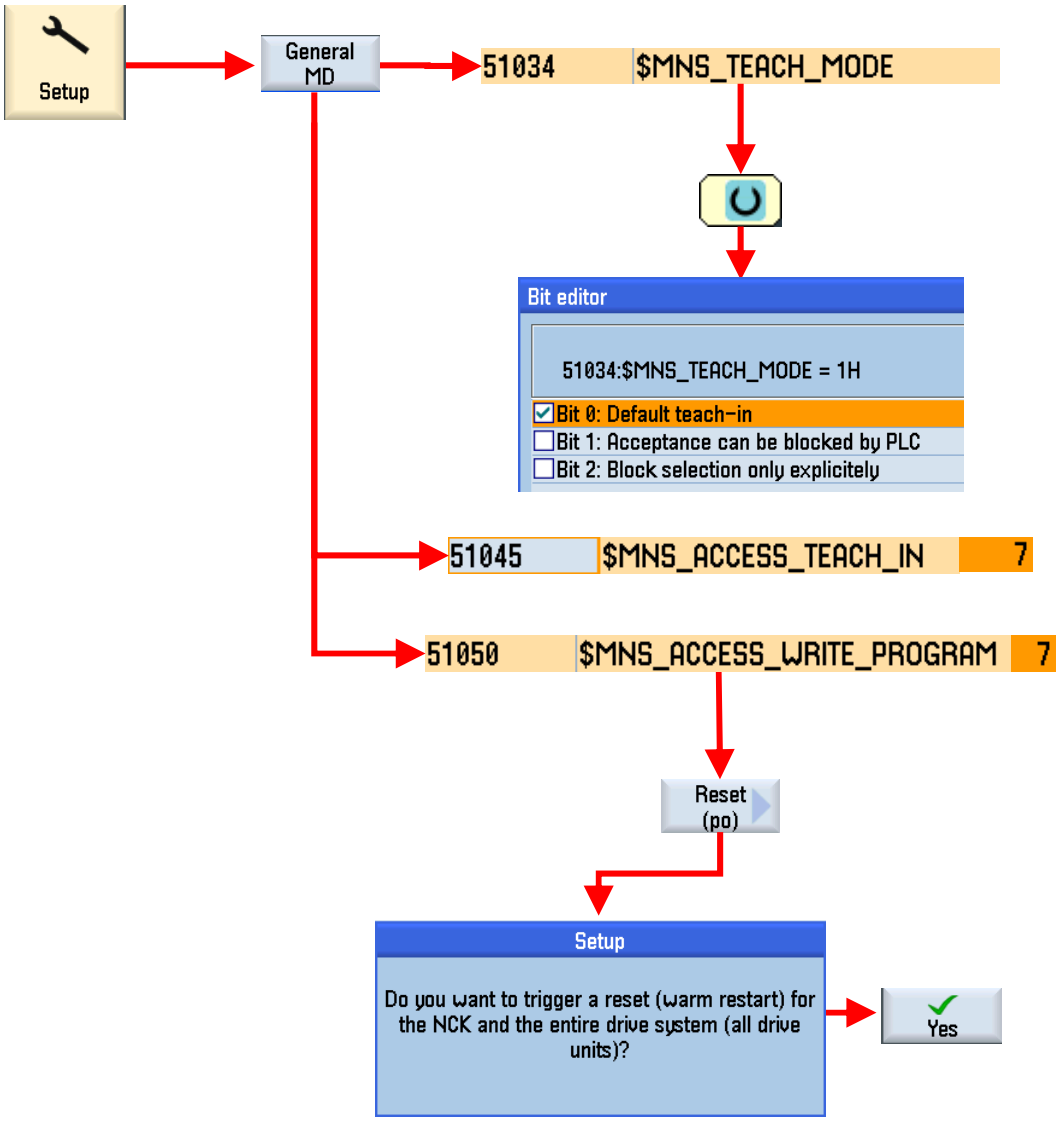
MD 51034 $MNS_TEACH_MODE bit 0=1 and bit 1=0

Bit 0 = 1	The softkey "Teach prog." is displayed. The approached position is taken into the program with the "Accept" softkey.
Bit 1 = 0	Acceptance of the teach-in block is blocked by the PLC.

Access rights

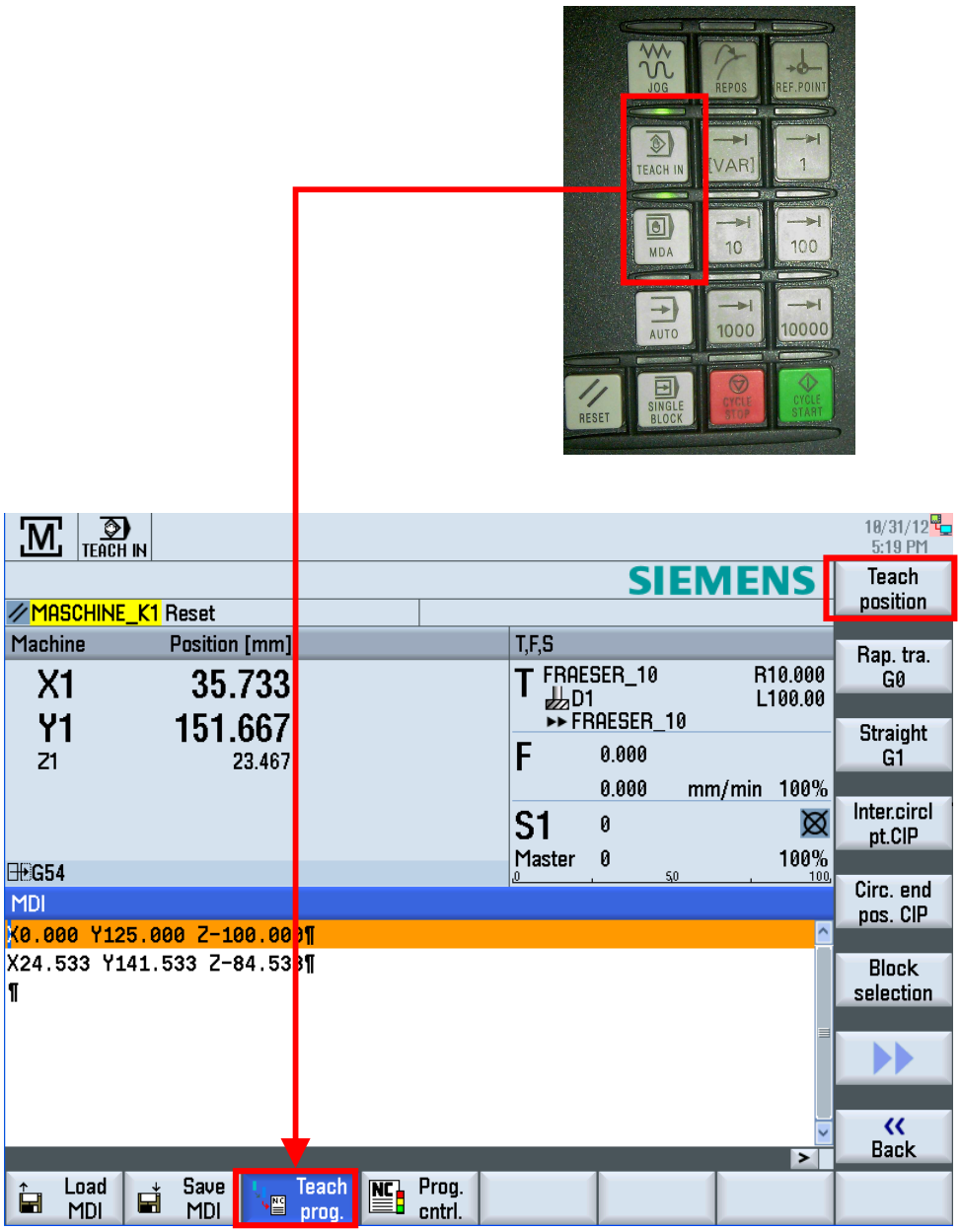
MD 51045 $MNS_ACCESS_TEACH_IN = 7 protection level TEACH IN.

MD 51050 $MNS_ACCESS_WRITE_PROGRAM = 7 Write part program protection level.



Function test

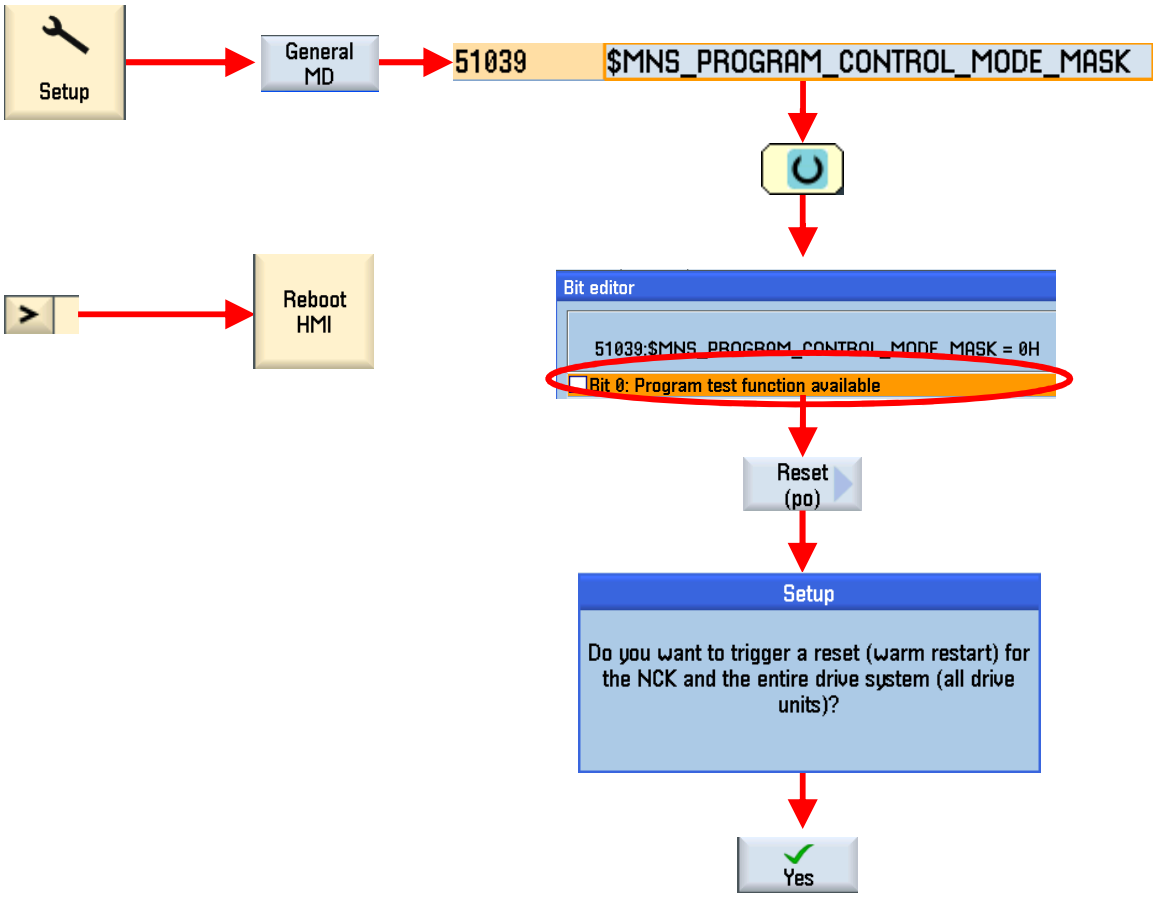
1. Switch to "AUTO" or "MDA" mode.
2. Select "TEACH IN" submode on the machine control panel.
3. Press the "Teach prog." softkey.
4. Test the functionality.



3. The program test function should **not** be available.

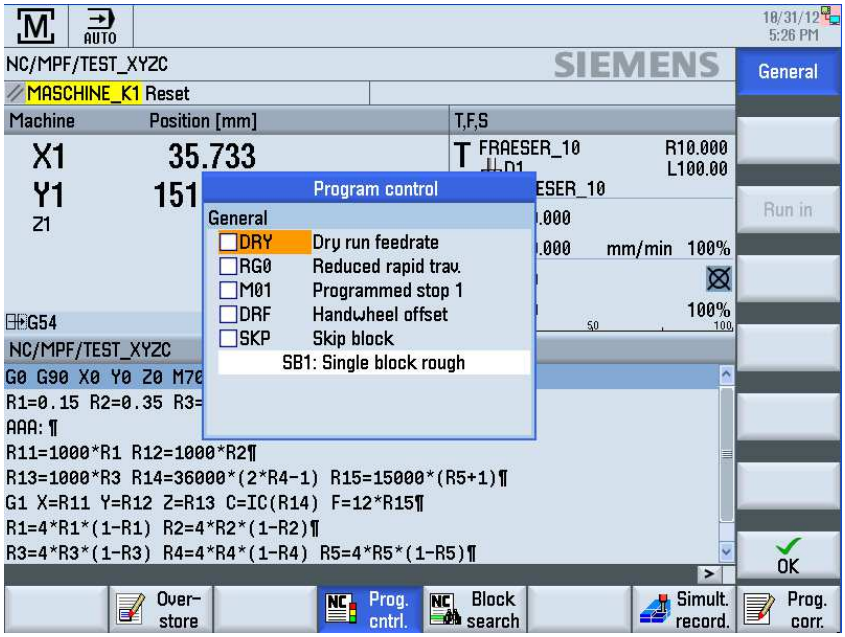
Deactivating the program test

MD 51039 $MNS_PROGRAM_CONTROL_MODE_MASK Bit 0 = 0 program test function not available



Function test

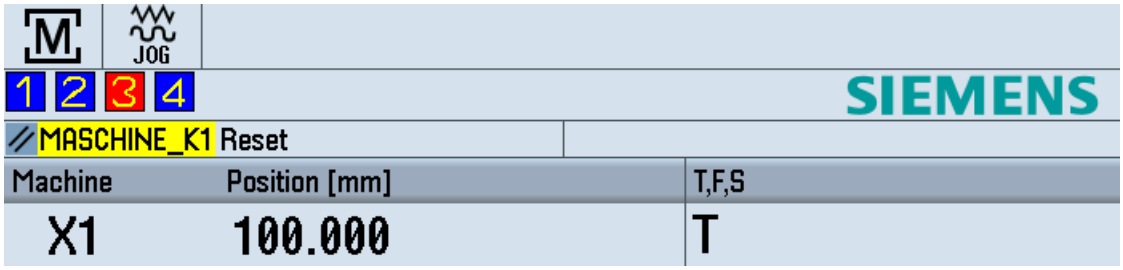
The program test should no longer be displayed in the program control window.



Exercise 3: OEM status display

Task

A display of 8 user icons is to be created. Blue number icons are to hide red user icons based on their weighting. (The icons for the resolution of OP012 are located in directory kurs_expsipsk/ico).



Procedure

3 steps must be taken.

1. Edit the file **slmahdconfig.ini** and store on the control. Note whether "your" SINUMERIK Operate is running on the NCU or on the PCU.
2. The OEM icons must be copied to the correct directory on the PCU or CF card.
3. The PLC program is extended so that the icons can be displayed via a digital input.

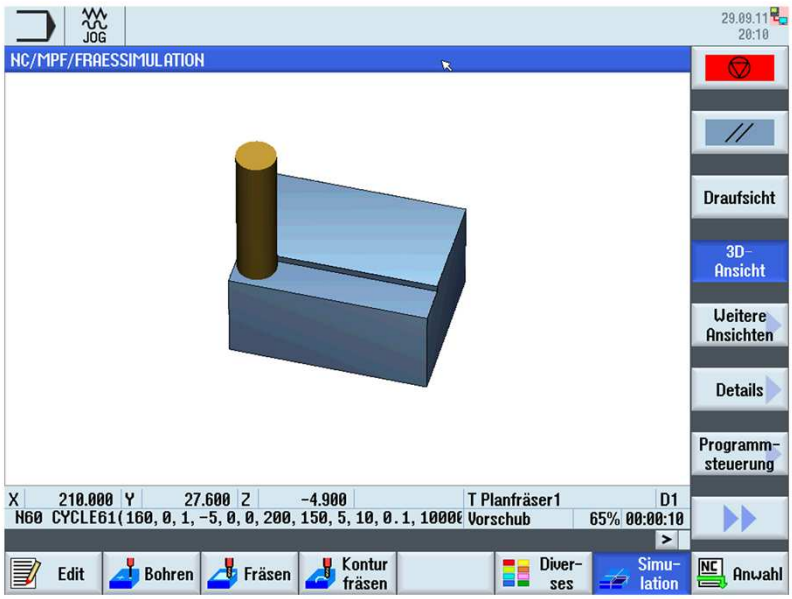
slmahdconfig.ini

```
[UserIcons]
 USER_ICON_BASE=DB19.DBB80  ; set the start byte of the plc double word that displays the icons
 UI_0 = icon1.png,1              ; show icon1.png on position 1 if bit 0 (DB19.DBX83.0) is set
 UI_1 = icon2.png,2              ; show icon2.png on position 2 if bit 1 (DB19.DBX83.1) is set
 UI_2 = icon3.png,3              ; show icon3.png on position 3 if bit 2 (DB19.DBX83.2) is set
 UI_3 = icon4.png,4              ; show icon4.png on position 4 if bit 3 (DB19.DBX83.3) is set

 UI_16 = icon1b.png,1            ; show icon1b.png on position 1 if bit 16 (DB19.DBX81.0) is set
 UI_17 = icon2b.png,2            ; show icon2b.png on position 2 if bit 17 (DB19.DBX81.1) is set
 UI_18 = icon3b.png,3            ; show icon3b.png on position 3 if bit 18 (DB19.DBX81.2) is set
 UI_19 = icon4b.png,4            ; show icon4b.png on position 4 if bit 19 (DB19.DBX81.3} is set
```

Exercises
4: Set the machine data for the simulation

SIEMENS



Course NC-84SL-SIP Page 13 - 42 SITRAIN © Siemens AG 2017

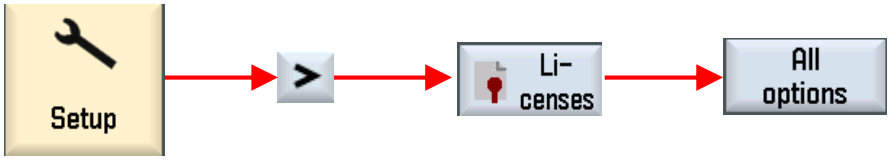
Objective

To set the necessary machine data so that the program simulation can be used for a milling machine.

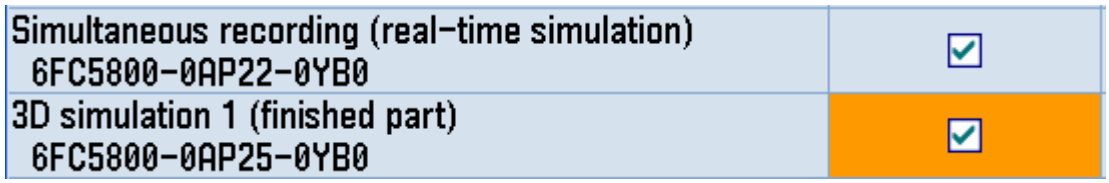
Sequence

Proceed as follows:

1. In the Setup operating area, select the menu extension followed by the Licenses > All options function.



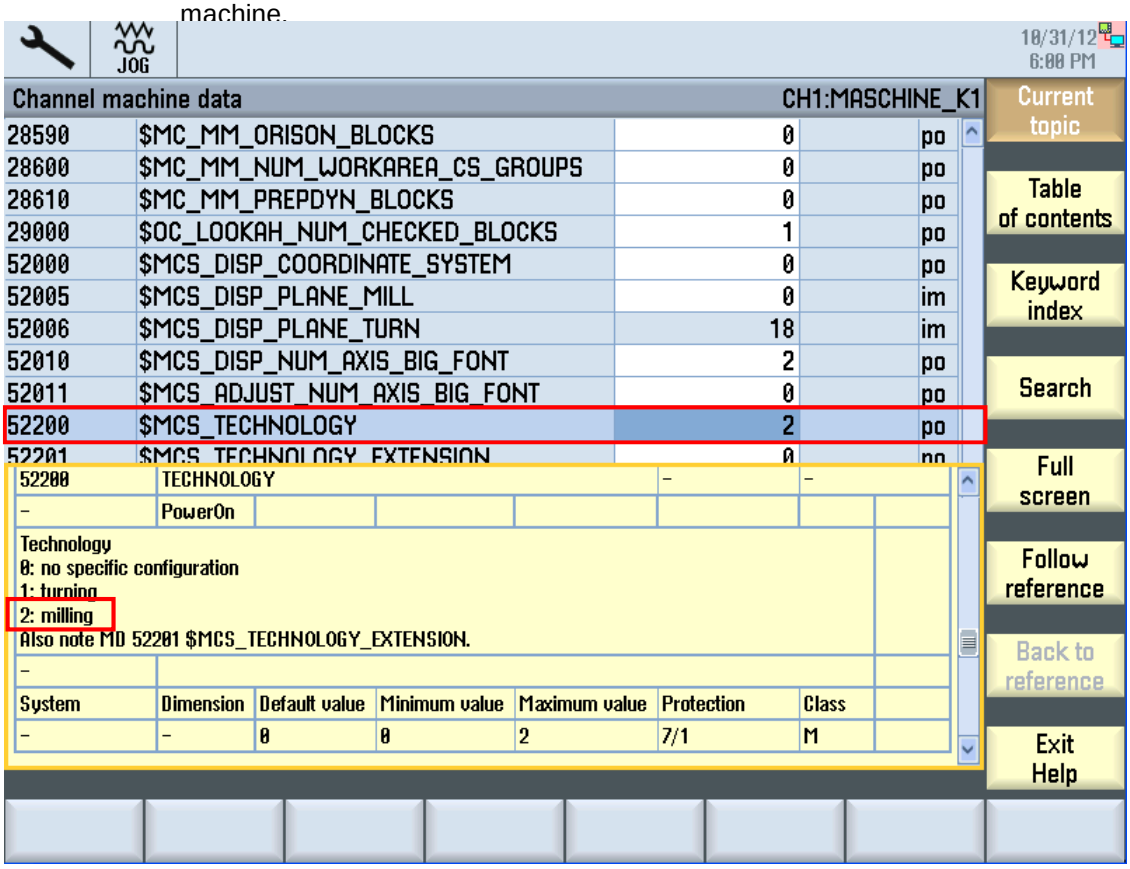
2. Select the options for simultaneous recording and simulation (simultaneous recording is not needed specifically here).





3. Once you have made the selections, perform a power ON reset.

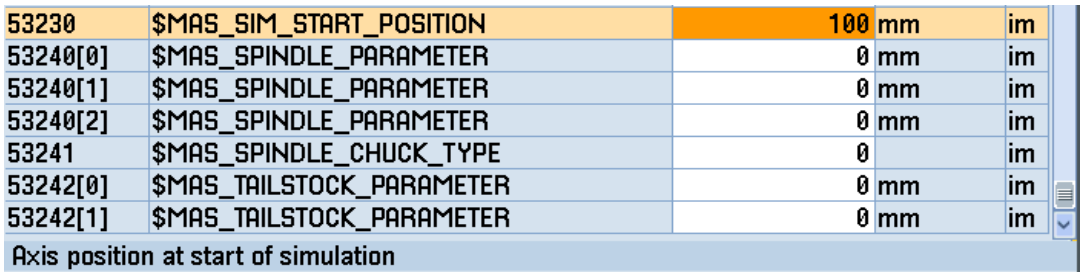
4. Next, set the technology and the extended technology for the machine.



5. At this point, MD 24030 is set to the value "1" for simulation.
The SZS coordinate system (settable zero system SZS, holding position of the active tool) is used for the simulation to avoid jumps in the display when various offsets are factored in.

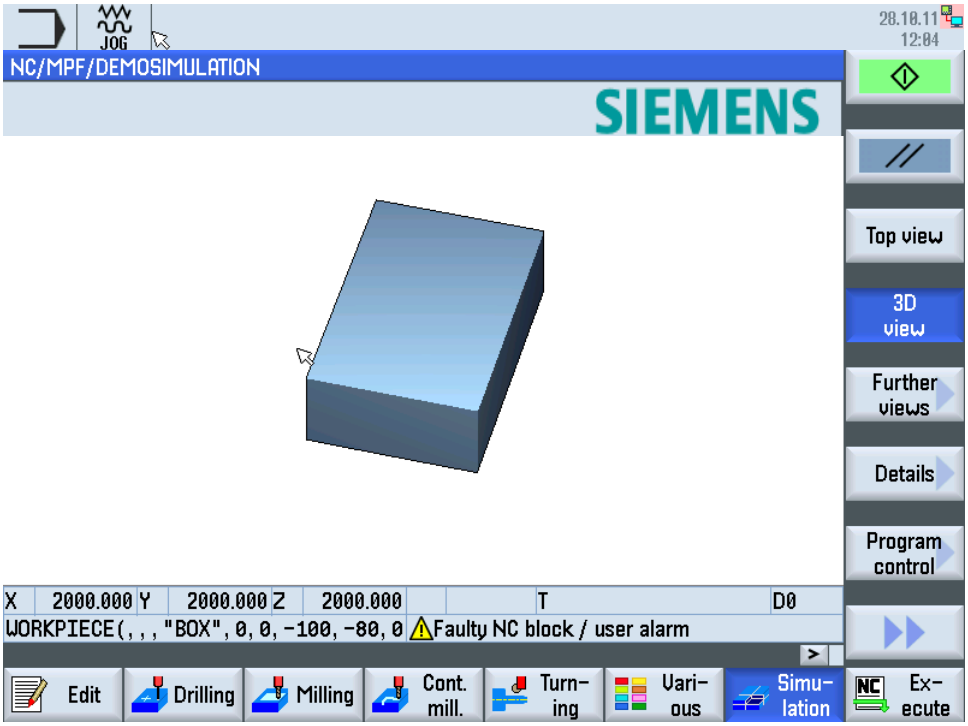


In the context of simulation, you should also set a suitable starting position for the axis in the simulation. Monitor the axes as they move into the simulation range (MD 53230).

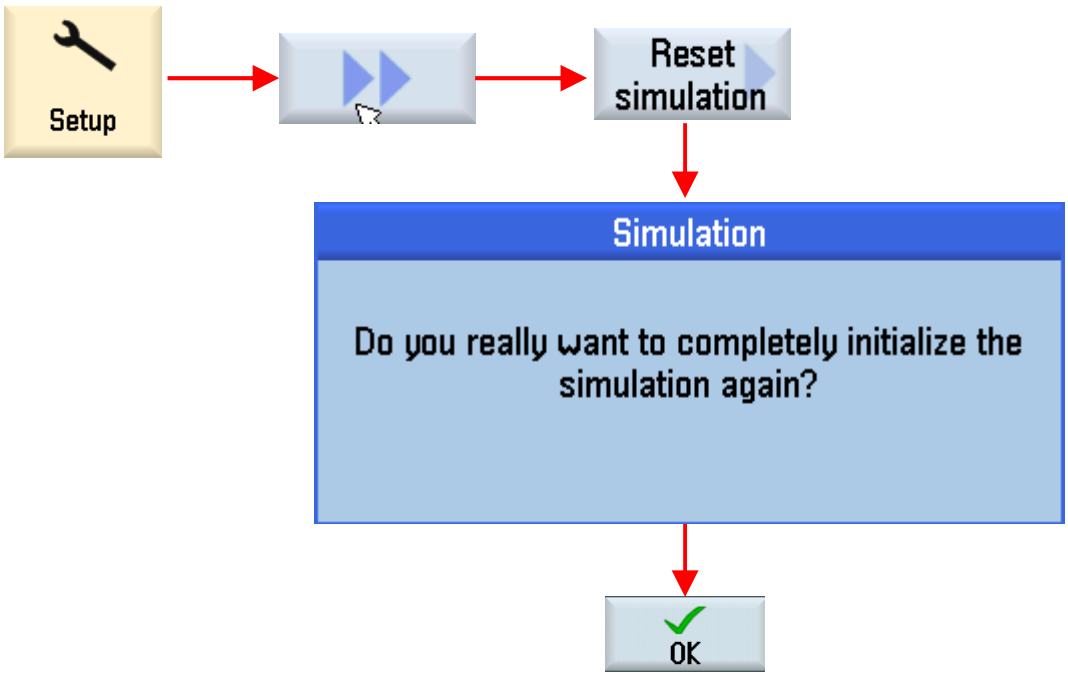




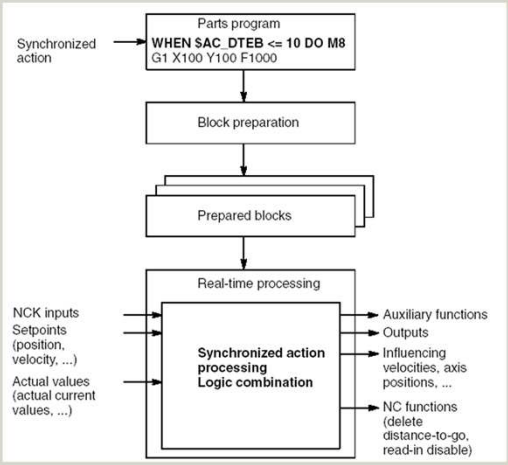
6. Once you have made the selections, perform a power ON reset.
Simulation itself does not get underway yet, as the tools are missing.



If you are unable to start simulation, you will have to reload the simulation data. You can do this by performing a memory reset of the simulation data, for example. This data is reloaded automatically when simulation is subsequently restarted.



SIEMENS



Chapter 14

Synchronized actions

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
What are synchronized actions? 3
Components of a synchronized action 4
Structure of a synchronized action 5
Validity, identification number 6
Frequency 7
G code for condition/action 8
Condition 9
Action/technology cycle 10
Actions in synchronized actions 11
Technology cycle 12
SYNFCT evaluation function 13
Function definition and evaluation function 14
Example evaluation function SYNFCT 15
Special variables for synchronized actions 16
Coordination of synchronized actions by NC and PLC 19
Protected synchronized actions 20
Configuration of the number of synchronized actions 21
Event-controlled program call (K1) 22
Signal flow digital inputs 23
Signal flow digital outputs 24
System variable for fast I/Os 25
Number of fast I/Os 26
Exercise: Tracing the function of a synchronized action 27
Additional exercises 29
Solutions 31

Learning objectives

- You will be familiar with the functionality of synchronized actions
- You will be familiar with the use of synchronized actions
- You will be familiar with the SYNFCT evaluation function
- You will be familiar with event-controlled program calls



SITRAIN © Siemens AG 2017

Notes

What are synchronized actions?

Example: Activate coolant 10 mm before the end of the block:

```
WHEN $AC_DTEW <= 10  DO  M8
G1 X100 F1000
```



Course NC-84SL-SIP Page 14 - 3 SITRAIN © Siemens AG 2017

Definition

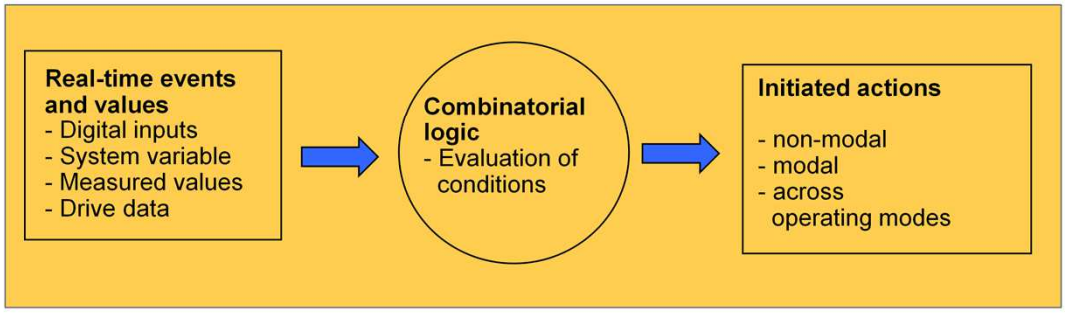
Synchronized actions (also called "motion-synchronous actions") are instructions programmed by the user that are evaluated in the interpolation cycle of the NCK in synchronization with execution of the part program.
If the condition programmed in the synchronized action is fulfilled or if none is specified, actions assigned to the instruction are activated in synchronism with the remainder of the part program run.

Applications

Synchronized actions are frequently used for:

- Outputting auxiliary functions to the PLC
- Reading and writing real-time variables
- Positioning axes/spindles
- Activating synchronous procedures, such as:
  - Read-in disable
  - Delete distance-to-go
  - End of preprocessing stop
- Activating technology cycles
- Online calculation of function values
- Online tool offsets
- Activating/deactivating couplings/coupled motion
- Taking measurements
- Enabling/disabling synchronized actions

Components of a synchronized action



 SITRAIN © Siemens AG 2017

Notes

Structure of a synchronized action

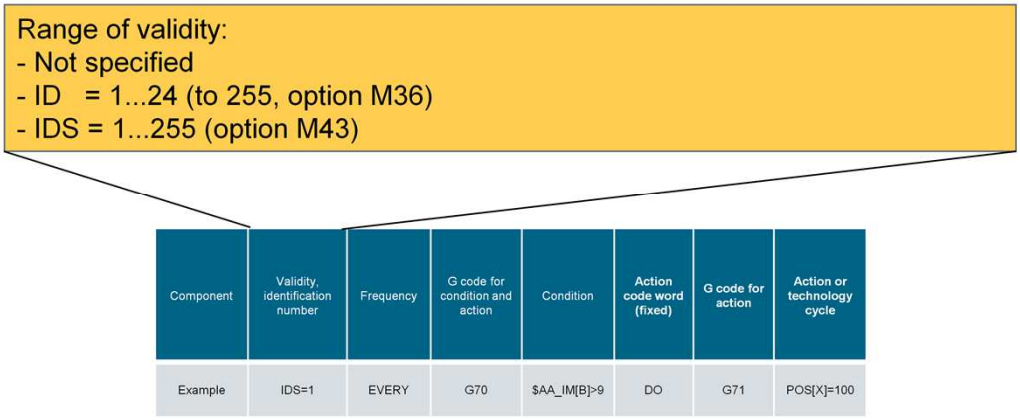
Component	Validity, identification number	Frequency	G code for condition and action	Condition	Action code word (fixed)	G code for action	Action or technology cycle
Example	IDS=1	EVERY	G70	$AA_IM[B]>9	DO	G71	POS[X]=100

SITRAIN © Siemens AG 2017

Components of a synchronized action

- Validity with or without identification number
- Frequency
- G code for condition and action
- Condition
- G code for actions
- Action(s) / technology cycle

Validity, identification number



SITRAIN © Siemens AG 2017

Not specified

Synchronized actions that have no specified validity have a non-modal action, i.e. they apply only to the next block.

Non-modal synchronized actions are operative only in AUTOMATIC mode.

Non-modal synchronized actions are active modally for all preprocessing stop blocks (incl. implicitly generated ones) and for implicitly generated intermediate blocks.

ID

Synchronized actions with validity identifier "ID" have a modal effect until the end of the part program.

IDS

Static synchronized actions (option M43) that are programmed with keyword "IDS" are active in all operating modes. They are also referred to as static synchronized actions.

Canceling/disabling

Modal and static synchronized actions are active until:

- another synchronized action with the same identification number is programmed
- they are canceled with CANCEL(n) or RESET(n)
- they are disabled with LOCK(n)

Order of execution

Within an interpolation cycle, modal synchronized action instructions are processed in order of their ID number (i.e. block with ID number 1 before block with ID number 2, etc.). Once the modal synchronized action instructions have been executed, non-modal synchronized action instructions are processed in the order in which they are programmed.

Notice

ID and IDS access the same number range.

Frequency

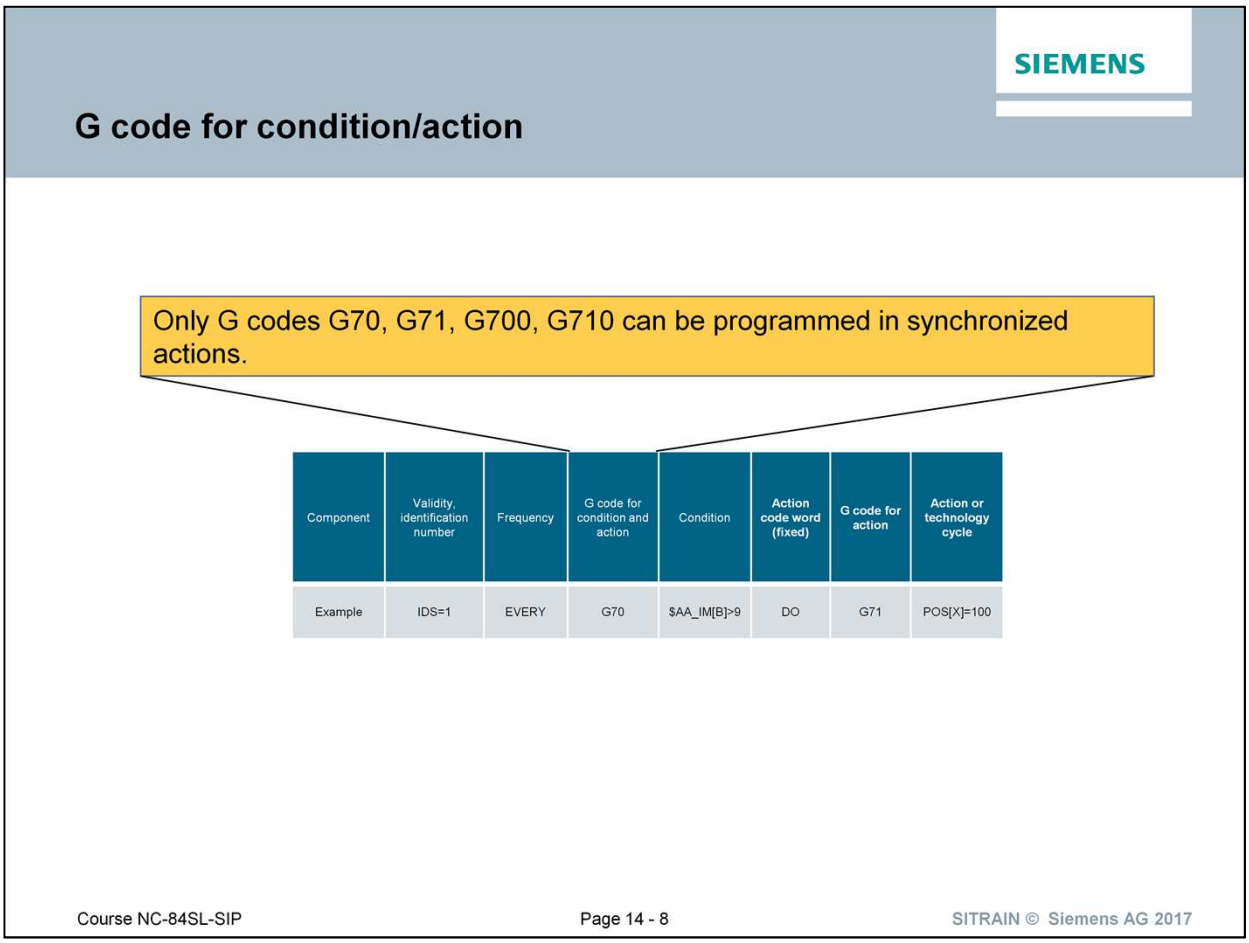
Frequency	Meaning
"Not specified"	If no frequency is specified, the action is executed cyclically in every interpolation cycle.
WHENEVER	If the condition is fulfilled, the action is executed cyclically in every interpolation cycle.
FROM	After the condition has been fulfilled once, the action is executed cyclically in every interpolation clock cycle for as long as the synchronized action is active.
WHEN	If the condition is fulfilled, the action is executed once and then the condition is no longer checked.
EVERY	On every state change of the condition from FALSE to TRUE (rising edge), the action is executed once.

Component	Validity, identification number	Frequency	G code for condition and action	Condition	Action code word (fixed)	G code for action	Action or technology cycle
Example	IDS=1	EVERY	G70	$AA_IM[B]>9	DO	G71	POS[X]=100

Course NC-84SL-SIP Page 14 - 7 SITRAIN © Siemens AG 2017

Frequency Keywords (see table) are programmed to indicate how often the condition following them must be queried and the associated action must be executed if the condition is fulfilled. These keywords are an integral component of the synchronized action condition.

Notice The duration of execution also depends on ID/IDS.



G code for condition and action

G codes can be programmed in synchronized actions.

This allows defined settings to exist for the evaluation of the condition and the action/technology cycle to be executed, irrespective of the current part program status. It is necessary to separate the synchronized actions from the program environment because synchronized actions are required to execute their actions at any time from a defined initial state as a result of fulfilled trigger conditions.

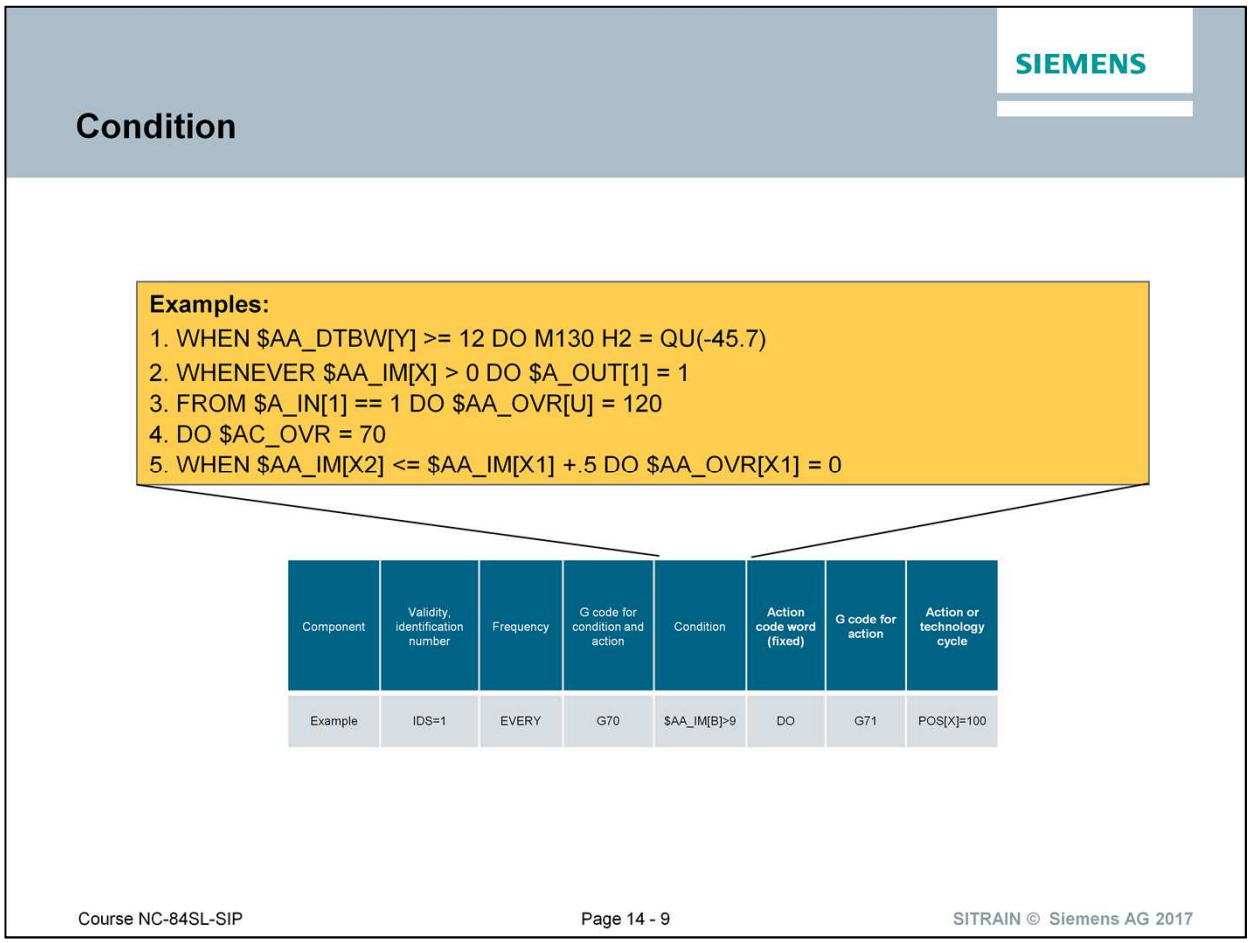
Application cases:

Definition of the systems of measurement for condition evaluation and action through G codes G70, G71, G700, G710.

Only the 4 listed G codes are permissible here.

Only one G code of the G code group may be programmed for each part of the condition.

G70 Dimensions in inch
G71 Dimensions in mm
G700 Dimensions in inch and inch/min
G710 Dimensions in mm and mm/min



Examples

1. Additional function output after 12 mm distance from block start (center-point path distance) Execute the action once when the condition is fulfilled.
2. When the axis actual value of X is positive, set the digital output 1 to 1 mV. Perform the action until the condition is fulfilled.
3. When digital input 1 is 1, set the axial override to 120%. Perform the action cyclically once the condition is fulfilled.
4. Set the path override to 70%. Perform the action cyclically.
5. Stop if the safety margin of 0.5 mm between X1 and X2 has been exceeded.

Conditions

Execution of actions/technology cycles can be made dependent upon a condition (logical expression).

The condition is checked in the interpolation cycle. If no condition is programmed, the action is performed once in every IPO cycle.

The conditions supported are:

- Compare a variable with an expression calculated in the block search or the comparison of two variables.

Examples:

WHENEVER **$AA_IM[X] > 10.5*SIN(45)** DO .... or

WHENEVER **$AA_IM[X] > $$AA_IM[X1]** DO ...

- Compare with Boolean logic between variables of the **same type**: Boolean operators of the NC language may be used for this purpose:

  NOT, AND, OR, XOR, B_OR, B_AND, B_XOR, B_NOT

Action/technology cycle

Each synchronized action contains up to 16 actions or one technology cycle after the DO
Actions contained in a synchronized action are activated in the same interpolation cycle if the relevant condition is fulfilled.

Component	Validity, identification number	Frequency	G code for condition and action	Condition	Action code word (fixed)	G code for action	Action or technology cycle
Example	IDS=1	EVERY	G70	$AA_IM[B]>9	DO	G71	POS[X]=100

SITRAIN © Siemens AG 2017

Actions

Every synchronized action contains one or more programmed actions or one technology cycle. These are executed when the appropriate condition is fulfilled. If several actions are programmed in one synchronized action, they are executed within the same interpolation cycle.

Example:
WHEN $AA_IM[Y] >= 35.7 DO **M135 $A_OUT[1]=1**

If the actual value of the Y axis is greater than or equal to 35.7, M135 is output to the PLC and at the same time output 1 is set.

Program/technology cycle

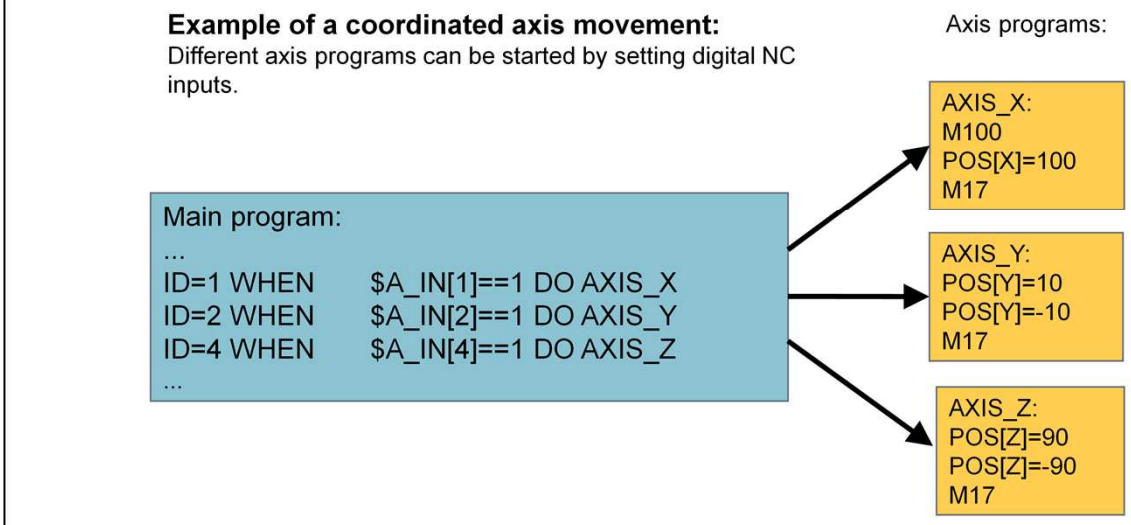
A program (name) can also be specified as an action.

Actions in synchronized actions

Polynomial evaluation (SYNFCT)	Output of M, S and H auxiliary functions to PLC
Online tool offset (FTOC)	Reading and writing of system variables, e.g. $AC_OVR, $AC_VC…
Delete distance-to-go (DELDTG)	Setting the measuring system (G70, G71, G700, G710)
Set actual value (PRESETON)	Query of position in specified reference range (POSRANGE)
Measurement (MEAWA, MEAC)	Starting/stopping axes/spindles and setting the axial feedrate
Call of technology cycles	Axis replacement (GET, RELEASE, AXTOCHAN)
Channel synchronization (SETM, CLEARM)	Withdrawing the enable for the axis container rotation (AXCTSWEC)
Cancel preprocessing stop (STOPREOF)	Couplings (CP..., LEAD..., TRAIL..., CTAB...)
Traversing command axes (POS)	Travel to fixed stop (FXS, FXST, FXSW, FOCON, FOCOF, FOC)
Programmed read-in disable (RDISABLE)	Coordination between synchronized actions (LOCK, UNLOCK, RESET)
User-specific errors (SETAL)	

SITRAIN © Siemens AG 2017

Technology cycle



SITRAIN © Siemens AG 2017

Definition

Technology cycles are a sequence of actions that are executed sequentially in the interpolation cycle. From the user's point of view, these programs are subprograms without parameters.
All actions are permissible that can be programmed individually in synchronized actions.

Several technology cycles or actions can be processed simultaneously in the same channel. These cycles and actions are processed in parallel in the channel in one interpolation cycle.

Application

One possible application of technology cycles is to move each axis using a separate axis program.

Programming

A technology cycle can be activated as a function of a condition in a modal/static synchronized action.

End of program is programmed with M02/M17/M30/RET.

Search path

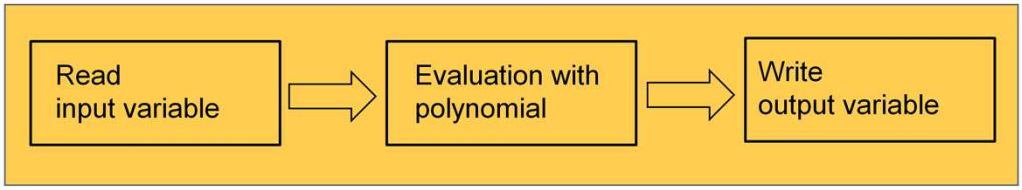
The call search path is the same as for subprograms and cycles.

Example of a call:

```
...
ID=1 EVERY $AA_IM[Y]>=10 DO AX_X      ; AX_X subroutine name

...
```

SYNFCT evaluation function



Definition of the polynomial:
FCTDEF(PolynomialNo., lower limit output, upper limit output, a0, a1, a2, a3)

Call of the evaluation function:
ID=1 DO SYNFCT(Polynomial No., input variable, output variable)

SITRAIN © Siemens AG 2017

Evaluation function

By applying an evaluation function, it is possible to read a variable, evaluate it with a polynomial and write the result to another variable in synchronism with the machining process.

It can be used, e.g. to influence the feedrate of an axis as a function of the measured current of the grinding spindle.

FCTDEF parameter

The parameters are stored in system variables:

$AC_FCTLL[n]: Lower limit,
$AC_FCTUL[n]: Upper limit,
$AC_FCT0[n]: a0-coefficient,
$AC_FCT1[n]: a1-coefficient,
$AC_FCT2[n]: a2-coefficient,
$AC_FCT3[n]: a3-coefficient,

n: Polynomial number

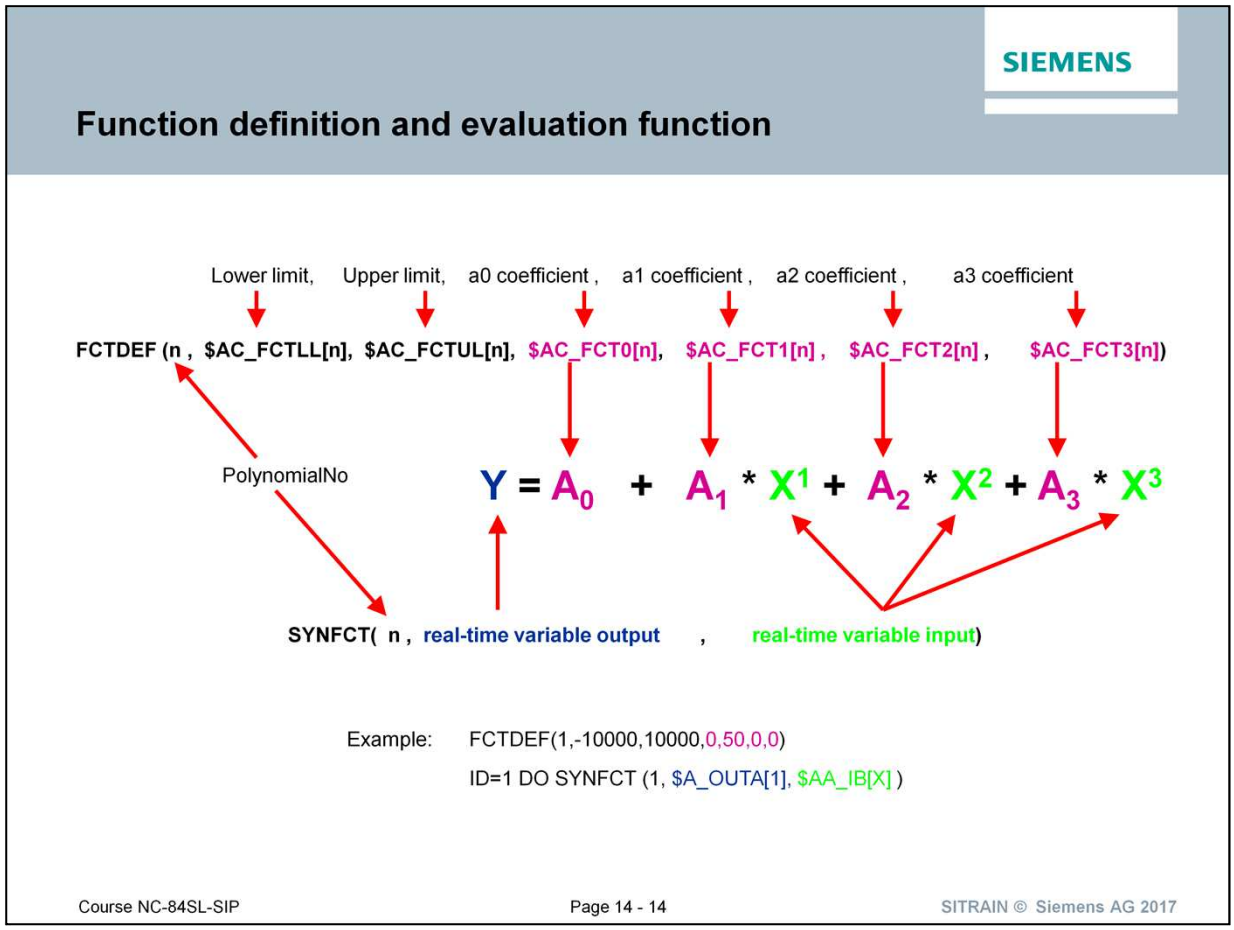
Memory requirement

This must be reserved in the SRAM in MD28252

Example

ID=1 DO **SYNFCT**(1,$AA_VC[U1], $A_INA[2])

The additive offset value of the axis U1 is calculated in each interpolation cycle from the analog input value 2 via polynomial 1



Polynomial

An evaluation function consists of a 3rd order polynomial and 4 polynomial coefficients each.

Polynomial: $y = a_0 + a_1*x^1 + a_2*x^2 + a_3*x^3$
It is programmed in the following way:

SYNFCT(...)

SYNFCT(Polynomial No., real-time variable output, real-time variable input)

The polynomial identified by 'Polynomial number' is evaluated with the value of the 'Real-time variable input'. The result is then limited by maximum and minimum limits and assigned to the 'Real-time variable output'.

Example:

```
FCTDEF(1,0,100,0,0.8,0,0) ; Polynomial 1
...
```

Synchronized action:

```
ID=1 DO SYNFCT(1,$AA_VC[U1], $A_INA[2])
```

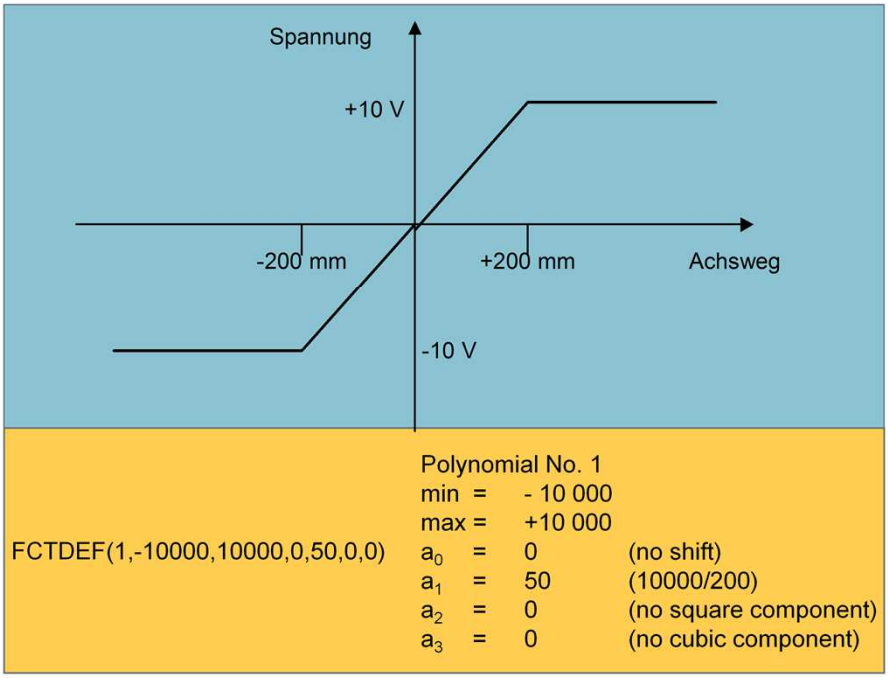
The additive offset value of the axis U1 is calculated in each interpolation cycle from the analog input value 2 via polynomial 1

Real-time variable

For the 'Real-time variable output', it is possible to select variables that are included in the machining process as follows:

- with additive influencing (e.g. feedrate)
- with multiplicative influencing (e.g. override)
- as a position offset or
- directly

Example evaluation function SYNFCT



FCTDEF(1,-10000,10000,0,50,0,0)

Polynomial No. 1

min	=	- 10 000	
max	=	+10 000	
a_0	=	0	(no shift)
a_1	=	50	(10000/200)
a_2	=	0	(no square component)
a_3	=	0	(no cubic component)

Course NC-84SL-SIP Page 14 - 15 SITRAIN © Siemens AG 2017

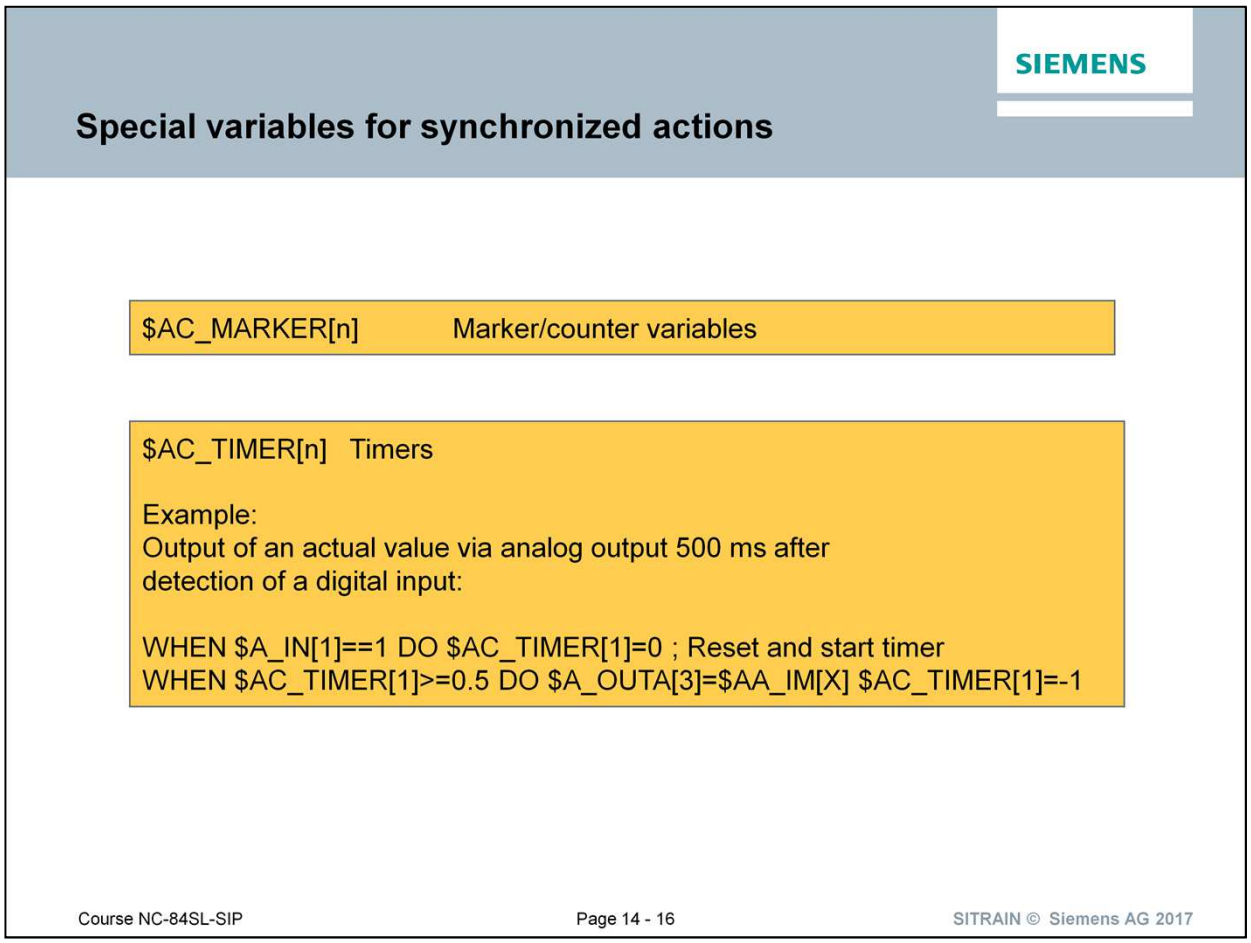
Example

Two analog voltages (for X and Y) are required for controlling the electron beam for electron beam welding, depending on the axis setpoints. The output values are passed to the PLC via FC 21 ($A_DBR[1] and $A_DBR[5].

This is to machine workpieces in the axis range of ± 200 mm respectively.

Program

```
FCTDEF(1,-10000,10000,0,50,0,0)
ID=1 DO SYNFCT (1, $AA_IB[X], $A_DBR[1])
ID=2 DO SYNFCT (1, $AA_IB[Y], $A_DBR[5])
```



Markers

The channel-specific $AC_MARKER[n] variable serves as a marker or counter of INTEGER data type. They can be read and written.

The number of markers per channel is set via machine data

MD 28256: NUM_AC_MARKER (max. 20000).

The markers are available once per channel under the same name.
They are kept in dynamic memories. The markers are set to 0 on Power ON, NC reset, and end of program.

Timers

System variable **$AC_TIMER[n]** permits actions to be started after defined periods of delay.

The number of available timer variables is defined in machine data

MD 28258: MM_NUM_AC_TIMER.

Set timer

Incrementation of a timer variable is started by means of value assignment:

$AC_TIMER[n] = start value (REAL in s)

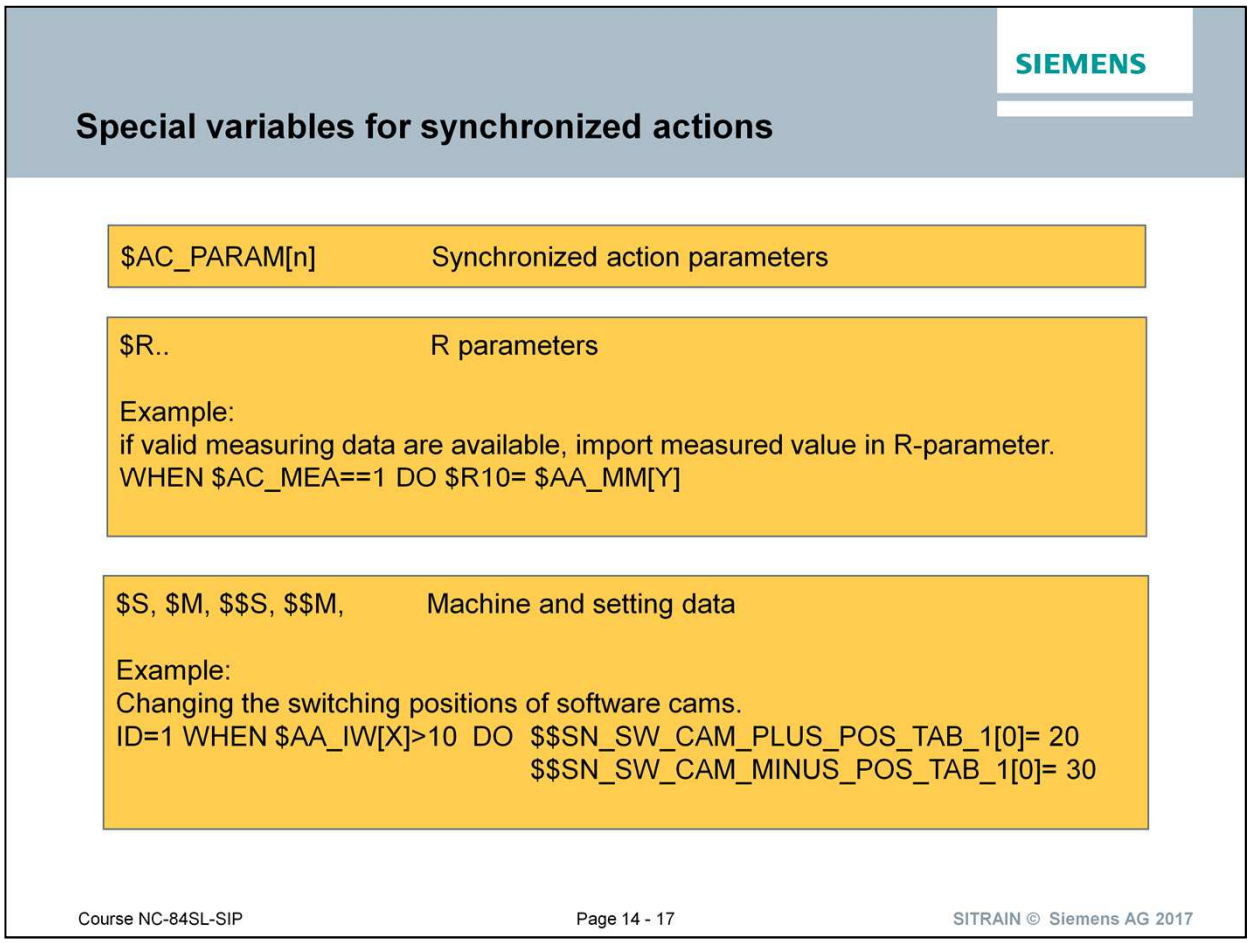
Stop timer

Incrementation of a timer variable can be stopped by assigning a negative value:

$AC_TIMER[n] = -1

Read timer

The current timer value can be read whether the timer variable is running or has been stopped. After a timer variable has been stopped by assignment of -1, the current time value remains stored and can be read.



Synchronized action parameters

The channel-specific **$AC_PARAM[n]** variables are used as a buffer in synchronized actions. They can be read and written.

The number of available AC parameter variables in each channel is defined in machine data

MD 28254: MM_NUM_AC_PARAM.

$AC_PARAM parameters are stored in the dynamic memory. The parameters are set to 0 on POWER ON, NC RESET and End of Program.

R parameters

By programming the $ sign in front of R parameters, they can also be used as real-time variables in synchronized actions.

If an R variable used in synchronized actions is used "normally" again in the part program, synchronization must be ensured with STOPRE.

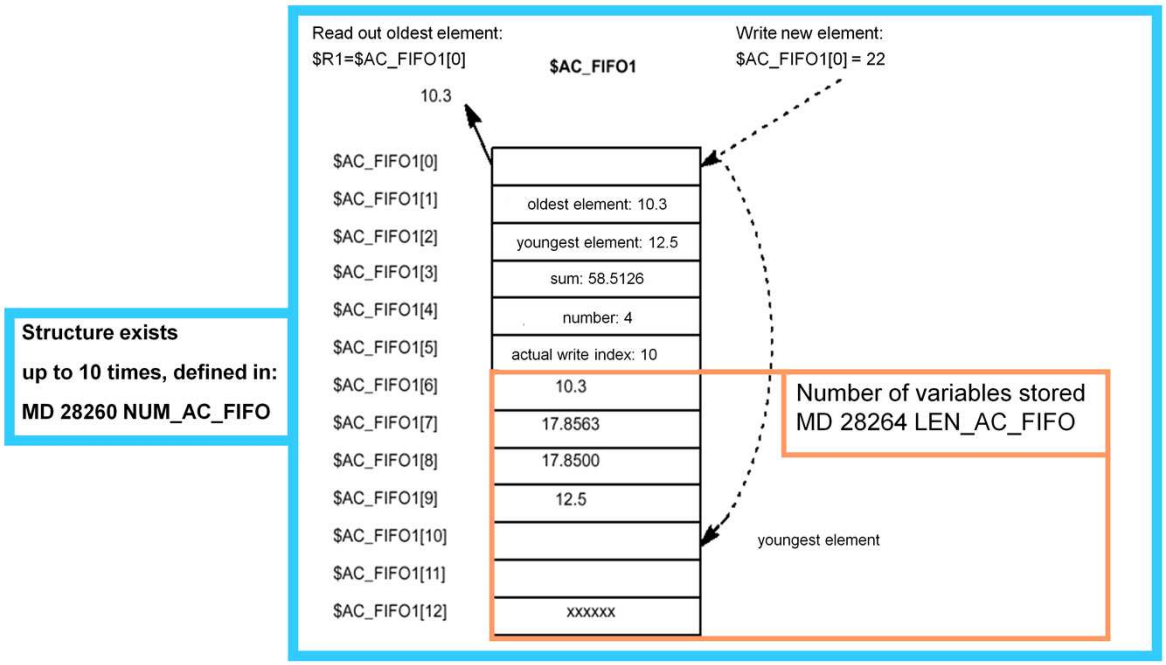
Reading invariable MD, SD

Machine and setting data whose settings do not vary are addressed from synchronized actions in the same way as in normal part program commands. They are preceded by a $ sign.

Reading/writing variable MD, SD

Machine data and setting data whose values may vary during machining are addressed from a synchronized action preceded by the **$$** sign.
This can be described with the relevant access right.

Special variables for synchronized actions
Real-time FIFO variables



 SITRAIN © Siemens AG 2017

FIFO variable

Up to 10 FIFO variables are provided to allow storage of related data sequences: **$AC_FIFO1[n]** to **$AC_FIFO10[n]**.

The number of available AC FIFO variables is defined in machine data **MD 28260: NUM_AC_FIFO**.

The number of values that can be stored in a FIFO variable is defined via machine data **MD 28264: LEN_AC_FIFO**.

Values in FIFO variables are of the REAL data type. Read and write access can be assigned to each element of the FIFO.

Storage

FIFO values are stored in the R parameter area, i.e. they are retained beyond the end of program, reset, and power ON and also archived.

MD 28262

START_AC_FIFO defines the number of the R parameter, which marks the beginning of FIFO variable storage in the R parameter area.

Index n

Indices 0 to 5 have a special significance:

- n= 0: When a variable is written with index 0, a new value is stored. When a variable is read with index 0, the oldest element is deleted from the FIFO
- n=1: Access to oldest stored element
- n=2: Access to most recent stored element
- n=3: Sum of all FIFO elements
- n=4: Number of elements available in FIFO. FIFO variables are reset by resetting the number of elements, to 0.
- n=5 Current write index relative to the FIFO beginning

MD 28266

MODE_AC_FIFO defines the mode for summation in index n=3:

- Bit 0 = 1 Sum (total) updated on each write operation
- Bit 0 = 0 No summation.

Coordination of synchronized actions by NC and PLC

	Meaning	TP[1)]	SA[2)]	PLC
LOCK(ID)	Lock synchronized action Positioning action is interrupted		X	
UNLOCK(ID)	Continue interrupted synchronized action/positioning		X	
RESET(ID)	Cancel synchronized action/positioning		X	
CANCEL(ID)	Abort technology cycle	X		
DB21.DBX1.2	Disabling of the synchronized actions			X

1) From part program
2) From synchronized action

Course NC-84SL-SIP Page 14 - 19 SITRAIN © Siemens AG 2017

PLC Modal synchronized actions (ID, IDS) can be locked or enabled by the PLC program.

Area of influence The PLC can control up to the first 64 modal synchronized actions by applying locks (ID, IDS 1-64). The synchronized actions **that can be locked** by the PLC are designated by the NC with 1 in the 64-bit field of the interface:

DB21-30, DBB308-315.

Disabling all synchronized actions

The PLC program can set **DB 21-30, DBB1 bit 2** to disable (lock against activation) all modal synchronized actions that are already defined in the NC and stored against activation.

Resetting DB 21-30, DBB1 bit 2 cancels the general disable by the PLC again .

Targeted disable One bit is reserved for each of first 64 IDs (1-64) in the PLC interface:

DB21-30, DBB 300 bit 0 to DB21-30 DBB 307 bit 7.

These functions are enabled by default (bits = 0).

When the allocated bit is set, evaluation of the condition and execution of the associated function are disabled in the NCK.

Cancellation of the disable

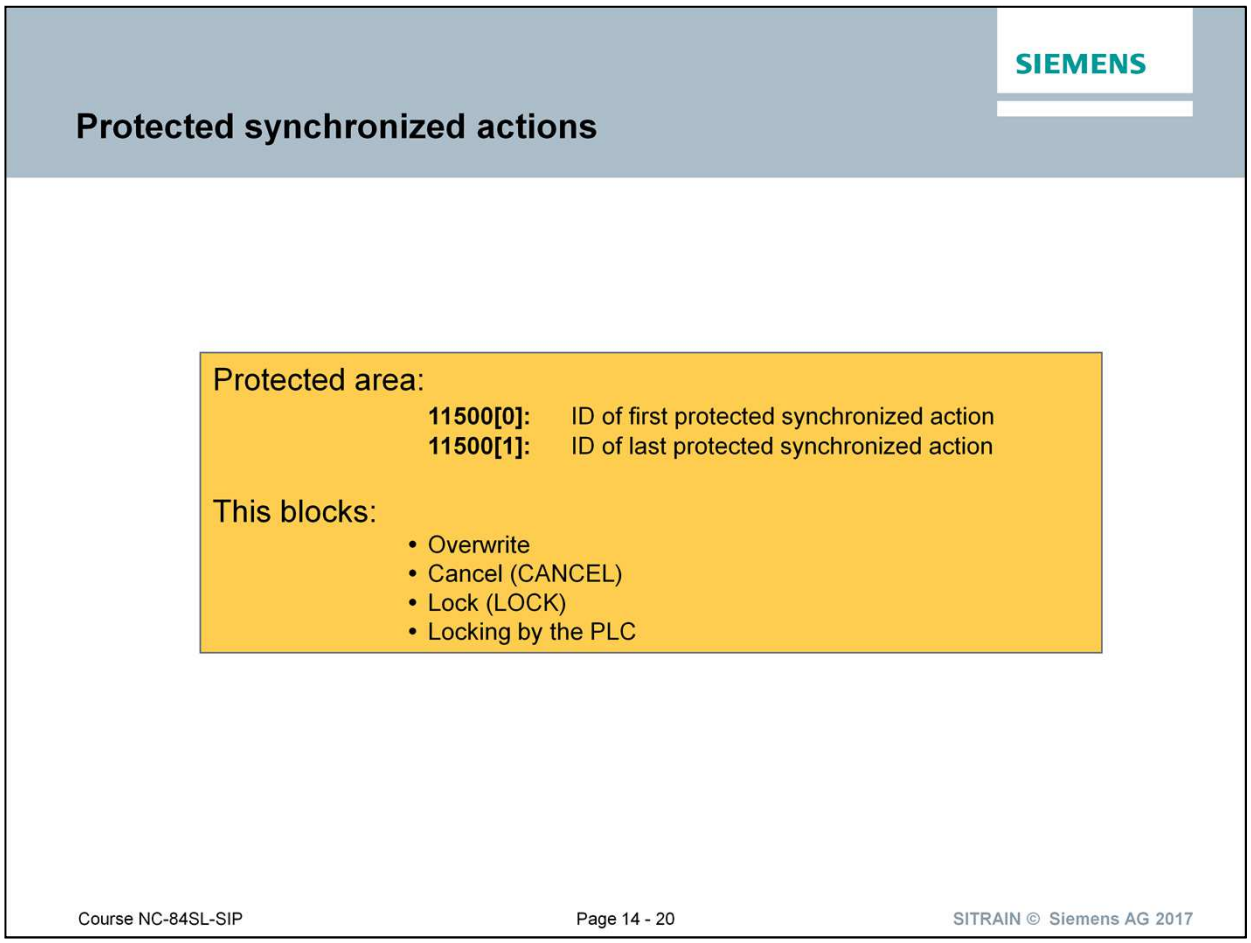
Resetting the bits corresponding to the ID, IDS number in DB 21-30, DBB 300, bit 0 to DB 21-30, DBB 307 bit 7 causes the PLC to enable a previously disabled synchronized action.

Update If the PLC user program has made changes in the range DB 21-30, DBB 300 bit 0 to DB 21-30, DBB 307 bit 7, it must activate these changes with

DB 21-30 DBX280.1.

Feedback If selective disabling was activated by the NCK, this is indicated in

DB 21-30 DBX.281.1.



Function RESET(ID)

The synchronized action can be started once; blocking is then active.

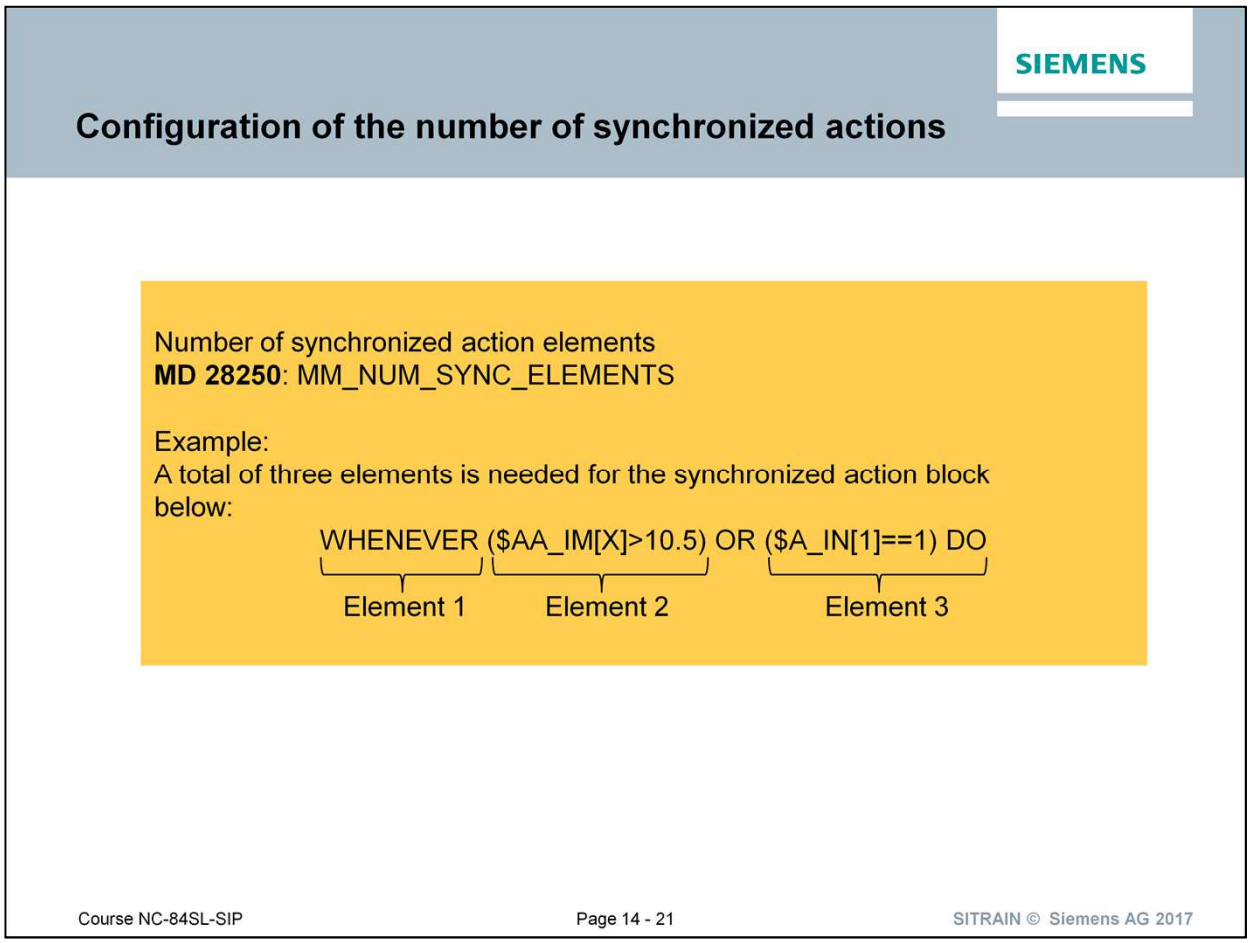
The RESET command is also active for protected synchronized actions.

Application

The end customer must be prevented from modifying reactions to certain states defined by the machine manufacturer.

The machine is commissioned by the manufacturer without protection. This enables the gating logic to be defined and tested. However, the manufacturer declares the range of synchronized actions he has used as protected before the system is delivered to the end customer, thus preventing the end customer from changing synchronized actions within this protected area.

A further application is the Safety Integrated function.



Number of blocks The number of programmable synchronized action blocks depends entirely on the configurable number of synchronized action elements.

Event-controlled program call (K1)

Call by	MD 20108 Bit n=1	Value $P_PROG_EVENT
Part program start	0	1
Part program end	1	2
Operator panel reset	2	3
Control system power up	3	4

SITRAIN © Siemens AG 2017

Function

With this function, a user program is run when certain events occurs. These events can be

- Part program start
- Part program end
- Operator panel reset
- Power up

Conditions

On which of these events the user program is actually called is defined in machine data MD20108. The program itself must be in a specific path and have a specific name so that it can be called.

The user program must be loaded into the NCK.

Storage directory

Directory of manufacturer cycles

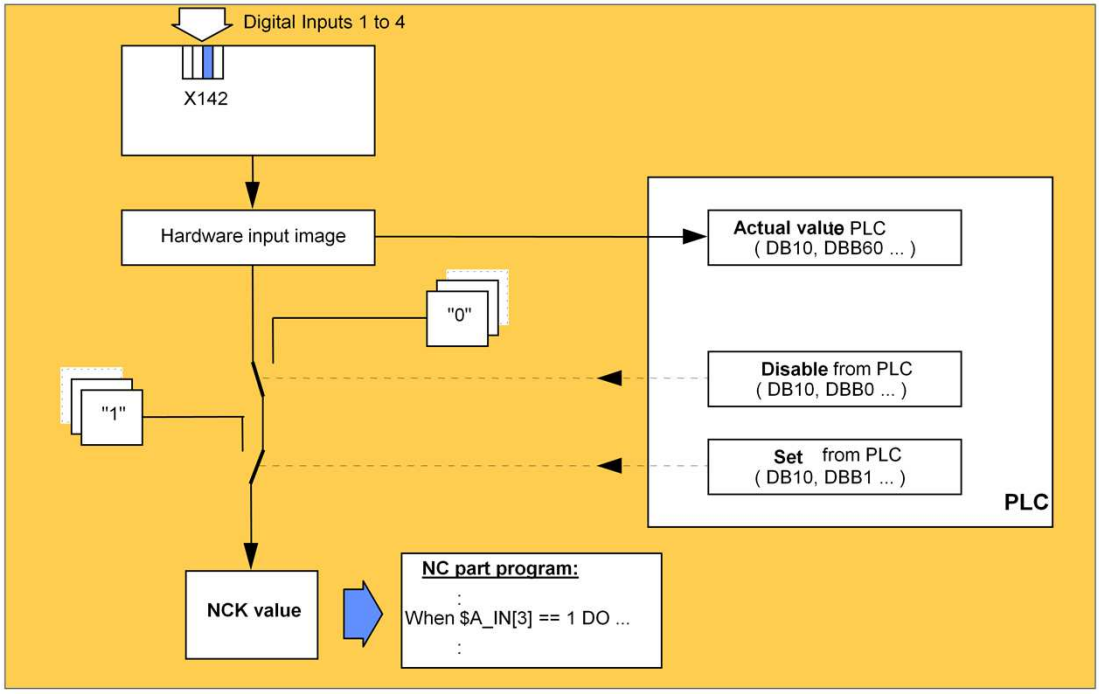
Program name

PROG_EVENT.SPF

Event dependence

In the program PROG_EVENT.SPF, a program can be written by querying the system variable $P_PROG_EVENT, depending on the event resulting in the call.

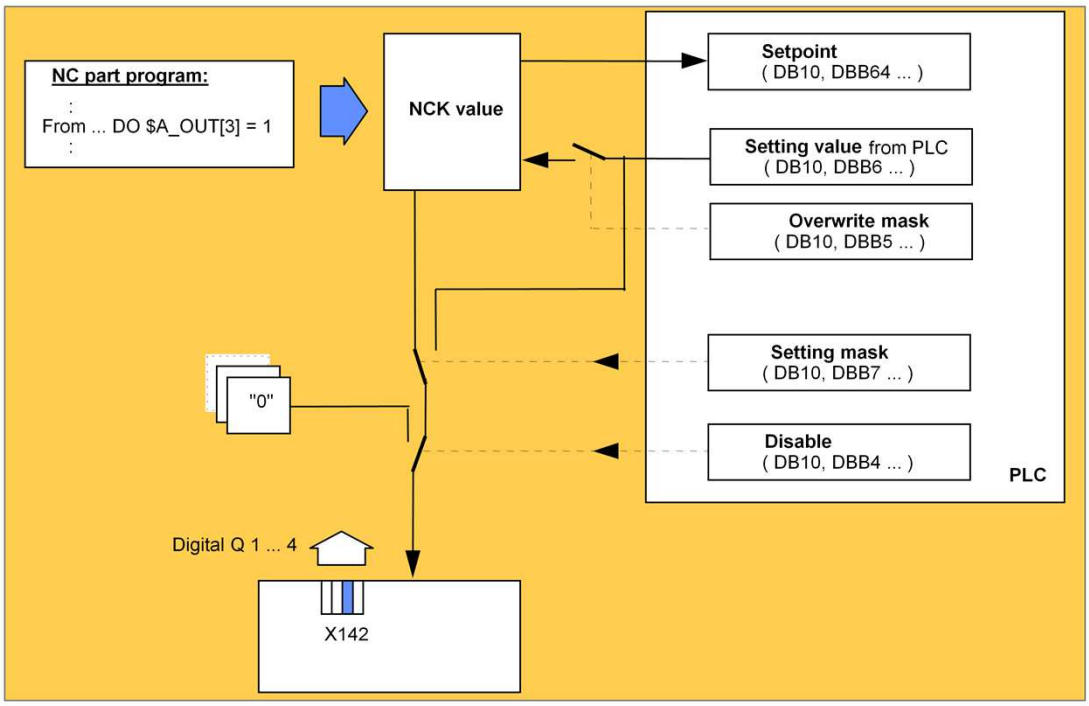
Signal flow digital inputs



Course NC-84SL-SIP Page 14 - 23 SITRAIN © Siemens AG 2017

Notes

Signal flow digital outputs



Course NC-84SL-SIP Page 14 - 24 SITRAIN © Siemens AG 2017

Notes

System variable for fast I/Os

X142

Pin	Signal
1	
2	
3	$A_IN[1]
4	$A_IN[2]
5	M
6	$A_IN[3]
7	$A_IN[4]
8	M
9	$A_OUT[1]
10	$A_OUT[2]
11	M
12	$A_OUT[3]
13	$A_OUT[4]
14	M



System variable	Meaning
$A_IN[n]	NCK value of the digital input n
$A_OUT[n]	NCK value of the digital output n

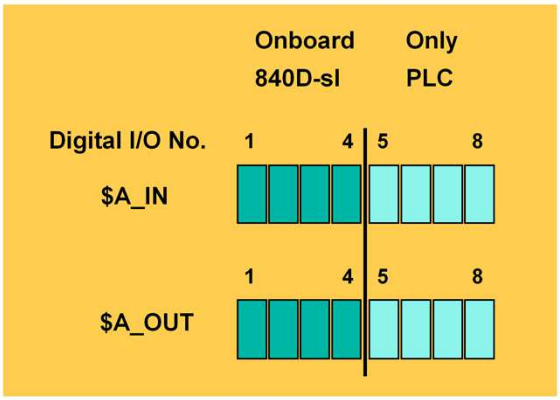
Course NC-84SL-SIP Page 14 - 25 SITRAIN © Siemens AG 2017

Switches and LEDs on the simulator

The PLC input switches 0-3 are also wired to the terminals for $A_IN[1]-[4] of the X142.

The PLC output LEDs 0-3 are also wired to the terminals for $A_OUT[1]-[4] of the X142.

Number of fast I/Os



Course NC-84SL-SIP Page 14 - 26 SITRAIN © Siemens AG 2017

Activation of the I/Os

Number of digital I/O bytes

MD 10 350: FASTIO_DIG_NUM_INPUTS = 0 ... 1;

MD 10 360: FASTIO_DIG_NUM_OUPUTS = 0 ... 1; 0

The number of existing digital input or output bytes (default value 1 or 0, i.e. the on-board inputs of the NCU module are already active, but not the outputs!) must be entered.

Input value = 1 1st byte: Inputs/outputs 1 ... 4 (8)

On-board I/Os

In the 1st byte, only 4 inputs/outputs - Nos. 1 to 4 - are present in the hardware on the SINUMERIK 840D-sl.
Via the NC/PLC interface, signals may also be passed by the PLC to the absent inputs 5 to 8. They can then be evaluated in the NC part program.

Interrupts

Interrupts 1-4 are permanently assigned to inputs 1-4.
That is important when NC programs are started by the PLC (FB4/FB7, function ASUP and FC9).

For this reason, other interrupt numbers must also be assigned (1-16 are possible) if on-board inputs of the NCU are used.

Exercise: Tracing the function of a synchronized action

Objective: A check is to be made to see whether an output signal is output with reference to a position on the traversing path.

Task Write a synchronized action that is valid during runtime of an NC program. This must then activate the digital NC output 1 at 10 mm from the beginning of the programmed traversing path.
Check that the objective has been met using the NC trace.

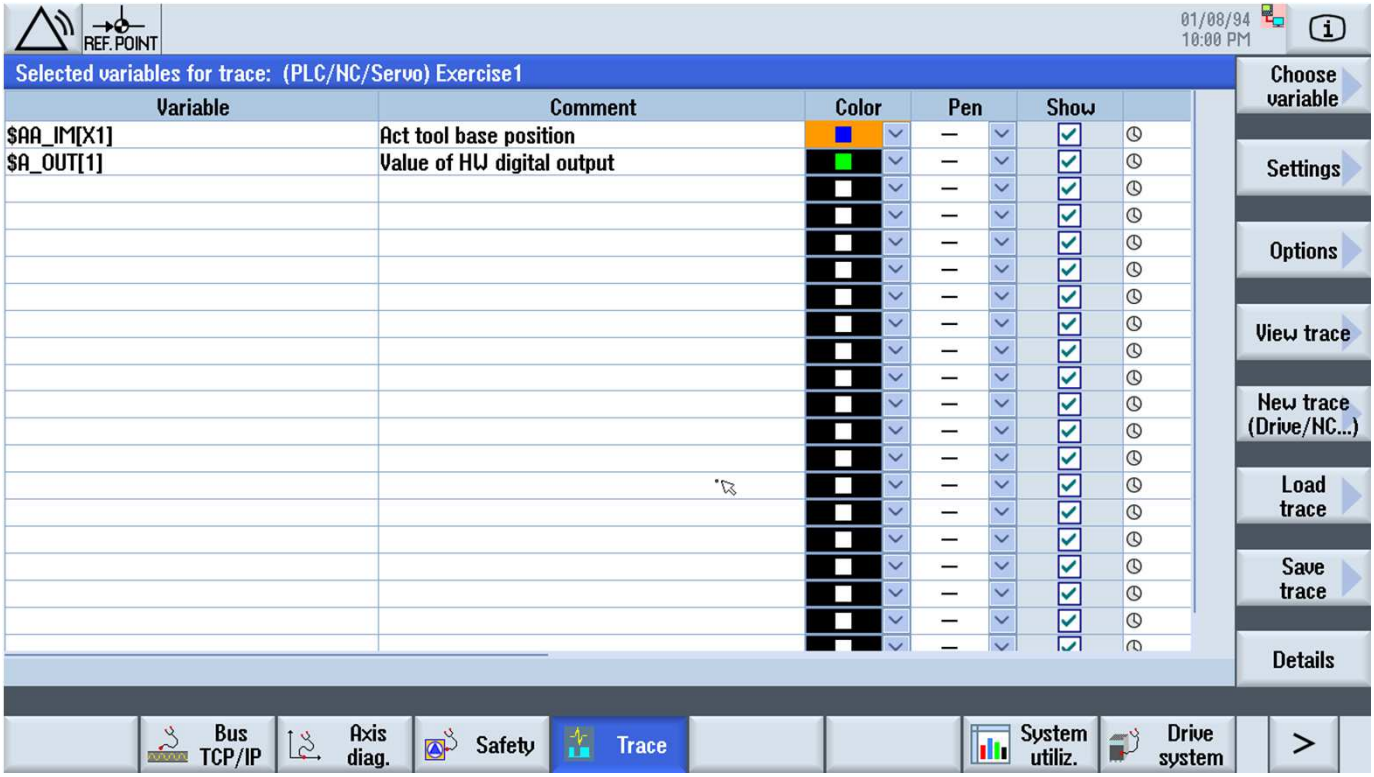
Note If a synchronized action is longer, it will be shown with a line break in this document, **a synchronized action is always one line on the NC!**

Write the NC program:

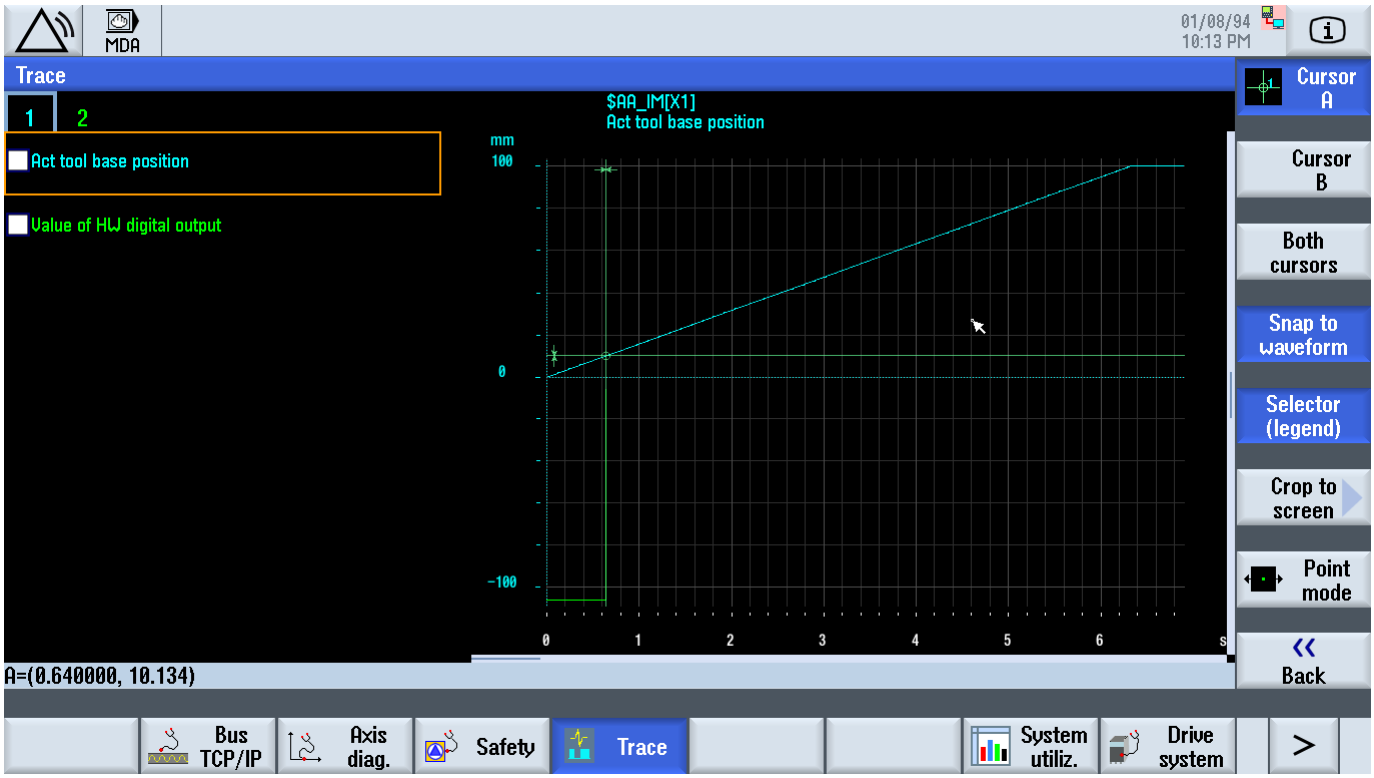
```
N10 G0 X0 Y0 Z0
N20 ID=1 WHEN $AC_DTBW >= 10 DO $A_OUT[1]=1
N30 $AN_SLTRACE=1
N40 G1 F1000 X100
N50 $AN_SLTRACE=2
N60 X0
N70 M01
N80 DO $A_OUT[1]=0
N90 M30
```

With the function "NC/PLC Trace", check the function.
Here, it is also possible to display the progress of the traced signals over time.

Select the operating area Diagnostics > Extended and then the "NC/PLC Trace" function. Enter the required signals. Press the "View Trace" softkey to start recording.



In the diagram, you can see a possible result of a synchronized action, in which a digital output is output starting from a certain distance from the beginning of the block.



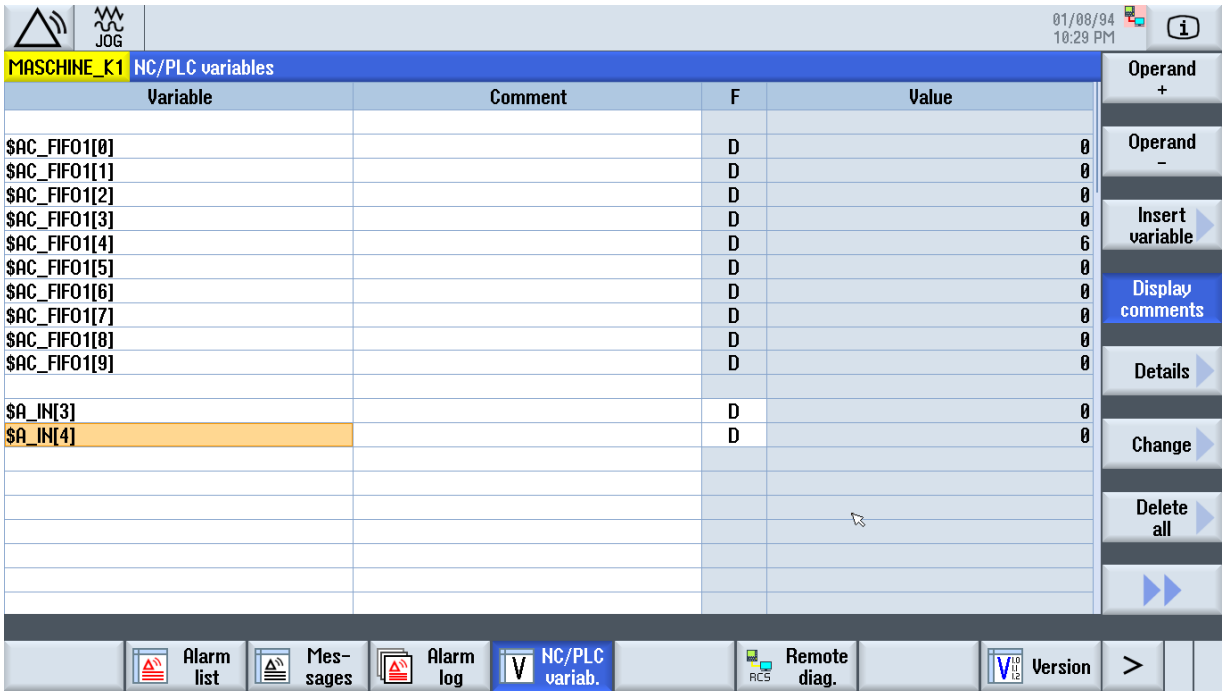
Additional exercise: FIFO handling

Set up a FIFO buffer with space to store 5 variables.
With a synchronized action, the aim is to store in this buffer axis actual values from an axis moving back and forth. The values are read out with a second synchronized action into an R Parameter (R3) (which empties the FIFO buffer).
In the variables view, create an image of the FIFO buffer to observe the effect of the read and write operations.

Step 1: Create a FIFO.

MD28260 $MC_NUM_AC_FIFO = 1 >> Creates a FIFO buffer
MD28262 $MC_START_AC_FIFO = 10 >> As of R10, the FIFO values are stored in the R parameter area
MD28264 $MC_LEN_AC_FIFO = 5 >> Create 5 FIFO variables

Step 2: Create the variables view



Step 3: NC programming

```
N10 G0x0
N20 R0=20
N30 ID=3 EVERY (($A_IN[3]>0) AND ($AC_FIFO1[4]<5)) DO
                                    $AC_FIFO[0.0]=$AA_IM[X1]
N40  ID=4 EVERY (($A_IN[4]>0) AND ($AC_FIFO1[4]>=1)) DO
                                    R3=$AC_FIFO[0.0]
N50 G1 F1000
N60 LOOP
N70 X0
N80 X100
N90 ENDLOOP
N100 M30
```

Step 4: Test

Additional exercises:

Use the DocOnCD to find the necessary system variables.

Tip

You can thus monitor the variables $A_IN[..], $A_OUT[..]:

> OA 'Diagnostics' > SK: 'NC/PLC variab.' > SK: 'Insert variable'

> Select **Filter = All (no filter) >** Search for **"$A_IN" or $A_OUT"**

> Select the variable **"digitInpVal"** or **"digitOutpVal"**

> specify Index **0 / 1 / 2 / …**

1st exercise

Program an axis movement from the starting point X0, Y0 to the end point X100, Y100 with 500 mm/min.

15 mm after the beginning of the movement, digital output 1 must be set to "1".

2nd exercise

Program an axis movement from the starting point X0, Y0 to the end point X100, Y100 with 500 mm/min.

At X=40mm, digital output 1 is to be set to "1".

60 mm before the end of the movement, the additive path feedrate override $AC_VC of 100 mm/min must take effect.

3rd exercise

Program an axis movement from the starting point X0, Y0 to the end point X100, Y100 with 100 mm/min.

Fast deletion of distance-to-go must be possible at any time through digital input 1.

4th exercise

Evaluation function

Program a square with side length 100 mm.
It must be traversed at feedrate 500 mm/min.

During the entire program the following applies:

The override must be 100% at the beginning of the block (each page) and be reduced continuously to 30% by the end of the movement.

5th exercise

Online correction

During the movement from X0 to X100, the Y axis must be corrected according to the function $y = 1/100000 * x^3$ to the end position 10.

6th exercise

Move from position X0 Y0 in one block to X500 Y500.
As the program runs, the override is to be reduced to 10% when the drive load $AA_LOAD for the X axis exceeds 2%.

The drive load =<1%, the override is to be set back to 100%.

Notice: Check the **MD 36730 DRIVE_SIGNAL_TRACKING =1**.
The option "evaluation of internal drive variables" 6FC5800-0A**M41**-0YB0 is also needed.

7th exercise

Event-controlled program call

When the control is run up, a synchronized action is to be started that sets on-board output 1 of the NCK every time NCK input 1 is set.
This input is to be reset by NCK input 2.

Solutions:

1st exercise

```
G90 G0 X0 Y0
WHENEVER $AC_DTBW > 15 DO $A_OUT[1]=1
G91 G01 X100 Y100 F500
M30
```

2nd exercise

```
G90 G0 X0 Y0
WHENEVER $AA_IM[X]>=40 DO $A_OUT[1]=1
WHENEVER $AC_DTEW<=60 DO $AC_VC=100
G91 G01 X100 Y100 F500
M30
```

3rd exercise

```
G90 G0 X0 Y0
WHEN $A_IN[1]==1 DO DELDTG
G91 G01 X100 Y100 F100
M30
```

4th exercise

```
G90 G0 X0 Y0
FCTDEF(1,0,100,100,-70)
ID=1 DO SYNFCT(1, $AC_OVR, $AC_PATHN)
G01 X100 F500
Y100
X0
Y0
M30
```

5th exercise

```
G90 G0 X0
FCTDEF(1, 0, 10, 0, 0, 0, 1/100000)
DO SYNFCT(1,$AA_OFF[Y],$AA_IM[X])
G1 X100 F500
M30
```

6th exercise

```
G90 G0 X0 Y0
ID=1 WHENEVER $AA_LOAD[X]>2 DO $AC_OVR=10
ID=2 FROM $AA_LOAD[X]<=1 DO $AC_OVR=100
G1 X500 Y500 F5000
```

7th exercise

Set MD 20108 bit 3 to "1" and perform NCK Reset.

Copy PROG_EVT.SPF and rename it to PROG_EVENT.SPF.
Change content:

```
IF ($P_PROG_EVEVT= = 4)
      IDS=1 WHENEVER $A_IN[1] DO $A_OUT[1]=1
      IDS=2 WHENEVER $A_IN[2] DO $A_OUT[1]=0
ENDIF
M17
```

Load program; perform NCK Reset

Description of Functions 11/2002 Edition

sinumerik

Synchronized Actions
SINUMERIK 840D/840Di/810D

SIEMENS

2.3.8 List of system variables relevant to synchronized actions

Overview

The following table shows a list of all system variables (sorted according to category) that can be read or write-accessed from synchronized actions. The access options are specified.

Legend:

r	Read
w	Write
R	Read with implicit preprocessor stop
W	Write with implicit preprocessor stop
PP	Part program
SA	Synchronized action
SW	For SW version see note

Notice

Type specifies the software version (e.g. /4) in which the system variables were introduced if they have not existed since SW 2.

SA access specifies the software version in which **access to the system variable** from synchronized actions was introduced if this was not available since the introduction of the system variables.
The name component “ACT” in system variables for synchronized actions (e.g. $AA_V**ACT**M) identifies **setpoints** which are calculated in the interpolator and used as input variables for axis control.
The name prefix “$VA_...” identifies genuine actual values of a machine axis which are reproduced by evaluating the encoder information.

User variables

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_MARKER[n]	INT	Marker variable for motion-synchronous actions	Counter	R/W	r/w
$AC_PARAM[n]	DOUBLE	Arithmetic variable for motion-synchronous actions	Counter	R/W	r/w
$AC_FIFOi[n]	DOUBLE /4	i: 1-10, No. of FIFO variable n: Parameter number, 0 – max. FIFO element. Meanings of n: n=0: When the variable is written with index 0, a new value is stored in the FIFO. When a variable is read with index 0, the oldest element is read and deleted from the FIFO. n=1: Read access to oldest element n=2: Read access to latest element n=3: Total of all elements stored in the FIFO if bit 0 is enabled in MD $MC_MM_MODE_FIFO. n=4: Read access to the current number of FIFO elements n= 5 – m: Read access to individual FIFO elements. 5 is the oldest element, 6 is the second oldest, etc.	No. of parameter	R/w	r/w

© Siemens AG 2002. All rights reserved

R parameters

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$R[n], Rn	DOUBLE	The max. number of R parameters is defined in the machine data. R parameters are addressed from synchronized actions with $R or $R[i]. Otherwise ...Rn or R[n] is used.	Counter	r/w	r/w /4

System data

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AN_SETUP_TIME	DOUBLE	IF $AN_SETUP_TIME > 60000 GOTOF MARK01 time since last control start-up with default data.	Minutes	R	r /5
$AN_POWERON_TIME	DOUBLE	IF $AN_POWERON_TIME == 480 GOTOF MARK02 time since last normal control start-up.	Minutes	R	r /5
$AN_NCK_VERSION	DOUBLE	NCK version: Only the part of the floating point number before the decimal point is interpreted; the part after the decimal point may contain IDs for intermediate development releases. The part before the decimal point contains the official ID of the software release for the NCK: For example, the value of the variable is 200000.0 for NCK Release 20.00.00.	NCK version	R	r /6

Tool data

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_TOOLMN[t]	INT	Magazine - number of tool t	T number	R	r /4
$A_TOOLMLN[t]	INT	Magazine - number of tool t	T number	R	r /4
$A_MONIFACT	REAL	Factor for tool life monitoring		R/W	r /4
$AC_MONMIN	REAL	Actual value to setpoint value ratio for tool monitoring. Threshold for tool search strategy "Only load tools with actual value greater than threshold"		R/W	r /5
$A_DNO[i]	INT	Read a D number specified by PLC via VDI interface	Index	R	r /4

© Siemens AG 2002. All rights reserved

G groups

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_GG	INT	$A_GG[n] Read current G function of a group from synchronized action n: number of G group	as for PLC interface		r /5
$P_ACTID[n]	BOOL	Modal synchronized action with ID n active, if TRUE n: 1 – 16	Synchro-nized action with ID	R	r /2
$P_GG[n]	INT	Read current G function of a group from parts program n: number of G group	as for PLC interface	R	/2

Channel statuses

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_STAT	INT /4	Channel status -1: Invalid 0: Channel reset 1: Channel interrupted 2: Channel active		R	r /4
$AC_PROG	INT /4	Program status -1: Invalid 0: Program reset 1: Program stopped 2: Program active 3: Program waiting 4: Program interrupted		R	r /4
$AC_SYNA_MEM	INT /4	Free memory for motion-synchronous actions indicates how many elements of the memory allocated with $MC_MM_NUM_SYNC_ELEMENTS are still free.		R	r /4
$AC_IPO_BUF	INT /4	Fill level of interpolation buffer		R	r /4
$AC_IW_STAT	INT /5	Positional information for articulated joints (transformation-specific) for Cart. PTP travel		R	r /5
$AC_IW_TU	INT /5	Positional information for axes (MCS) for Cart. PTP travel		R	r /5
$A_PROBE[n]	INT /4	$A_PROBE[1]: Status of first probe $A_PROBE[2]: Status of second probe 0: Not deflected 1: Deflected	No. of probe	R	r
$AC_MEA[n]	INT	Probe activated if TRUE (1) 1 – MAXNUM_PROBE	No. of probe	R	r /4
$AC_TRAFO	INT	Code number of the active transformation with reference to machine data $MC_TRAFO_TYPE n	-	R	r /4
$AC_LIFTFAST	INT	Rapid lift: 0: No reverse stroke was active 1: Reverse stroke was active	-	R/W	r/w

© Siemens AG 2002. All rights reserved

ASUBs

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_ASUP	INT /4	See also: **References:** /FB/, K1, (Mode Group, Channel, Program Operation Mode) Remarks: Reasons for ASUB activation. Act. cau.: Cause of activation Act. by: Source of activation Cont.: Option(s) for continuation **BIT 0:** Act. cau.: User interrupt “ASUB with Blsync” (block synchronization), Act. by: Vdi signal, dig./anal. interface, Cont. : Either Reorg or Ret **BIT 1:** Act. cau.: User interrupt “ASUB”. (The position after the block in which the stoppage took place is stored for program continuation with Repos.) Act. by: Vdi signal, dig./anal. interface Cont.: Either Reorg or Ret **BIT 2:** Act. cau.: User interrupt “ASUB from Ready channel status”, Act. by: Vdi signal, dig./anal. interface Cont.: Either Reorg or Ret **BIT 3:** Act. cau: User interrupt “ASUB in a manual mode and channel status not READY ” Act. by: Vdi signal, dig./anal. interface Cont.: Either Reorg or Ret **BIT 4:** Act. cau: User interrupt “ASUB”. (The position at the time of the interrupt is stored for program continuation with Repos). Act. by: Vdi signal, dig./anal. interface Cont.: Either Reorg or Ret **BIT 5:** Act. cau.: Cancellation of subprogram repetition Act. by: Vdi signal Cont.: Use of the REPOS system ASUB **BIT 6:** Act. cau.: Activation of decoding single block Act. by: Vdi signal (+OPI) Cont.: Use of the REPOS system ASUB **BIT 7:** Act. cau.: Activation of delete distance−to−go; Act. by: Vdi signal Cont.: Use of the Ret system ASUB **BIT 8:** Act. cau: Activation of axis synchronization Act. by: Vdi signal Cont.: Use of the REPOS system ASUB **BIT 9:** Act. cau.: Mode change Act. by: Vdi signal Cont.: Use of the REPOS or RET system ASUB (see MD)		R	r /4

© Siemens AG 2002. All rights reserved

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_ASUP ff.	INT /4	**BIT 10:** Act. cau.: Program continuation under teach-in or after teach-in deactivation Act. by: Vdi signal Cont.: Use of the Ret system ASUB **BIT 11:** Act. cau.: Overstore selection Act. by: PI service Cont.: Use of the REPOS system ASUB **BIT 12:** Act. cau.: Alarm with reaction correction block with Repos Act: by: Internal Cont.: Use of the REPOS system ASUB **BIT 13:** Act. cau.: Retraction movement with G33 and Stop Act. by: Internal Cont.: Use of the Ret system ASUB **BIT 14:** Act. cau.: Activation of dry run feedrate; Act. by: Vdi signal Cont.: Use of the REPOS system ASUB **BIT 15:** Act. cau.: Deactivation of dry run feedrate; Act. by: Vdi signal Cont.: Use of the REPOS system ASUB **BIT 16:** Act. cau.: Activation of skip block; Act. by: Vdi signal Cont.: Use of the REPOS system ASUB **BIT 17:** Act. cau.: Deactivation of block skip ; Act. by: Vdi signal Cont.: Use of the REPOS system ASUB **BIT 18:** Act. cau: Activation of machine data Act. by: PI service Cont.: Use of the REPOS system ASUB **Bit 19**: Act. cau: Set tool offset active Act. by: PI service "_N_SETUDT" Cont.: Use of the REPOS system ASUB **Bit 20**: Act. cau: System ASUB after SERUPRO type search run has reached the destination. Act. by: Pi serv. "_N_FINDBL" parameter == 5 Cont.: Use of the REPOS system ASUB		R	r /4

Commands to / from channel (diagnostics)

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_PROTO	BOOL /4	Activate/deactivate log function for first user 0: Deactivate 1: Activate		R/W	r/w
$A_PROTOC	BOOL /4	Activate/deactivate log function for a user 0: Deactivate 1: Activate		R/W	r/w

© Siemens AG 2002. All rights reserved

Inputs/outputs

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_IN[n]	BOOL	Digital input of NC	No. of input	R	r
$A_OUT[n]	BOOL	Digital output of NC	No. of output	R/w	r/w
$A_INA[n]	DOUBLE	Analog input of NC	No. of input	R	r
$A_OUTA[n]	DOUBLE	Analog output of NC After the write operation, the value does not become active until the next interpolator cycle; it can then be read back.	No. of output	R/w	r/w
$A_INCO[n]	BOOL	Comparator input	No. of input	R	r

Read and write PLC variables

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_DBB[n]	INT /4	Read/write data byte (8 bits) from/to PLC	Offset in I/O area	R/w	r/w
$A_DBW[n]	INT /4	Read/write data word (16 bits) from/to PLC	Offset in I/O area	R/w	r/w
$A_DBD[n]	INT /4	Read/write double data word (32 bits) from/to PLC	Offset in I/O area	R/w	r/w
$A_DBR[n]	DOUBLE /4	Read/write real data (32 bits) from/to PLC	Offset in I/O area	R/w	r/w

Link variables

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_DLB[n]	INT /5	Read/write data byte (8 bits) from/to NCU link	Position offset in link memory	R/w	r/w
$A_DLW[n]	INT /5	Read/write data word (16 bits) from/to NCU link	Position offset in link memory	R/w	r/w
$A_DLD[n]	INT /5	Read/write double data word (32 bits) from/to NCU link	Position offset in link memory	R/w	r/w

© Siemens AG 2002. All rights reserved

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_DLR[n]	DOUBLE /5	Read/write real data (32 bits) from/to NCU link	Position offset in link memory	R/w	r/w
$A_LINK_TRANS _RATE	INT /5	Number of bytes which can still be transferred via the NCU link in the present interpolation cycle.	-	r/	r/

Direct PLC_IO

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_PBB_IN[n]	INT /5	Read data byte (8 bits) directly from PLC I/O	Position offset in PLC input area	R/	r/
$A_PBW_IN[n]	INT /5	Read data word (16 bits) directly from PLC I/O	Position offset in PLC input area	R/	r/
$A_PBD_IN[n]	INT /5	Read double data word (32 bits) directly from PLC I/O	Position offset in PLC input area	R/	r/
$A_PBR_IN[n]	DOUBL E /5	Read real data (32 bits) directly to PLC I/O	Position offset in PLC input area	R/w	r/w
$A_PBB_OUT[n]	INT /5	Write data byte (8 bits) directly to PLC I/O	Position offset in PLC output area	R/w	r/w
$A_PBW_OUT[n]	INT /5	Write data word (16 bits) directly to PLC I/O	Position offset in PLC output area	R/w	r/w

© Siemens AG 2002. All rights reserved

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_PBD_OUT[n]	INT /5	Write double data word (32 bits) directly to PLC I/O	Position offset in PLC output area	R/w	r/w
$A_PBR_OUT[n]	DOUBLE /5	Write real data (32 bits) directly to PLC I/O	Position offset in PLC output area	R/w	r/w

Tool management

References: /FBW/, Tool Management

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_TC_FCT	INT /5	Command number. This specifies the desired operation.	-	R/	r/
$AC_TC_STATUS	INT /5	Status of the command to be read with $AC_TC_FCT.	-	R/	r/
$AC_TC_THNO	INT /5	Number of the toolholder (spindle no.) in which the new tool is to be loaded.	-	R/	r/
$AC_TC_TNO	INT /5	NCK-internal T number of the new tool (to be loaded). 0: There is no new tool.	-	R/	r/
$AC_TC_MFN	INT /5	Source magazine number of the new tool. 0: There is no new tool.	-	R/	r/
$AC_TC_LFN	INT /5	Source location number of the new tool. 0: There is no new tool.	-	R/	r/
$AC_TC_MTN	INT /5	Destination magazine number of the new tool. 0: There is no new tool.	-	R/	r/
$AC_TC_LTN	INT /5	Destination location number of the new tool. 0: There is no new tool.	-	R/	r/
$AC_TC_MFO	INT /5	Source magazine number of the old tool (to be replaced). 0: There is no old tool.	-	R/	r/
$AC_TC_LFO	INT /5	Source location number of the old tool (to be replaced). 0: There is no old tool.	-	R/	r/
$AC_TC_MTO	INT /5	Destination magazine number of the old tool (to be replaced). 0: There is no old tool.	-	R/	r/
$AC_TC_LTO	INT /5	Destination location number of the old tool (to be replaced). 0: There is no old tool.	-	R/	r/

© Siemens AG 2002. All rights reserved

Timers

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_YEAR	INT	System time, year		R	r /4
$A_MONTH	INT	System time, month		R	r /4
$A_DAY	INT	System time, day		R	r /4
$A_HOUR	INT	System time, hour		R	r /4
$A_MINUTE	INT	System time, minute		R	r /4
$A_SECOND	INT	System time, second		R	r /4
$A_MSECOND	INT	System time, millisecond		R	r /4

$AC_TIME	DOUBLE	Time from block start in seconds		R	r /4
$AC_TIMEC	DOUBLE	Time from block start in interpolation cycles		R	r /4
$AC_TIMER[n]	DOUBLE /4	Timer unit in seconds. The time is counted internally in multiples of the interpolation cycle. Incrementation of a timer variable is started by means of value assignment $AC_TIMER[n]=<start value>. Incrementation of a timer variable is stopped through the assignment of a negative value: $AC_TIMER[n]=-1 The current time value can be read whether the timer variable is running or halted. After a timer variable has been stopped through the assignment of -1, the current time value remains stored and can be read. The value is defined in MD 28258: MM_NUM_AC_TIMER	Counter	R/W	r/w

© Siemens AG 2002. All rights reserved

Path motion

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_PATHN	DOUBLE	Normalized path parameter, value between 0 = start of block and 1 = end of block		R	r
$AC_DTBW	DOUBLE	Geometric distance from start of block in workpiece coordinate system. The programmed position alone determines the distance calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.		R	r
$AC_DTBB	DOUBLE	Geometric distance from start of block in basic coordinate system. The programmed position alone determines the distance calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.		R	r
$AC_DTEW	DOUBLE	Geometric distance from end of block in workpiece coordinate system. The programmed position alone determines the distance calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.		R	r
$AC_DTEB	DOUBLE /3	Geometric distance from end of block in basic coordinate system. The programmed position alone determines the distance calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.		R	r
$AC_PLTBB	DOUBLE /3	Path distance from start of block in basic coordinate system. The variable can only be accessed from synchronized actions.		R	r
$AC_PLTEB	DOUBLE	Path distance to end of block in basic coordinate system. The variable can only be accessed from synchronized actions.		R	r
$AC_DELT	DOUBLE	Path distance-to-go in workpiece coordinate system after deletion of distance-to-go for motion synchronized actions		R	r
$P_APDV	BOOL	Returns TRUE if the positional values that can be read with $P_APR[X] or $P_AEP[X] (start point/contour point with smooth approach and retraction) are valid.		R	r /4

© Siemens AG 2002. All rights reserved

Velocities, channel-specific

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$P_F	DOUBLE	Path feed last programmed		R	r
$AC_OVR	DOUBLE	Path override for synchronized actions: Multiplicative override component, acts in addition to operator override, programmed override and transformation override. The total factor is limited to 200%. It must be written again in every interpolation cycle, otherwise the value 100% applies. $AA_OVR[S1] changes the spindle override. The override defined by the machine data MD 12100: OVR_FACTOR_LIMIT_BIN, MD 12030: OVR_FACTOR_FEEDRATE[30], MD 12010: OVR_FACTOR_AX_SPEED[30] is not exceeded		R/W	r/w /4
$AC_VC	DOUBLE	Additive path feedrate override for synchronized actions The compensation value is not active with G0, G33, G331, G332 or G63. The compensation value must be written again in every interpolator cycle, otherwise the value 0 applies. An override of 0 cancels the compensation value; otherwise the override has no effect on the compensation value. The total feedrate cannot be negative as a result of the compensation value. The upper value is limited such that the maximum axis velocities and accelerations are not exceeded. The calculation of other feedrate components is not affected by $AC_VC. The override values defined by the machine data MD 12100: OVR_FACTOR_LIMIT_BIN, MD 12030: OVR_FACTOR_FEEDRATE[30], MD 12010: OVR_FACTOR_AX_SPEED[30], MD 12070: OVR_FACTOR_SPIND_SPEED are not exceeded. The additive feedrate override is limited such that the resulting feedrate does not exceed the maximum override value of the programmed feedrate.		R/W	r/w /4
$AC_VACTB	DOUBLE	Path velocity in basic coordinate system		R	r
$AC_VACTW	DOUBLE	Path velocity in workpiece coordinate system		R	r

Spindle data

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_S[n]	DOUBLE /4	Actual spindle speed The sign identifies the direction of rotation n: Spindle number, 0 ... max. spindle number	Spindle no.	RS	r /4
$AC_CONSTCUT_S[n]	DOUBLE /6	Current constant cutting rate. n: Spindle number, 0 ... max. spindle number	Spindle no.	RS	r /6

© Siemens AG 2002. All rights reserved

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_SDIR[n]	INT /3	Spindle rotation currently active 3: Spindle rotation, clockwise, 4: Spindle rotation, counterclockwise, 5: Spindle stop n: Spindle number, 0 ... max. spindle number	Spindle no.	RS	r /3
$AC_SMODE[n]	INT /3	Spindle mode currently active: 0: No spindle present in channel 1: Speed control mode 2: Positioning mode 3: Synchronous mode 4: Axis mode n: Spindle number, 0 ... max. spindle number	Spindle no.	RS	r /3
$AC_SGEAR[n]	INT /5	Gear stage currently active 1: 1st gear stage is active 2: 2nd gear stage is active 3: 3rd gear stage is active 4: 4th gear stage is active 5: 5th gear stage is active n: Spindle number, 0 ... max. spindle number	Spindle no.	RS	r /5
$AC_MSNUM	INT /3	Returns the number of the active master spindle: 0: No spindle present 1..n: Number of master spindle		RS	r /3
$AC_MTHNUM	INT /5	Returns the number of the active master toolholder: 0: No master toolholder present 1..n: Number of master toolholder		RS	r /5

Polynomial values for synchronized actions

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_FCTiLL,** **$AC_FCTLL[j]	DOUBLE /4	Lower limit of polynomial for synchronized actions (SYNFCT) i: 1–3, evaluation function FCTDEF 1 – 3 j: Polynomial number		R/W	r/w /4
$AC_FCTiUL,** **$AC_FCTUL[j]	DOUBLE /4	Upper limit of polynomial for synchronized actions (SYNFCT) i: 1–3, evaluation function FCTDEF 1 – 3 j: Polynomial number		R/W	r/w /4
$AC_FCTiC[n],** **$AC_FCT0[n]	DOUBLE /(4)	i: 1 – 3, polynomials 1 to 3; coefficient n: 0 – 3 a_0 coefficient for polynomial n		R/W	r/w /4
$AC_FCT1[n]	DOUBLE /(4)	a_1 coefficient for polynomial n		R/W	r/w /4
$AC_FCT2[n]	DOUBLE /(4)	a_2 coefficient for polynomial n		R/W	r/w /4
$AC_FCT3[n]	DOUBLE /(4)	a_3 coefficient for polynomial n		R/W	r/w /4

© Siemens AG 2002. All rights reserved

Channel statuses

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AC_ALARM_STAT	INT /5	(Selected) alarm reactions for synchronized actions (SYNFCT) Bit 2 = 1 NOREADY (active rapid deceleration and cancellation of servo enable) Bit 6 = 1 STOPBYALARM (ramp stop of all channel axes) Bit 9 = 1 SETVDI (VDI interface signal alarm is set) Bit 13 = 1 FOLLOWUPBYALARM (follow-up)	-	R	r /5
$AN_ESR_TRIGGER	BOOL /5	$AN_ESR_TRIGGER = 1 triggering of "Extended stop and retract"	-	R/W	r/w /5
$AC_ESR_TRIGGER	BOOL /5	$AC_ESR_TRIGGER = 1 Triggering of "NC-driven ESR"	-	R/W	r/w /5
$AC_OPERATING_ TIME	DOUBLE /5	IF $AC_OPERATING_TIME < 12000 GOTOB STARTMARK Total operating time for NC programs in AUTOMATIC mode (in seconds)	-	R	r /5
$AC_CYCLE_TIME	DOUBLE /5	IF $AC_CYCLE_TIME > 2400 GOTOF ALARM01 Operating time of selected NC program (in seconds)	-	R	r /5
$AC_CUTTING_TIME	DOUBLE /5	IF $AC_CUTTING_TIME > 6000 GOTOF ACT_M06 Tool operation time (in seconds)	-	R	r /5
$AC_REQUIRED_ PARTS	DOUBLE /5	$AC_REQUIRED_PARTS = ACTUAL_LOS Definition of number of required workpieces (workpiece setpoint), e.g. for definition of a batch size, day production ...	-	R/W	r/w /5
$AC_TOTAL_PARTS	DOUBLE /5	IF $AC_TOTAL_PARTS > SERVICE_COUNT GOTOF MARK_END Total time for all manufactured workpieces	-	R/W	r/w /5
$AC_ACTUAL_PARTS	DOUBLE /5	IF $AC_ACTUAL_PARTS == 0 GOTOF NEW_RUN Number of workpieces actually produced (actual workpiece value). With $AC_ACTUAL_PARTS == $AC_REQUIRED_PARTS is automatically $AC_ACTUAL_PARTS = 0.	-	R/W	r/w /5
$AC_SPECIAL_PARTS	DOUBLE /5	$AC_SPECIAL_PARTS = R20 Number of workpieces counted acc. to user strategy. Without internal control.	-	R/W	r/w /5
$AC_G0MODE	INT /6	Interpolation behavior with G0 mode 0: G0 active 1: G0 and linear interpolation active 2: G0 and non-linear interpolation active With G0 the behavior of the path axis is in accordance with machine data MD 20730: G0_LINEAR_MODE (Siemens mode) or machine data MD 20732: EXTERN_G0_LINEAR_MODE (ISO mode). With linear interpolation the path axes are traversed together. With non-linear interpolation the path axes are traversed as positioning axes.	-	R	r /6.1
$AC_MEAS_LATCH	DOUBLE /6	$AC_MEAS_LATCH[0] = 1 1. Write actual value to measuring point. 0: corresponds to 1st measuring point, .. , 3: 4th measuring point	Measuring point	R/W	r/w /6.1

© Siemens AG 2002. All rights reserved

Positions

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_IW[X]	DOUBLE	Actual value in workpiece coordinate system (WCS)	Axis	R	r
$AA_IEN[X]	DOUBLE /5	Actual value in settable zero coordinate system (SZS)	Axis	R	r /5
$AA_IBN[X]	DOUBLE /5	Actual value in basic zero coordinate system (BCS).		R	r /5
$AA_IB[X]	DOUBLE	Actual value in basic coordinate system (BCS)	Axis	R	r
$AA_IM[X]	DOUBLE	Actual value in machine coordinate system (MCS)	Axis	R	r

Indexing axes

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_ACT_INDEX_AX_ POS_NO[X]	INT /5	0: No indexing axis, therefore no indexing position available. > 0: Number of last indexing position to be reached or overtraveled		R	r /5
$AA_PROG_INDEX_AX _POS_NO[X]	INT /5	0: No indexing axis, therefore no indexing position available or the indexing axis is not currently approaching an indexing position > 0: Number of the indexing position programmed		R	r /5

Encoder limit frequency

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_ENC_ACTIVE[X]	BOOL /4	Active measuring system is operating below encoder limit frequency (valid values)	Axis	R	r /4
$AA_ENCi_ACTIVE[X]	BOOL /4	i: 1 – 2 encoder number; Measuring system i is operating below encoder limit frequency (valid values)	Axis	R	r /4

© Siemens AG 2002. All rights reserved

Encoder values

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$VA_IM[X]	DOUBLE /4	Encoder actual value in machine coordinate system (measured by active measuring system), actual value compensations are corrected (leadscrew error compensation, backlash compensation, quadrant error compensation). No modulo conversion is performed.	Axis	R	r /4
$VA_IM[X]	DOUBLE /4	Actual value in machine coordinate system (measured by encoder 1), compensations are corrected	Axis	R	r /4
$VA_IM[X]	DOUBLE /4	Actual value in machine coordinate system (measured by encoder 2), compensations are corrected	Axis	R	r /4
$AA_MW[X]	DOUBLE	Measured value in workpiece coordinate system	Axis	R/W	r/w
$AA_MM[X]	DOUBLE	Measured value in machine coordinate system	Axis	R/W	r/w /4
$AA_MWi[X]	DOUBLE /4	Measurement result of axial measurement i: 1-4 for trigger events 1-4	Axis	R/W	r/w

Axial measurement

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_MMi[X]	DOUBLE /4	Measurement result of axial measurement i: 1-4 for trigger events 1-4	Axis	R/W	r/w
$AA_MEAACT[X]	BOOL /4	Value is TRUE (1) if axial measurement is active for axis X	Axis	R	r /4

© Siemens AG 2002. All rights reserved

Offsets

Name	Type /SW	Description/values	In-dex	PP access	SA access /SW
$AC_DRF[X]	DOUBLE	DRF offset	Axis	R	r
$AC_PRESET[X]	DOUBLE	Last Preset value	Axis	R	r
$AA_MEAS_P1_ VALID[X]	INT /6	$AA_MEAS_P1_VALID[X] = 1 Write actual value to 1st measuring point	Axis	R/W	r/w /6.1
$AA_MEAS_P2_ VALID[X]	INT /6	$AA_MEAS_P2_VALID[X] = 1 Write actual value to 2nd measuring point	Axis	R/W	r/w /6.1
$AA_MEAS_P3_ VALID[X]	INT /6	$AA_MEAS_P3_VALID[X] = 1 Write actual value to 3rd measuring point	Axis	R/W	r/w /6.1
$AA_MEAS_P4_ VALID[X]	INT /6	$AA_MEAS_P4_VALID[X] = 1 Write actual value to 4th measuring point	Axis	R/W	r/w /6.1
$AA_OFF[X]	DOUBLE	Overlaid motion for programmed axis	Axis	R/W	r/w
$AA_OFF_LIMIT[X]	INT /4	Limit value for axial compensation $AA_OFF[X] 0: Limit value not reached 1: Limit value reached in positive axis direction -1: Limit value reached in negative axis direction	Axis	R	r
$AA_OFF_VAL[X]	DOUBLE	$AA_OFF_VAL[X] Integrated value of overlaid motion for one axis. An overlaid motion can be canceled using a negative value for this variable, e.g. $AA_OFF[axis] = -$AA_OFF_VAL[axis]	Axis	R	r /5
$AC_RETPOINT[X]	DOUBLE	Return point on contour for repositioning	Axis	R	r /4
$AA_SOFTENDP[X]	DOUBLE	Software limit position, positive direction	Axis	R	r /4
$AA_SOFTENDN[X]	DOUBLE	Software limit position, negative direction	Axis	R	r /4

© Siemens AG 2002. All rights reserved

Axial paths

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_DTBW[X]	DOUBLE	Axial path from start of block in workpiece coordinate system for positioning and synchronized axes during motion-synchronous actions. The programmed position alone determines the path calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.	Axis	R	r
$AA_DTBB	DOUBLE	Axial path from start of block in basic coordinate system for positioning and synchronized axes during motion-synchronous actions. The programmed position alone determines the path calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.	Axis	R	r
$AA_DTEW	DOUBLE	Axial path to end of block in workpiece coordinate system for positioning and synchronized axes during motion-synchronous actions. The programmed position alone determines the path calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.	Axis	R	r
$AA_DTEB	DOUBLE	Axial path to end of block in basic coordinate system for positioning and synchronized axes during motion-synchronous actions. The programmed position alone determines the path calculation. If the axis is a coupled-motion axis, the position component resulting from the axis coupling is ignored.	Axis	R	r

Oscillation

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_DTEPW	DOUBLE	Axial distance-to-go for infeed oscillation in workpiece coordinate system	Axis	R	r
$AA_DTEPB	DOUBLE	Axial distance-to-go for infeed oscillation in basic coordinate system	Axis	R	r
$AA_OSCILL_REVERSE_POS1[X]	DOUBLE	Current reversal position 1 for oscillation In synchronized actions, the value of setting data $SA_OSCILL_REVERSE_POS1 is evaluated online.	Axis	R	r
$AA_OSCILL_REVERSE_POS2[X]	DOUBLE	Current reversal position 2 for oscillation In synchronized actions, the value of setting data $SA_OSCILL_REVERSE_POS2 is evaluated online.	Axis	R	r

$AA_DELT	DOUBLE	Axial distance-to-go in workpiece coordinate system after axial deletion of distance-to-go for motion-synchronous actions	Axis	R	r

© Siemens AG 2002. All rights reserved

Velocities, axis-specific

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_OVR[X]	DOUBLE	Axial override for motion-synchronous actions. Multiplicative override component, acts in addition to operator override, programmed override and transformation override, the total factor is limited to 200%. It must be written again in every interpolation cycle, otherwise the value 100% applies. $AA_OVR[S1] changes the spindle override. The override defined by the machine data MD 12100: OVR_FACTOR_LIMIT_BIN, MD 12030: OVR_FACTOR_FEEDRATE[30], MD 12010: OVR_FACTOR_AX_[30], $AA_OVR_FACTOR_SPIND_SPEED is not exceeded.	Axis	R/W	r/w /4
$AA_VC[X]	DOUBLE	Additive axial feedrate override for motion-synchronous actions. The compensation value must be written again in every interpolator cycle, otherwise the value 0 applies. An override of 0 cancels the compensation value; otherwise the override has no effect on the compensation value. The total feedrate cannot be negative as a result of the compensation value. The upper value is limited such that the maximum axis velocities and accelerations are not exceeded. The calculation of other feedrate components is not affected by $AA_VC. The override values defined by the machine data MD 12100: OVR_FACTOR_LIMIT_BIN, MD 12030: OVR_FACTOR_FEEDRATE[30], MD 12010: OVR_FACTOR_AX_SPEED[30], MD 12070: OVR_FACTOR_SPIND_SPEED are not exceeded. The additive feedrate override is limited such that the resulting feedrate does not exceed the maximum override value of the programmed feedrate.	Axis	R/W	r/w /4
$AA_VACTB[X]	DOUBLE	Axis speed in basic coordinate system. The variable can only be accessed from synchronized actions.	Axis	R	r
$AA_VACTW[X]	DOUBLE	Axis speed in workpiece coordinate system. The variable can only be accessed from synchronized actions.	Axis	R	r
$AA_VACTM[X]	DOUBLE	Axis velocity setpoint in machine coordinate system Can also be read by replacement/PLC axes The variable can only be accessed from synchronized actions.	Axis	R	r /4
$VA_VACTM[X]	DOUBLE	Axis velocity actual value in machine coordinate system. The variable returns an undefined value if the encoder limit frequency is exceeded.	Axis	R	r /4

© Siemens AG 2002. All rights reserved

Drive data

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_LOAD[X]	DOUBLE	Drive load in % (for 611D only)	Axis	R	r
$VA_LOAD[X]	DOUBLE	Drive load in % (for 611D only)	Axis	R	r /5
$AA_TORQUE[X]	DOUBLE	Drive torque setpoint in Nm (with 611D only) Actual force in N (with 611D HLA only)	Axis	R	r
$VA_TORQUE[X]	DOUBLE	Drive torque setpoint in Nm (with 611D only) Actual force in N (with 611D HLA only)	Axis	R	r /5
$AA_POWER[X]	DOUBLE	Active drive power in W (for 611D only)	Axis	R	r
$VA_POWER[X]	DOUBLE	Active drive power in W (for 611D only)	Axis	R	r /5
$AA_CURR[X]	DOUBLE	Actual current value for axis or spindle (for 611D only)	Axis	R	r
$VA_CURR[X]	DOUBLE	Actual current value for axis or spindle (for 611D only)	Axis	R	r /5
$VA_VAVELIFT[X]	DOUBLE	Actual valve stroke in mm (with 611D hydraulic system only)	Axis	R	r /5
$VA_PRESSURE_A[X]	DOUBLE	Pressure on drive end of cylinder in bar (with 611D hydraulic system only)	Axis	R	r /5
$VA_PRESSURE_B[X]	DOUBLE	Pressure on non-drive end of cylinder in bar (with 611D hydraulic system only)	Axis	R	r /5

Axis states

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_STAT[X]	INT /4	Axis status: 0: No axis status available 1: Traversing movement active 2: Axis has reached end of interpolation, only for axes of channel 3: Axis in position (exact stop coarse) for all axes 4: Axis in position (exact stop fine) for all axes	Axis	R	r /4
$AA_REF[X]	INT /5	Axis status: 0: Axis is not referenced 1: Axis is referenced		R	r /5
$AA_TYP[X]	INT /4	Axis type: 0: Axis in another channel 1: Channel axis of the same channel 2: Neutral axis 3: PLC axis 4: Oscillating axis 5: Neutral axis currently traversing in JOG mode 6: Master value-coupled following axis 7: Coupled-motion following axis 8: Command axis 9: Compile cycle axis	Axis	R	r /4
$AA_MASL_STAT[X]	INT 6	Current status if a master-slave coupling $AA_MASL_STAT set to = 0: Axis is not a slave axis or no coupling is active. > 0: Coupling active supplies the associated machine axis number of the master axis.	Slave axis	R	r /6

© Siemens AG 2002. All rights reserved

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_FXS[X]	INT 5	Status “travel to fixed stop” 0: Axis not at stop 1: Stop approached successfully 2: Fixed stop approach unsuccessful 3: Selection of travel to fixed stop active 4: Fixed stop was detected 5: Deselection of travel to fixed stop active	Axis	R/W	r /5
$VA_TORQUE_AT_ LIMIT[X]	INT 5	Status ”Effective torque corresponds to the specified torque limit” 0: Torque limit not yet reached 1: Torque limit reached In digital 611D systems the drive returns a status, indicating whether the programmed torque limit has been reached.	Axis	R	r /5
$AA_FOC[X]	INT 5	Status of ”ForceControl” (FOC) function 0: FOC not active 1: FOC modally active 2: FOC non-modally active	Axis	R/W	r /5
$AA_COUP_ ACT[SPI(2)]	INT /4	Current coupling status of following spindle/axis 0: Axis/sp. not coupled with a LS/LA 3: Axis is tangentially corrected 4: Synchronous spindle coupling 8: Coupled-motion following axis 16: Master value-coupled following axis The respective values are applicable for one coupling. If the following axis has more active couplings, they are indicated by the sum of the respective numerical values.	Following spindle/ axis	R	r /5

Electronic gear 1

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_EG_SYNFA[a]	DOUBLE /5	Synchronized position of following axis a: Following axis	Following axis	R	r /5
$AA_EG_NUM_LA[a]	INT /5	Number of leading axes specified with EGDEF a: Following axis	Following axis	R	r /5
$AA_EG_SYNCDIFF[a]	DOUBLE /5	Synchronization difference a: Following axis	Following axis	R	r /5
$AA_EG_AX[n,a]	AXIS /5	Identifier for the nth leading axis n: Index for leading axis a: Following axis	Following axis	R	r /5

© Siemens AG 2002. All rights reserved

Master value coupling

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_LEAD_SP[MV]	DOUBLE /4	Simulated master value - position	Master value	R/W	r/w
$AA_LEAD_SV[MV]	DOUBLE /4	Simulated master value - velocity	Master value	R/W	r/w
$AA_LEAD_P_ TURN[MV]	DOUBLE /4	Current master value - position component lost as a result of modulo reduction. The actual master value position (used for internal calculations by the controller) is $AA_LEAD_P[MV] + $AA_LEAD_P_TURN[MV]. If MV is a modulo axis, $AA_LEAD_P_TURN is a whole multiple of $MA_MODULO_RANGE. If MV is not a modulo axis, $AA_LEAD_P_TURN is always 0. Example 1: $MA_MODULO_RANGE[MV] = 360 $AA_LEAD_P[MV] = 290 $AA_LEAD_P_TURN[MV] = 720 The actual master value position (used for internal calculations by the controller) is 1010. Example 2: $MA_MODULO_RANGE[MV] =360 $AA_LEAD_P[MV] = 290 $AA_LEAD_P_TURN[MV] =-360 The actual master value position (used for internal calculations by the controller) is -70.	Master value	R	r /4
$AA_LEAD_P[MV]	DOUBLE /4	Current master value - position (modulo reduced) If master value (MV) is a modulo axis, the following is always true: 0 <= $AA_LEAD_P[MV] <= $MA_MODULO_RANGE[MV]	Master value	R	r /4
$AA_LEAD_V[MV]	DOUBLE /4	Actual master value - velocity	Master value	R	r /4
$AA_SYNC[FA]	INT /4	Coupling status of following axis in master value coupling 0: No synchronism 1: Coarse synchronism 2: Fine synchronism 3: Coarse and fine	Following axis	R	r /4

Synchronous spindle

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$P_COUP_OFFS[S2]	DOUBLE	Programmed offset of the synchronous spindle	Following spindle	R	r /6
$AA_COUP_OFFS[S2]	DOUBLE /2	Position offset for synchronous spindle (setpoint)	Following spindle	R	r /4
$VA_COUP_ OFFS[SPI(2)]	DOUBLE /2	Position offset for synchronous spindle (actual value)	Following spindle	R	r /4

© Siemens AG 2002. All rights reserved

Safety Integrated 1

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$VA_IS[X]	DOUBLE /3	Safe actual position (SISITEC)	Axis	R	r /4
$AA_SCTRACE[X]	BOOL /4	$AA_SCTRACE[X] = 1 Write: Activate IPO trigger for servo trace. 0: No action 1: Activate trigger Read: Always value 0, as the self-clearing trigger bit is read back from the interface. 0: Current value (no status)	Axis	R/W	r /4
$VA_DPE[X1]	BOOL /5	Status performance release of a machine axis (611D and 611D hydraulic system) FALSE: No performance release TRUE: Performance release	Machine axis	R/	r /5
$AA_ACC	DOUBLE /5	Current acceleration value of axis for single-axis interpolation. $AA_ACC = $MA_MAX_AX_ACCEL * progr. acceleration override	Axis	R/	r /5
$AA_MOTEND	INT /5	Current end of movement condition for single-axis interpolation. 1: End of movement on exact stop FINE 2: End of movement on exact stop COARSE 3: End of movement on exact stop interpolator stop 4: Block change in braking ramp of axis motion	Axis	R/	r /5 r /6
$AA_SCPAR	INT /5	Read current servo parameter set	Axis	R/	r /5

Extended stop and retract

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_ESR_STAT[X]	INT /5	Status of “Extended stop and retract”, bit-coded: BIT0: Generator mode is triggered BIT1: Retraction is triggered BIT2: Ext. stop is triggered BIT3: DC link undervoltage BIT4: Generator minimum speed		R/	r/
$AA_ESR_ENABLE[X]	BOOL /5	$AA_ESR_ENABLE[X] = 1 Enable “Extended stop and retract”		R/W	r/w

© Siemens AG 2002. All rights reserved

Axis container rotation

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AN_AXCTSWA[n]	BOOL /5	Axis container rotation active 1: An axis container rotation is currently being performed on axis container name n 0: No axis container rotation is currently active	Axis container	r/	r/
$AN_AXCTAS[n]	INT /5	Axis container rotation, current degree of rotation. This specifies for the axis container name n how many slots the axis container has advanced. The value range is from 0 to the maximum number of assigned slots in the axis container -1	Axis container	r/	r/
$AC_AXCTSWA[n]	BOOL /5	Enable axis container rotation 1: The channel has enabled axis container rotation for axis container name n and the rotation is still in progress. 0: The axis container rotation has finished.	Axis container	r/	r/

Electronic gear 2

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$AA_EG_TYPE	INT /5	$AA_EG_TYPE[a,b] a: Following axis b: Leading axis Type of coupling for leading axis b 0: Actual value coupling 1: Setpoint value coupling	Axis	R	r /5
$AA_EG_NUMERA	DOUBLE /5	$AA_EG_NUMERA[a,b] a: Following axis b: Leading axis Numerator for coupling factor for leading axis b	Axis	R	r /5
$AA_EG_DENOM	DOUBLE /5	$AA_EG_DENOM[a,b] a: Following axis b: Leading axis Denominator for coupling factor for leading axis b	Axis	R	r /5
$AA_EG_SYN	DOUBLE /5	$AA_EG_SYN[a,b] a: Following axis b: Leading axis Synchronized position of leading axis b	Axis	R	r /5
$AA_EG_ACTIVE	BOOL /5	$AA_EG_ACTIVE[a,b] a: Following axis b: Leading axis Coupling for leading axis b is active, i.e. switched on	Axis	R	r /5

© Siemens AG 2002. All rights reserved

Safety Integrated (S. I.)

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_INSE[n]	BOOL /4	Image of a safety input signal (ext. NCK interface)	No. of input	R	r
$A_INSED[n]	INT /5	Image of safety input signals (ext. NCK interface)	Number of input word 1 – ...	R/	r/
$A_INSEP[n]	BOOL /5	Image of a safety input signal (ext. PLC interface)	Number of input 1 – ...	R/	r/
$A_INSEPD[n]	INT /5	Image of safety input signals (ext. PLC interface)	Number of input word 0 – ...	R/	r/
$A_OUTSE[n]	BOOL /5	Image of a safety output signal (ext. NCK interface)	Number of output 1 – ...	R/W	r/w
$A_OUTSED[n]	INT /5	Image of safety output signals (ext. NCK interface)	Number of output word 1 – ...	R/W	r/w
$A_OUTSEP[n]	BOOL /4	Image of a safety output signal (ext. PLC interface)	Number of output 1 – ...	R	r/
$A_OUTSEPD[n]	INT /5	Image of safety output signals (ext. PLC interface)	Number of output word 0 – ...	R	r/

S. I. : Servo interpolator interface

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_INSI[n]	BOOL /4	Image of a safety input signal (int. NCK interface)	Number of input 1 – ...	R	r/
$A_INSID[n]	INT /5	Image of safety input signals (int. NCK interface)	Number of input word 1 – ...	R	r/
$A_INSIP[n]	BOOL /4	Image of a safety input signal (int. PLC interface)	Number of input word 1 – ...	R	r/

© Siemens AG 2002. All rights reserved

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_INSIPD[n]	INT /5	Image of safety input signals (int. PLC interface)	Number of input word 1 - ...	R	r/
$A_OUTSI[n]	BOOL /4	Image of a safety output signal (int. NCK interface)	Number of output 1 - ...	R/W	r/w
$A_OUTSID[n]	INT /5	Image of safety output signals (int. NCK interface)	Number of output word 1 - ...	R/W	r/w
$A_OUTSIP[n]	BOOL /4	Image of a safety output signal (int. PLC interface)	Number of output 1 - ...	R	r/
$A_OUTSIPD[n]	INT /5	Image of safety output signals (int. PLC interface)	Number of output word 1 - ...	R	r/

Safety markers and timers

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_MARKERSI[n]	BOOL /4	Markers for Safety Integrated programming	No. of marker 1 ...	R/W	r/w
$A_MARKER-SID[n]	INT /5	Flag word (32-bit) for safety programming	No. of flag word 1 ...	R/W	r/w
$A_MARKER-SIP[n]	BOOL /4	Image of PLC Safety Integrated markers	No. of marker 1 ...	R	r/
$A_MARKER-SIPD[n]	INT /5	Image of PLC Safety Integrated flag words	No. of flag word 1 ...	R	r/
$A_TIMERSI[n]	DOUBLE /4	Safety timer unit in seconds. The time is counted internally in multiples of the interpolation cycle. Incrementation of the timer variable is started by assigning $A_TIMERSI[n]=<start value>. Incrementation is stopped by assigning a negative value: $A_TIMERSI[n]=-1. The current time value can be read whether the timer variable is running or halted. After a timer variable has been stopped through the assignment of -1, the current time value remains stored and can be read.	No. of timer 1 ...	R/W	r/w

© Siemens AG 2002. All rights reserved

Safety cross-checking control and state variable

Name	Type /SW	Description/values	Index	PP access	SA access /SW
$A_STATSID	INT /5	Safety: Status of cross-checking between NCK and PLC. If the value is not equal to zero, an error has occurred during cross-checking	-	R/	r/
$A_CMDSI[n]	BOOL /5	Safety: Control word for cross-checking between NCK and PLC. Array index n = 1: Increase time for signal change monitoring to 10 s	Number of control signal	R/W	r/w
$A_LEVELSID	INT /5	Safety: Displays level of signal change monitoring. Specifies the number of signals currently tagged for cross-checking.	-	R	r/

© Siemens AG 2002. All rights reserved

2.4 Actions in synchronized actions

Actions

After action code **DO ...**, each synchronized action contains

- one or several (max. 16) actions or a technology cycle
  (these two components are referred to generally as **actions** in the following description).

These are executed when the appropriate condition is fulfilled.

Several actions

Several actions contained in a synchronized action are activated in the same interpolation cycle if the appropriate condition is fulfilled.

List of possible actions

The following actions can be programmed in the “Action” section of synchronized actions:

Table 2-2 Actions in synchronized actions

... DO ...	Meaning	Reference
Mxx Sxx Hxx	Output of auxiliary functions to PLC	2.4.1
SETAL(nr)	Set alarm, error reactions	2.4.20
$A...= ... $V... = ... $AA_OFF = $AC_OVR = $AA_OVR = $AC_VC = $AA_VC = $$SN_SW_CAM_ ... $AC_FCT...	Write real-time variables: - Overlaid motion - Velocity control: Path velocity Axis velocity Add. path feedrate override Add. axis compensation value Alter SW cam positions (setting data) and all other SD Overwrite FCTDEF parameters	2.4.2 2.4.3 2.4.4
 RDISABLE STOPREOF DELDTG FTOC SYNFCT ZYKL_T1 (e.g.)	synchronized action procedures: Activate read-in disable End feed stop Delete distance-to-go Online tool offset Polynomial evaluation Call of technology cycles	 2.4.8 2.4.9 2.4.10 2.4.7 2.4.5 2.5
 $AA_OVR[x]= 0 AXIS_X (e.g.) POS[u]= ... FA[u]= ... MOV[u]= >0 MOV[u] = <0 MOV[u] = =0	Control positioning axes: Disable an axis motion Call an axis program Position Define axis feedrate Move command axes continuously: - forwards - backwards - stop	 2.4.11 2.4.12 2.4.12 2.4.13 2.4.14 “ “ “
 SPOS M3, M4, M5, S = $AA_OVR[S1]= 0	Spindles: Position Direction of rotation, stop, speed Disable spindle motion	 2.4.15
PRESETON(,)	Set actual values	2.4.16

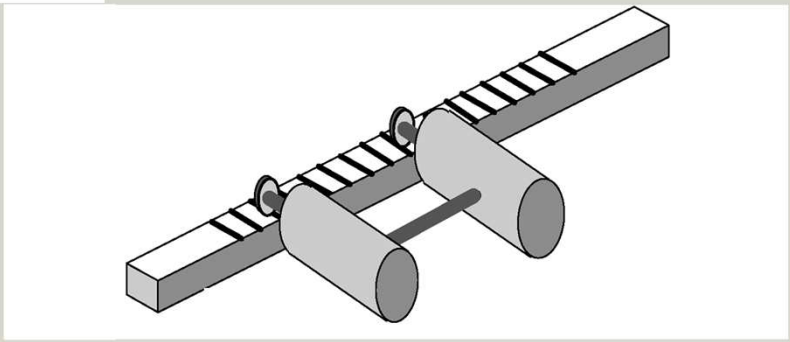
© Siemens AG 2002. All rights reserved

Table 2-2 Actions in synchronized actions

... DO ...	Meaning	Reference
LEADON LEADOF TRAILON TRAILOF	Activate/deactivate couplings: Couple slave axis to master axis Cancel coupling Asynchr. coupled motion ON Asynchr. coupled motion OFF	2.4.17
MEAWA, MEAC	Measurement without deletion of distance-to-go Cyclical measurement	2.4.18
SETM CLEARM	Channel synchronization: Set a wait marker Clear a wait marker	2.4.19
LOCK UNLOCK RESET	Coordination of synchronized actions: - Disable synchronized action/technology cycle - Enable synchronized action/technology cycle - Reset technology cycle	2.5.1

© Siemens AG 2002. All rights reserved

SIEMENS



Chapter 15

Axis couplings

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Overview 3
Axis coupling: Options 4
Coupled motion (FB M3) 5
Gantry 6
Synchronous spindle 12
Master-slave coupling 16
Appendix 20
Exercise: 22

Learning objectives

- You will become familiar with certain axis couplings and their application
- You will be able to commission certain axis couplings
- You will be able to configure a software cam



SITRAIN © Siemens AG 2017

Overview

Tangential control	**➢ Align rotary axis on the path tangent**
▪ Synchronous spindle/multi-edge turning	**➢ Speed/positional synchronism with speed ratio**
▪ Coupled motion	**➢ Setpoint coupling of 2 axes**
▪ Pair of synchronized axes (gantry axes)	**➢ 2 mechanically coupled axes Coupling factor 1:1**
▪ Master-slave for drives	**➢ Speed/torque coupling**
▪ Master value coupling and curve tables	**➢ Coupling via curve tables in the NC program**
▪ Electronic gear stage 1 Linear coupling	**➢ 1 following axis on max. 5 leading axes**
▪ Electronic gear stage 2 Non-linear couplings	

SITRAIN © Siemens AG 2017

SIEMENS

Axis coupling: Options

The axis couplings are available in a basic version as part of the NCK software and the four optional versions CP_STATIC, CP-BASIC, CP-COMFORT, and CP-EXPERT.

These couplings are based on what is known as the "generic coupling" (see Appendix) and the Function Manuals, chapter M3.

Type	CP versions allow one or more different CPSETTYPE coupling objects simultaneously:	Base version	CP-STATIC	CP-BASIC	CP-COMFORT	CP-EXPERT
A	Coupled motion	4	-	4	4	8
B	Synchronized spindle with 1:1 coupling	-	1	-	-	-
C	o./u. Synchronous spindle/multi-edge turning o./u. Master value coupling / curve tables interpolation o./u. MCS coupling	-	-	1	4	8
D	o./u. Electronic gearbox "plain" o./u. Free generic coupling "plain"	-	-	-	1	8
E	o./u. Electronic gearbox o./u. Free generic coupling	-	-	-	-	5

o./u. stands for over/under

SITRAIN © Siemens AG 2017

Notes

Coupled motion (FB M3)

Activate:
TRAILON(coupled-motion axis, leading axis, coupling factor)

Deactivating:
TRAILOF(coupled motion axis, leading axis)

Example:
The X axis is the leading axis of coupled-motion axes Y and Z.
The Y axis is to travel 2.5 times the distance of leading axis X.
The Z and X axis distances are to be identical!

```
%_N_MITSCHLEPP_MPF
        :
TRAILON(Y, X, 2.5)
TRAILON(Z, X)
        :

TRAILOF(Y, X)
TRAILOF(Z, X)
M30
```


SITRAIN © Siemens AG 2017

Function

Where 2 or more axes are meant to travel in relation to one another with a fixed distance ratio, this can be achieved using coupled-motion axes.

Each coupled axis grouping is made up of 2 axes (one leading axis and one coupled-motion axis). If the coupled-motion unit is to consist of more than 2 axes, several groupings are defined using the same leading axis every time.

The coupled axis grouping is defined purely by means of the program command in the NC part program. It is not necessary to provide a definition via machine or setting data.

There is no limit within the control as far as the number of coupled axis groupings is concerned. The maximum number of coupled axis groupings is determined solely by the number of machine axes and their permissible combinations. (One coupled-motion axis cannot be activated in more than 2 coupled axis groupings at the same time.)

Coupling factor

The distance ratio between the leading axis and the coupling axis in a coupled axis grouping is determined by the coupling factor programming.

$$\textbf{Coupling factor} = \frac{\textbf{Coupled-motion-axis distance}}{\textbf{Leading axis distance}}$$

If no coupling factor is programmed, the distance ratio is 1:1.

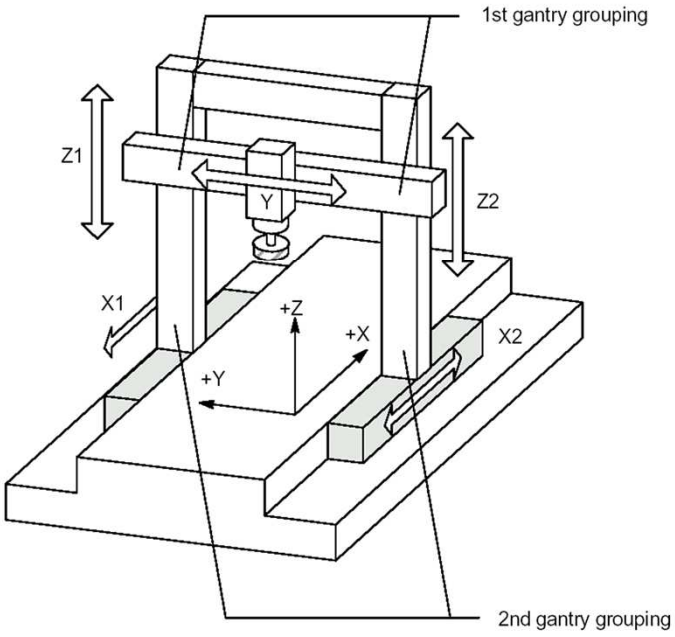
Coupling status

The coupling status of an axis is stored in the **$AA_COUP_ACT[axis identifier]** system variable and can be queried in the NC part program. For the coupled motion coupling, query bit 3.

Example:
IF $AA_COUP_ACT[X]==8 GOTOB LABEL_1

Gantry
Option M02

SIEMENS



Course NC-84SL-SIP Page 15 - 6 SITRAIN © Siemens AG 2017

Function

The term "gantry axes" denotes axis groupings whose individual axes feature a rigid mechanical coupling. Consequently, they are not permitted (or able) to perform substantial single movements under any circumstances, except when eliminating a mechanical misalignment.

Within the NC, gantry axes essentially consist of a setpoint-linked coupled axis grouping whose coupling is always active and whose actual position values are compared and monitored. This function is an option.

Gantry grouping

The SINUMERIK 840Dsl supports a maximum of 8 gantry groupings.

Each gantry grouping is made up of 1 guide axis (which is available to the operator/programmer and can, therefore, be controlled) and one or two synchronous axes, which are not available to the operator/programmer.

Switching on the machine

Before switching on the machine, or directly afterwards, the operator must check whether the mechanical misalignment of the gantry axes is permissible from the point of view of allowing the coupled grouping to travel (reference point approach).

a) Misalignment cannot be tolerated

The operator must implement suitable measures to reduce the misalignment to a tolerable dimension. This can be achieved using a purely mechanical method that involves moving the gantry axes in relation to one another with the drives switched off or carefully controlling the axis drives with the gantry coupling open (MD 37140=1) until the misalignment has been practically eliminated.

Mechanical markers on the axes to indicate the synchronous position are extremely useful during these adjustments.

b) Misalignment can be tolerated

You can initiate the reference point approach procedure for the gantry grouping.

Gantry
Axis machine data

SIEMENS

Axis type/grouping no.
MD 37100 GANTRY_AXIS_TYPE = <axis type, grouping no.>

Actual value difference
MD 37110 GANTRY_POS_TOL_WARNING = <actual value difference>

Trip limit after synchronizing the gantry grouping
MD 37120 GANTRY_POS_TOL_ERROR = <trip difference>

Trip limit before synchronizing the gantry grouping
MD 37130 GANTRY_POS_TOL_REF = <trip difference>

Forced coupling
MD 37140 GANTRY_BREAK_UP = <forced coupling OFF/ON>

SITRAIN © Siemens AG 2017

Axis type/grouping No. MD 37100 GANTRY_AXIS_TYPE = <axis type, grouping no.>

Axis type (tens digit)
0 Guide axis
1 Synchronous axis

Grouping no. (Units digit)
0 No gantry axis
1 Axis grouping 1
2 Axis grouping 2
3 Axis grouping 3
etc.

Actual value difference MD 37110 GANTRY_POS_TOL_WARNING = <actual value difference>

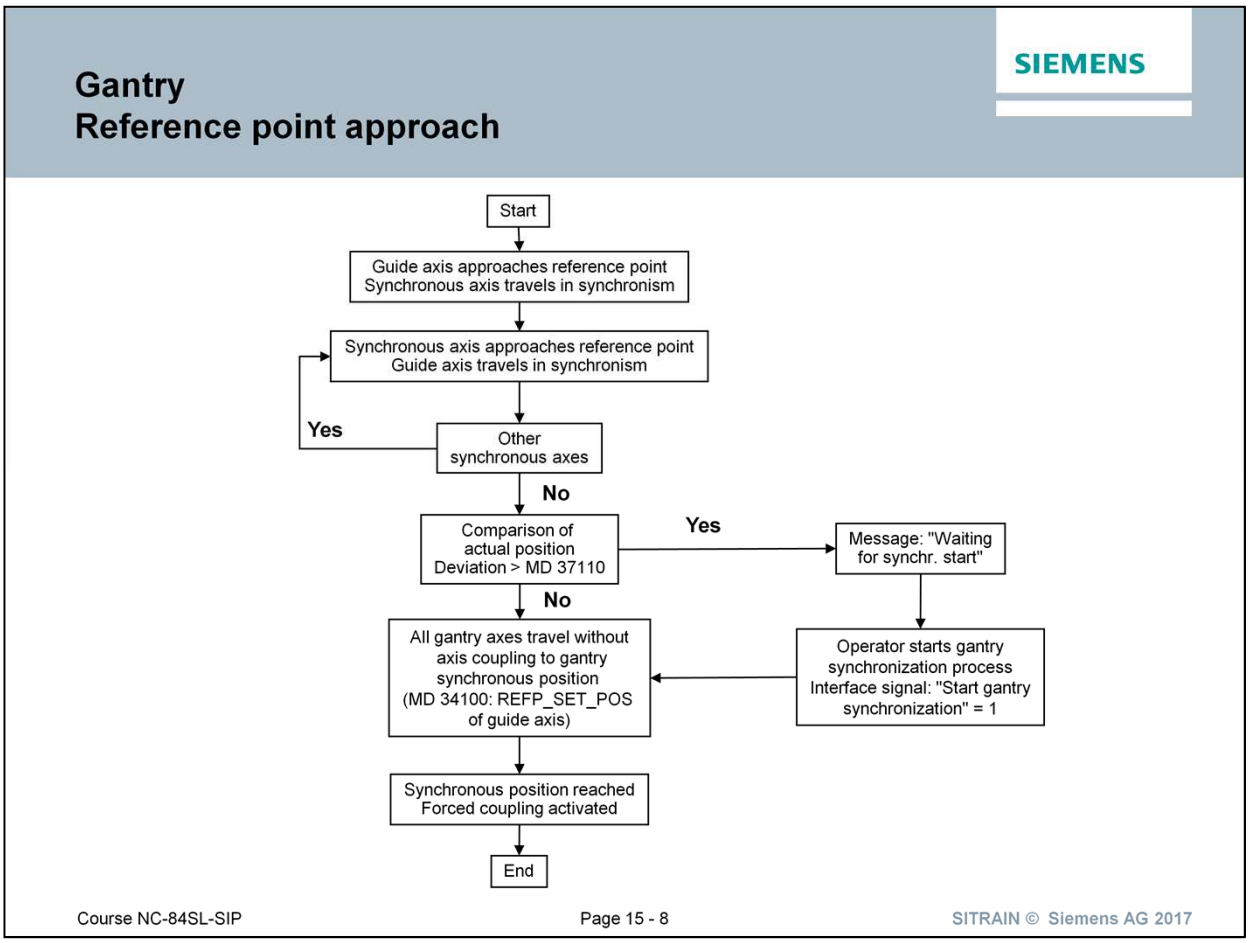
If this value is exceeded, alarm 10 652 "Warning limit exceeded" is output and interface signal "Gantry warning limit exceeded" (DB 31 - 61, DBX 101.3) is set. The messages are reset automatically when the value drops below the tolerance limit again.

Trip difference MD 37120 GANTRY_POS_TOL_ERROR = <trip difference>
Trip limit after synchronizing the gantry grouping.
If this value is exceeded, alarm 10 653 "Error limit exceeded" is output and interface signal "Gantry trip limit exceeded" (DB 31 - 61, DBX 101.2) is set. The gantry axes are shut down immediately within the NC.

Trip difference MD 37130 GANTRY_POS_TOL_ERROR = <trip difference>
Trip limit prior to synchronizing the gantry grouping. Same response as for MD 37120.

Forced coupling MD 37140 GANTRY_BREAK_UP = <forced coupling OFF/ON>

0 Gantry forced coupling and monitoring remain activated
1 Gantry grouping is ungrouped. The axes can be traversed individually.



Reference point approach

If gantry axes are fitted with incremental measuring systems, the actual position in the NC must be aligned with the true mechanical actual position after switch-on, i.e. the gantry must be referenced. The gantry axes are referenced one after the other (first the guide axis and then the synchronous axes). However, the gantry coupling remains activated so that all the axes in a grouping always travel at the same time. The referencing sequence is the same as for individual axes.

Guide axis

Guide axis referencing (the synchronous axis travels at the same time).
The referencing sequence is similar to that for a standard axis.

Synchronous axis

Once the guide axis reference point has been reached, the referencing sequence for the synchronous axis starts automatically (guide axis follows).

Once the synchronous axis end point has been reached, referencing is complete; the control actual values now match the mechanical positions. It is now also possible to see the extent to which the gantry axes deviate from their synchronous position.

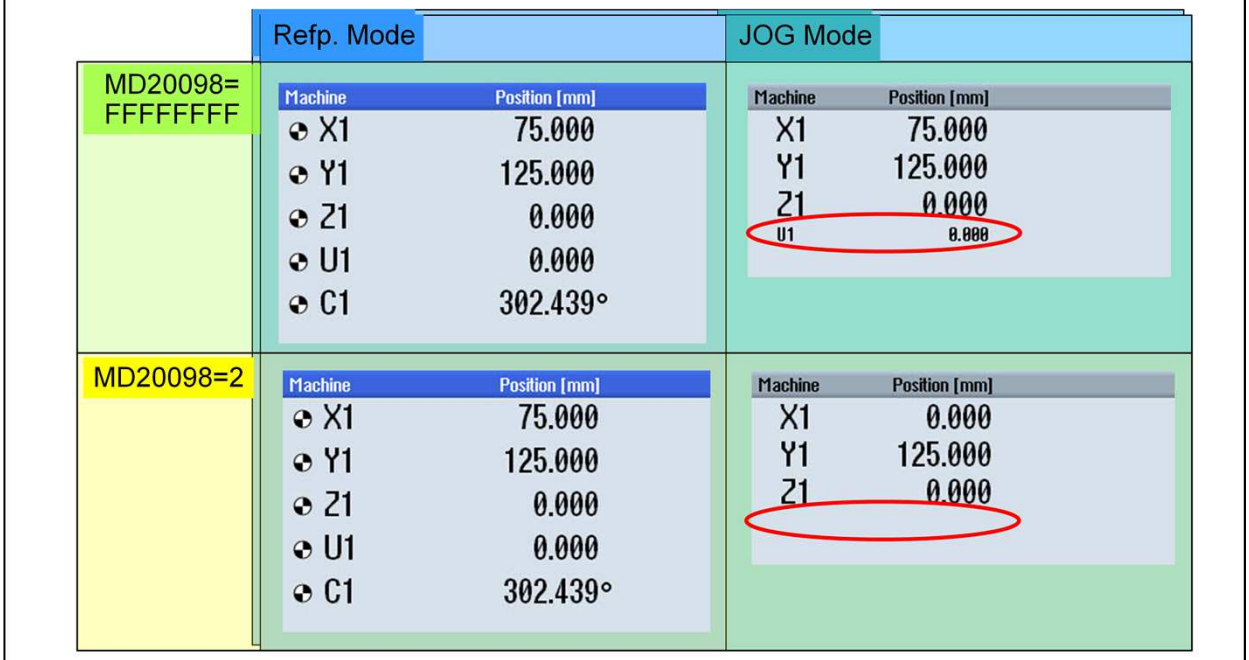
Synchronism

For each synchronous axis in the gantry grouping, the difference in position from the current value of the guide axis is determined and compared with the tolerance warning limit (MD 37110: GANTRY_POS_TOL_WARNING).

a) The differences are lower than the warning limits:
all gantry axes travel to the guide axis reference point value without coupling (all actual values are now identical.) After this, the gantry axis coupling is reactivated.

b) One difference is higher than the warning limit:
The synchronization is stopped. Interface signal DBxx.DBX101.4 "Gantry synchronization ready to start" is set by the NC for the guide axis. The NC waits for interface signal DBxx.DBX29.4 "Start gantry synchronization".
A 0/1 signal edge results in the process being continued and completed as described under a).

Gantry
Hiding the synchronous axes



 SITRAIN © Siemens AG 2017

Hiding

After the search for the reference, the synchronous axes are no longer available to the user. MD 20098 DISPLAY_AXIS can be used to hide axes so that they disappear from the display.

MD 20098

Bits 0 to 15: Display in the MCS

Bit 0=1	Display axis in the actual value windows
Bit 1=1	Display axis in the actual value window ref point
Bit 2=1	Display axis in preset/basic offset/scratching window
Bit 3=1	Machine axis in the handwheel selection window

Bit 16 to 31: Display in the WCS

Bit 16=1	Display axis in the actual value windows
Bit 18=1	Display axis in the zero offsets
Bit 19=1	Display axis in the handwheel selection window
Bit 20=1	Display axis in the JOG/manual windows

If the the relevant bit = 0, the axis there is hidden

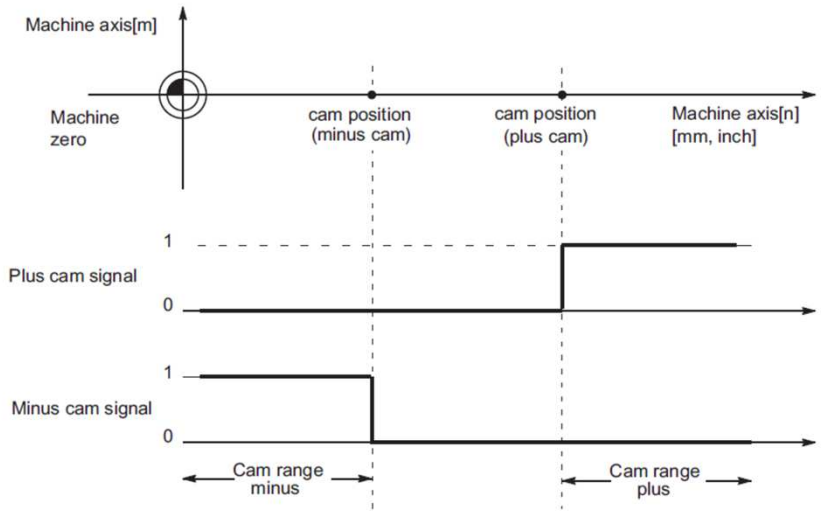
Index

The MD index represents the number of the channel axis – 1.

Example

If MD 20098 DISPLAY_AXIS[4] = 2, channel axis 5 is only displayed during reference point approach.

Gantry
Software cam controller (for gantry exercise)



Course NC-84SL-SIP Page 15 - 10 SITRAIN © Siemens AG 2017

Function The "Software cams" function allows position-dependent cam signals to be output to the PLC or NCK I/Os. With the **MD 10450 [0-31]**, a machine axis can be assigned to each of the 32 possible cam pairs. If you enter a "0", the relevant cam will be ignored.

Activation The cam signal output is activated via the axial IS/PLC interface signal **DB31, ... DBX2.0** (cam activation). The "Software cams" function can be activated and used in all operating modes. The function remains active in the event of RESET or EMERGENCY STOP.

Cam position The cam positions at which signal outputs are set can be defined and altered via setting data. The setting data can be read and written via the HMI, PLC, and part program.

Example MD 10450 $MN_SW_CAM_ASSIGN_TAB[0]= 3 addresses cam pair 1 on the 3rd machine axis.

Gantry
Software cam controller (for gantry exercise)

SIEMENS

SD41500 SW_CAM_MINUS_POS_TAB_1[n]	Position of minus cams 1-8
SD41500 SW_CAM_PLUS_POS_TAB_1[n]	Position of plus cams 1-8
SD41500 SW_CAM_MINUS_POS_TAB_2[n]	Position of minus cams 9-16
SD41500 SW_CAM_PLUS_POS_TAB_2[n]	Position of plus cams 9-16
SD41500 SW_CAM_MINUS_POS_TAB_3[n]	Position of minus cams 17-24
SD41500 SW_CAM_PLUS_POS_TAB_3[n]	Position of plus cams 17-24
SD41500 SW_CAM_MINUS_POS_TAB_4[n]	Position of minus cams 25-32
SD41500 SW_CAM_PLUS_POS_TAB_4[n]	Position of plus cams 25-32

DB10	Signals from the NC (NC → PLC)							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB110	Software cam minus /Z2-N3/							
	7	8	5	4	3	2	1	0
DBB111	Software cam minus /Z2-N3/							
	15	14	13	12	11	10	9	8
DBB112	Software cam minus /Z2-N3/							
	23	22	21	20	19	18	17	16
DBB113	Software cam minus /Z2-N3/							
	31	30	29	28	27	26	25	24
DBB114	Software cam plus /Z2-N3/							
	7	6	5	4	3	2	1	0
DBB115	Software cam plus /Z2-N3/							
	15	14	13	12	11	10	9	8
DBB116	Software cam plus /Z2-N3/							
	23	22	21	20	19	18	17	16
DBB117	Software cam plus /Z2-N3/							
	31	30	29	28	27	26	25	24

Course NC-84SL-SIP Page 15 - 11 SITRAIN © Siemens AG 2017

Cam positions

The positions of the plus and minus cams are defined using general setting data, e.g.

SD 41500 SW_CAM_MINUS_POS_TAB_1[n] position minus cams 1 – 8
SD41501 SW_CAM_PLUS_POS_TAB_1[n] position plus cams 1 – 8

Note

Due to the grouping of cam pairs into groups of eight, it is possible to assign different access authorization levels (e.g. for machine-related and workpiece-related cam positions). The positions are entered in the machine coordinate system. No check is made with respect to the maximum traversing range.

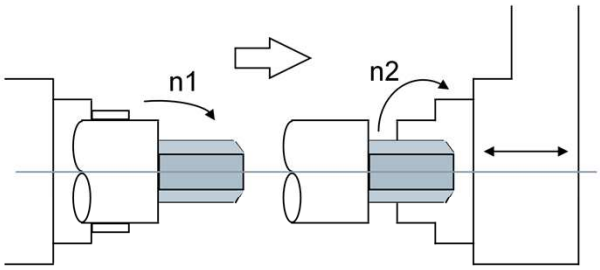
What is the software cam for in the gantry exercise?

Given that the training rack does not include a hardware reduction cam for the X axis, a Y axis software cam (absolute encoder) is used in the "Cam controller" exercise as a reduction cam for X.

Synchronous spindle
Option M14

SIEMENS

Leading spindle Following spindle



Coupling types:

- Setpoint coupling
- Actual value coupling
- Velocity coupling

The pairs of synchronous spindles can either be permanently configured using machine data (1 pair of spindles/channel) or can be freely defined by the user in the NC part program with NC commands.

 SITRAIN © Siemens AG 2017

Function

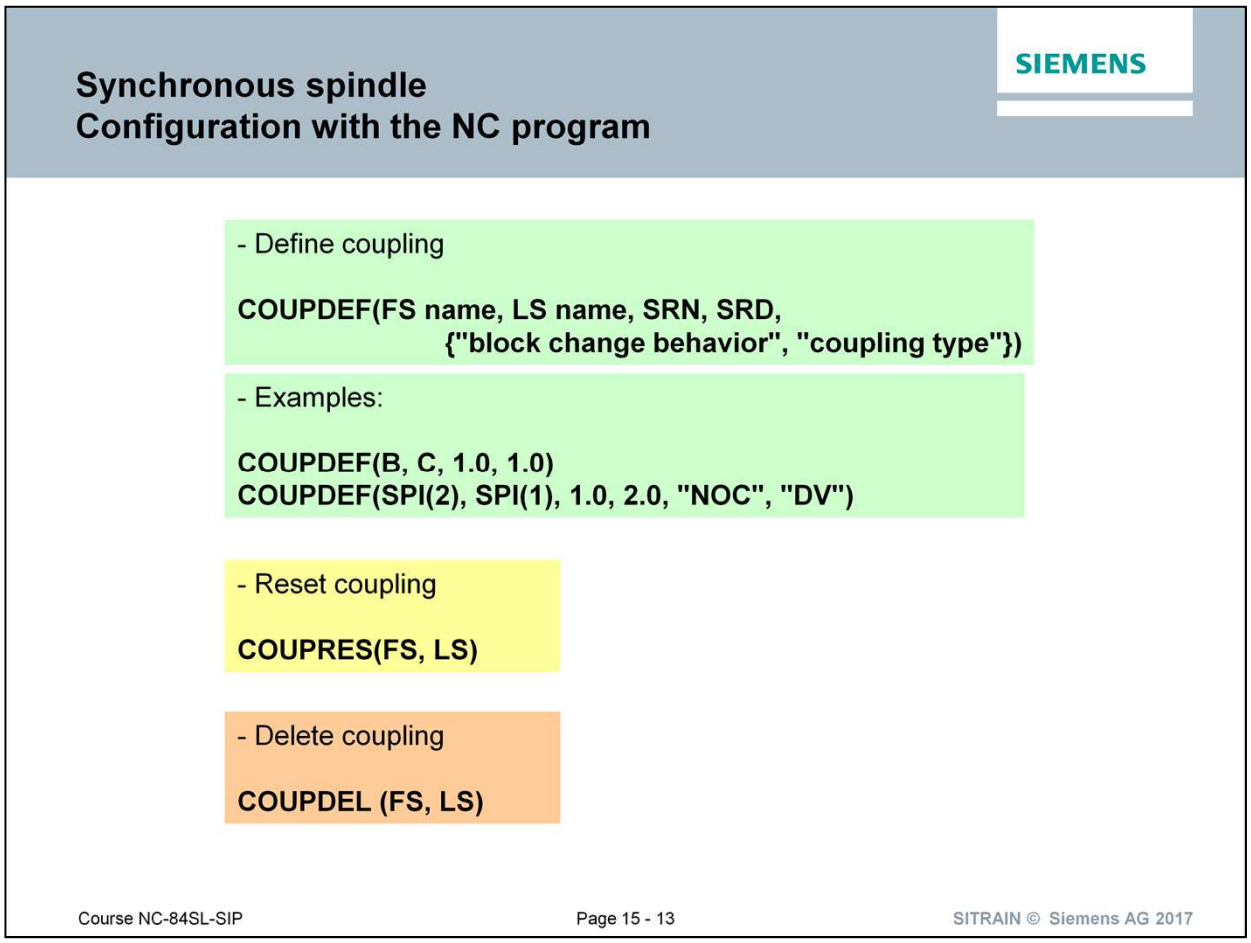
The "Synchronous spindles" function allows 2 spindles to be operated with a high degree of angular accuracy (1 leading spindle (LS), 1 following spindle (FS)). When the function is activated, only the leading spindle is programmed. The following spindle moves with angular synchronism or according to a selectable speed ratio. Any number of following spindles can be assigned to a leading spindle on any number of channels. The "Synchronous spindles" function is an option.

Coupling types for calculating the FS setpoint

1. Setpoint coupling – DV:
The FS setpoint is derived from the programmed LS value. Both spindle controllers receive their setpoints at the same time. Provided the control dynamics of both spindles are virtually the same, this coupling variant provides the best results. This is the preferred option, but it can only be used if there is a position-controlled LS.
2. Actual-value coupling – AV:
The FS setpoint is derived from the measured LS actual value. As a result, the setpoint for the FS is subject to a delay when output to the spindle controller. When faced with LS speed fluctuations, the synchronism quality is not as good with this type of coupling. To be used for spindles with very different control properties or in the case of an LS without position control.
3. Velocity coupling – VV:
Internally, this is also a form of setpoint coupling but it imposes less stringent requirements on the LS and FS. The spindles involved do not require any measuring systems, but this means that the position deviation of the spindles in relation to one another is undefined.

Control response

With coupling types DV and AV, the following spindle must be subject to position control (NC command: SPCON); as far as the leading spindle is concerned, both speed control and position control are permitted (although position control is the preferred option in this case as well).



Defining coupling

COUPDEF(FS name, LS name, ÜZ, ÜN,{"block change behavior", "coupling type"})

FS name:	Machine axis name of the following spindle (MD 10000)
LS name:	Machine axis name of the leading spindle (MD 10000)
ÜZ:	Speed ratio parameter numerator Value range: -1 000 ... +1 000
ÜN:	Speed ratio parameter denominator Value range: -1 000 ... +1 000

Block change behavior:

NOC	immediately
FINE	after "synchronism fine"
COARSE	after "synchronism coarse"
IPOSTOP	after the setpoints have been synchronized

Coupling type:

DV	Setpoint coupling
AV	Actual value coupling
VV	Velocity coupling

Coupling reset

COUPRES(FS, LS)
The COUPRES reset command resets the definition parameters of the COUPDEF command to the machine / setting data values for the permanently configured coupling and to the default settings for the user-defined coupling.

Deleting coupling

COUPDEL(FS, LS)
The COUPDEL delete command deletes the old user-defined coupling of this spindle pair and allows you to define a new coupling link. This means that more than 2 user-defined couplings are supported per channel in sequence.

Synchronous spindle
Coupling ON/OFF

SIEMENS

Activating coupling

1. Immediate activation of synchronous mode without angle reference between LS and FS:
   COUPON(FS, LS)
2. Activation of synchronous mode after a specific angle reference has been achieved between the LS and FS:
   COUPON(FS, LS, POS_{FS})
   POS_{FS} Angular offset between LS and FS.
   Refers to the 0° position of the leading spindle in the positive direction of rotation.

Deactivating coupling

1. Immediate deactivation of synchronous mode:
   COUPOF(FS, LS)
2. Deactivation of synchronous mode after the following spindle has reached a specific position:
   COUPOF(FS, LS, POSFS)
3. Deactivation of synchronous mode after the leading and following spindles have overtraveled a specific position:
   COUPOF(FS, LS, POSFS, POSLS)

SITRAIN © Siemens AG 2017

Speed ratio

$n_{FS} = n_{LS} * k_{Ü}$

The speed ratio $k_{Ü}$ results from the quotient of the setting data SD 42 300[n]: Speed ratio parameter numerator/speed ratio parameter denominator.

$$k_{Ü} = \frac{\text{COUPLE_RATIO_1[0]}}{\text{COUPLE_RATIO_1[1]}}$$

Value range of the speed ratio parameters: -1 000 ... +1 000

Read out coupling state

The current coupling state is stored in axial system variable $AA_COUP_ACT[SPI(n)]. It must be read out for the following spindle (e.g. $AA_COUPT_ACT[S2])

All bits = 0	No coupling active
Bit 2 = 1	Synchronized spindle coupling active

Read out angular offset

- Setpoint-based position offset between FS and LS:
  $AA_COUP_OFFS[SPI(n)]
- Actual-value-based position offset between FS and LS:
  $VA_COUP_OFFS[SPI(n)]

Enter the spindle number of the following spindle as spindle number n.

Coupling across channels

It is only possible to couple spindles from different channels by using the configuration variant 'with the NC program' (COUPDEF command).

For this, the corresponding spindles must also be configured in the channel in which the coupling command is issued.

If the FS is not in the channel of the LS, COUPON automatically generates a spindle interchange. However, the FS must first have been released with RELEASE!

Synchronous spindle
Fixed configuration with MD

SIEMENS

Following/leading spindle definition

MD 21300 COUPLE_AXIS_1[n] = <machine axis no.>
Index [n] 0 Following spindle
1 Leading spindle

Coupling types

MD 21310 COUPLE_IS_DES_POS_1 = <coupling type>
0 Actual value coupling
1 Setpoint value coupling
2 Velocity coupling

SITRAIN © Siemens AG 2017

Block change behavior

MD 21320 COUPLE_BLOCK_CHANGE_CTRL_1 = <block change behavior>

0 Block change performed with command COUPON
1 Block change after the spindles have fallen below "Synchronism fine" limit (MD 37210)
2 Block change after the spindles have fallen below the "Synchronism coarse" limit (MD 37200)
3 Block change after the interpolator has accelerated and decelerated

Reset condition

MD 21 330: COUPLE_RESET_MODE_1 = <reset condition>

Bit 0=0 After a program restart, the coupling is retained until deactivated with COUPOF
Bit 0=1 After RESET, the coupling is canceled with the next NC START
Bit 1=0 Synchronous operation remains even after end of program and RESET is active until it is canceled with COUPOF
Bit 1=1 Synchronous operation is canceled at end of program or RESET
Bit 5=1 The configured data are active on program start
Bit 6=1 The configured data are active on end of program or RESET.
Bit 9=1 Synchronous operation is activated on program start.

Write protection

MD 21340 COUPLE_IS_WRITE_PROT_1 = <write protection>

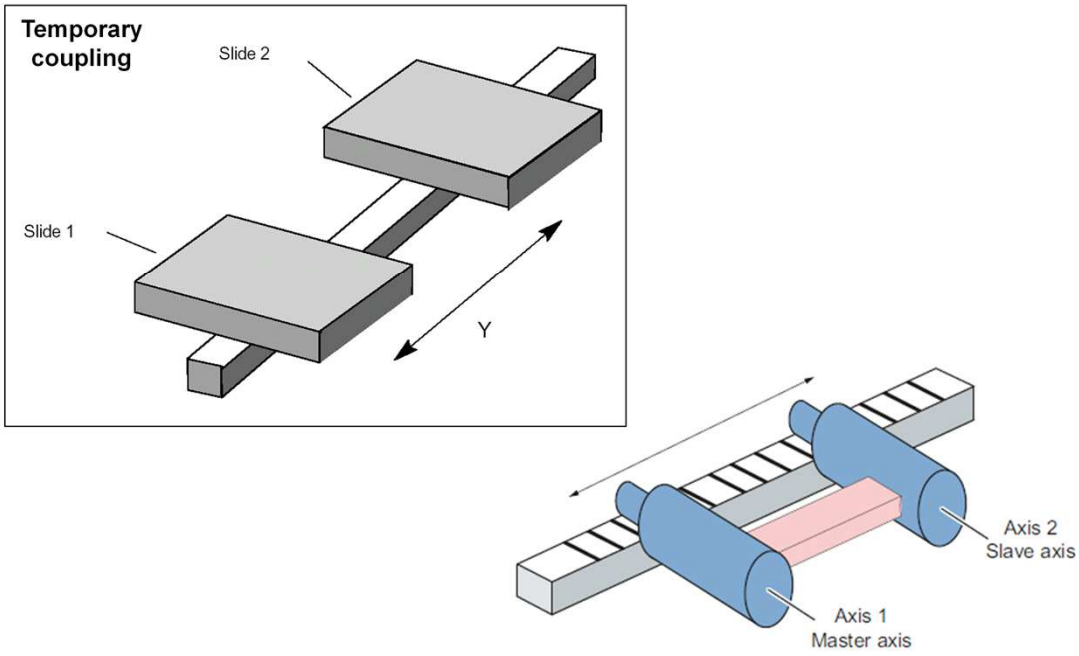
0 The coupling parameters can be changed with COUPDEF
1 Coupling parameters are write-protected

Positional deviation

MD 37200 COUPLE_POS_TOL_COARSE = <positional deviation>
MD 37210 COUPLE_POS_TOL_FINE = <positional deviation>

Master-slave coupling
FB TE3, option M03

SIEMENS



Course NC-84SL-SIP Page 15 - 16 SITRAIN © Siemens AG 2017

Master-slave Speed/torque coupling (master-slave) is a speed setpoint coupling between a master and a slave axis while using a torque compensatory controller for uniform torque distribution. This function is an option.

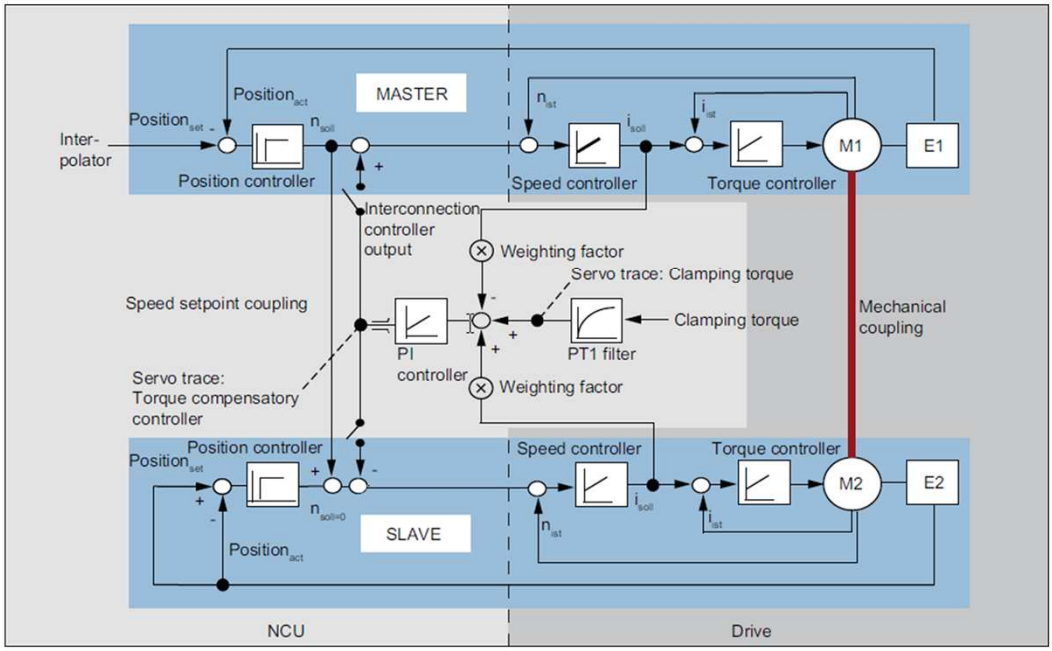
Fixed coupling This function is mainly used for boosting the power of mechanically-coupled drives. Another application is to compensate for gear backlash and tooth flank clearance by torque-biasing the drives against each other.

Temporary coupling The master-slave function can also be used to clamp workpieces between two slides on one axis. In this case, slide 1 is programmed, for example. Slide 2 travels such that a set torque is achieved in relation to slide 1.

Slave Each slave axis has exactly one master axis. A slave axis cannot be master of another slave axis.

Master A master axis can have several slave axes.

Master-slave coupling
Circuit diagram



 SITRAIN © Siemens AG 2017

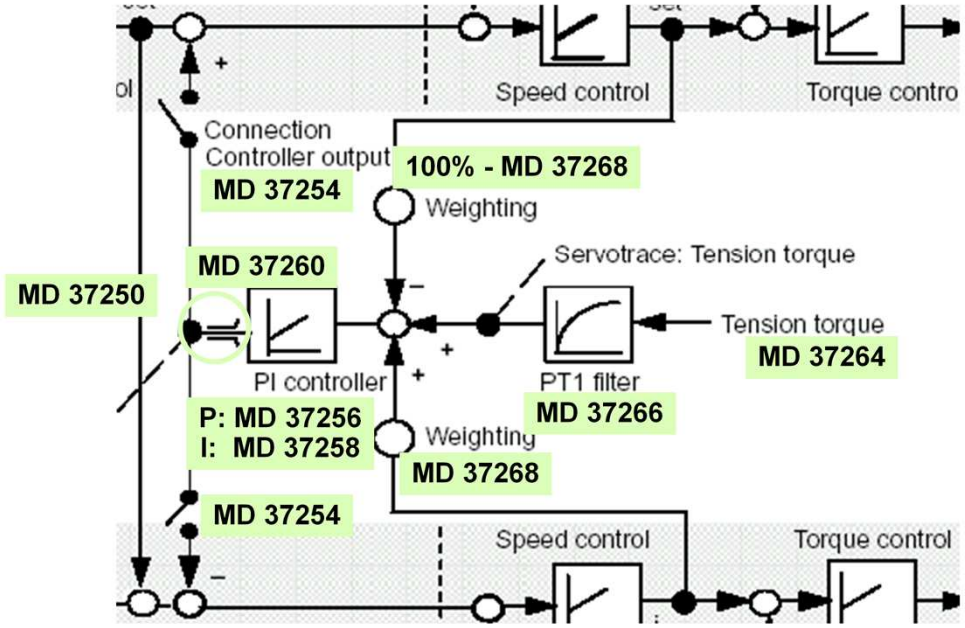
Position controller Slave axis position control is deactivated when the coupling is closed. The slave axis obtains the speed setpoint from the master axis.

Torque compensation The torque compensatory controller distributes the torque equally between master axis and slave axis.

Torque bias An additional torque bias can be specified on the torque compensatory controller.

Master-slave coupling
Configuration

SIEMENS



 SITRAIN © Siemens AG 2017

Slave axis

The machine data for configuring the master-slave coupling is configured in the slave axis.

Master axis

The master-slave coupling is configured by setting the master axis in MD 37250 $MA_MS_ASSIGN_MASTER_SPEED_CMD (speed coupling). The master axis for the torque compensatory controller can be selected in MD 37252 $MA_MS_ASSIGN_MASTER_TORQUE_CTR. A deviation from the default value 0 (-> torque compensation with the axis parameterized in MD 37250) is only necessary in special cases.

Controller output interconnection

In MD 37254 $MA_MS_TORQUE_CTRL_MODE, you can set the axes to which you want the output of the torque compensatory controller to be connected (0: Master and slave axis, 1: Slave axis, 2: Master axis, 3: None)

Torque compensatory controller

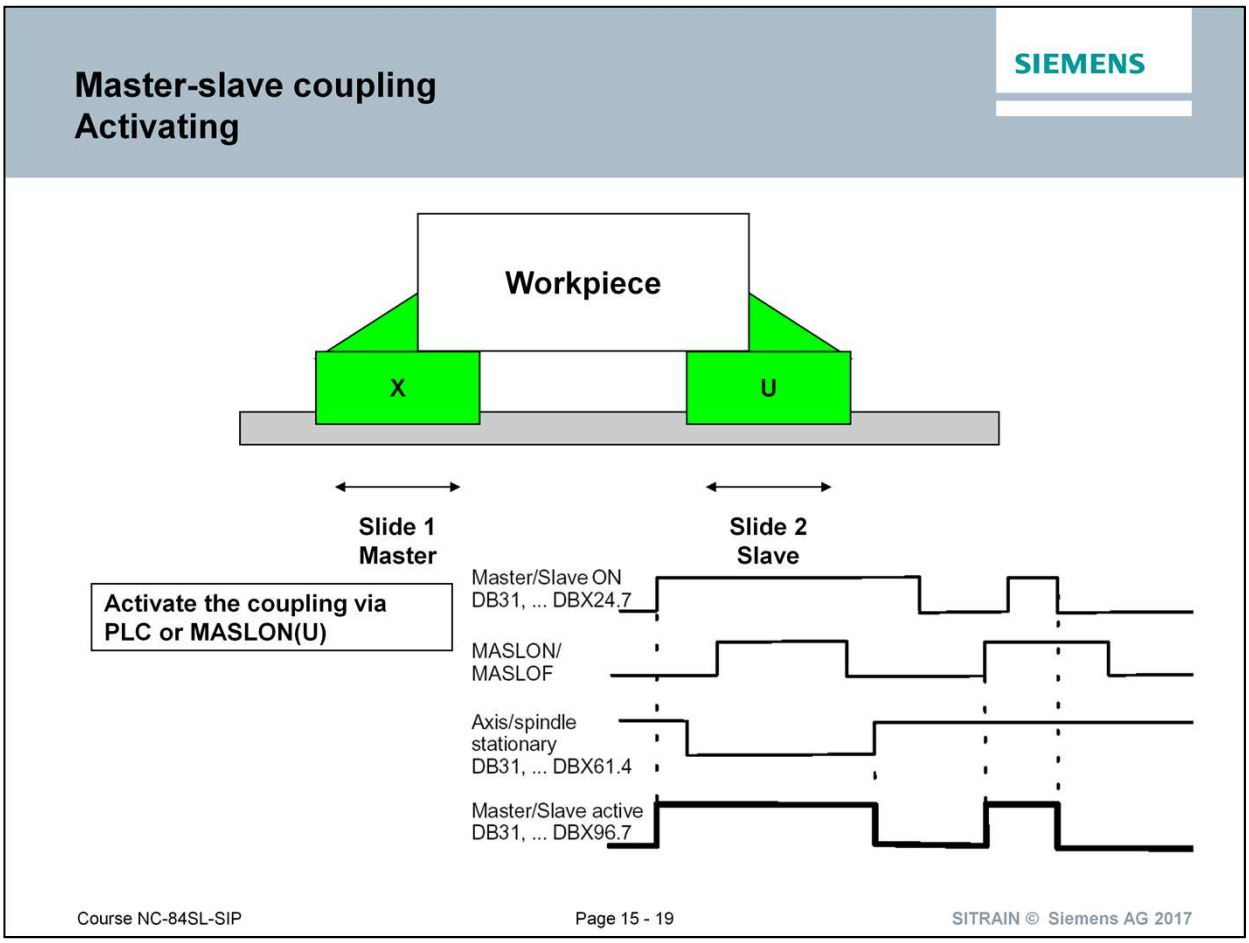
The torque compensatory controller is a PI controller. The controller is parameterized with the machine data MD 37256 $MA_MS_TORQUE_CTRL_P_GAIN and MD 37258 $MA_MS_TORQUE_CTRL_I_TIME. You can set a limit for the controller output in MD 37260 $MA_MS_MAX_CTRL_VELO.

Torque bias

You can set a torque bias for the coupled axes via MD 37264. Changes are effective immediately; therefore, the machine data can, for example, be modified from the part program or by synchronized actions. Changes are passed to the torque compensatory controller via the PT1 filter using a time constant (MD 37266).

Torque distribution

The torque value for the slave axis is set in MD 37268. The share of torque for the master axis is provided by 100 % minus MD37268.



Fixed coupling
For permanent coupling, set MD 37262 of the slave axis to 1.

Temporary coupling
Temporary coupling is active if MD 37262 of the slave axis has a value of zero. The coupling is then activated or deactivated by the PLC interface signal or a program command.

PLC signals
DB<slave-axis>.DBX24.7 Activate master-slave coupling The coupling is only active if the following conditions are met:

- Master and slave axis subject to position control: DB<axis>.DBX61.7
- Master and slave axis at standstill: DB<axis>.DBX61.4
- Channel status = reset: DB<channel>.DBX35.7

Checkback signal for coupling state: DB<slave axis>.DBX96.7

NC commands
Temporary coupling of the X axis as master with the U axis as slave – if configured in the machine data – can be activated with MASLON(U) and deactivated with MASLOF. For activation in the program, the same conditions apply as for activation with the PLC interface signals.

Release coupling
The command for releasing the coupling is only effective once both axes are at a standstill.

System variable
The system variable $AA_MASL_STAT[slave-axis] indicates the coupling state of an axis.
=0: The coupling is not active
>0: Machine axis number of the master axis if coupling active

Appendix
Generic coupling

SIEMENS

"Generic Coupling" is a general coupling function, combining all coupling properties of existing coupling types (coupled motion, master value coupling, electronic gear, and synchronous spindle).

The function allows flexible programming:

- Users are free to select the coupling properties required for their applications (building block principle).
- Each coupling property can be programmed individually.
- The coupling properties of a defined coupling (e.g. coupling factor) can be changed.
- Additional coupling properties can be used later on.
- The coordinate reference system of the following axis (basic coordinate system or machine coordinate system) is programmable.
- Certain coupling properties can also be programmed in synchronous actions.

TRAILON(X,Y,2) ➔ CPON=(X1) CPLA[X1]=(X2) CPLNUM[X1,X2]=2

The "old" programming approach is still possible.

Course NC-84SL-SIP Page 15 - 20 SITRAIN © Siemens AG 2017

Adaptive cycles

Previous coupling calls for coupled motion (TRAIL*), master value coupling (LEAD*), electronic gear (EG*), and synchronous spindle (COUP*) are still supported via adaptive cycles CYCLE700 … CYCLE717.

Coupling module

With the help of a coupling module, the motion of one axis (→ following axis) can be interpolated depending on other axes (→ leading axes).

Coupling rule

The relationships between the leading axis/values and the following axis are defined by a coupling rule (coupling factor or curve table). The individual motion components from the individual leading axes/values have an additive effect.

Programming

Old axis couplings are programmed via NC commands, e.g. coupled motion TRAILON(X,Y,2). With generic coupling, keywords are used instead of these NC commands.

Advantages:

- Coupling properties can be programmed individually (see following example).
- Multiple couplings can be programmed in one block (since keywords do not require their own block).
  Advantage: Reduction of work off time

Example:

Instead of TRAILON(X,Y,2) (FA, LA, coupling factor), which is what was used previously, the following is now defined:

CPON=(X1) CPLA[X1]=(X2) CPLNUM[X1,X2]=2

CPON=(X1)	Switch on coupling with the following axis X1.
CPLA[X1]=(X2)	Define axis X2 as the leading axis.
CPLNUM[X1,X2]=2	Set numerator of the coupling factor to 2.

Appendix
Adaptation by cycle 700…717

SIEMENS

The adaptive cycles ensure a syntactic and functional compatibility to coupling calls of existing coupling types (coupled motion, master value coupling, electronic gear and synchronous spindle). As long as the manufacturer/user does not need new coupling characteristics, it is not necessary to modify present coupling calls (e.g. evaluation of coupling signals in the PLC).

Adaptive cycle command	
TRAILON	**cycle700**
TRAILOF	**cycle701**
LEADON	**cycle702**
LEADOF	**cycle703**

Adaptive cycle command	
COUPDEF	**cycle704**
COUPON	**cycle705**
COUPONC	**cycle706**
COUPOF	**cycle707**
COUPOFS	**cycle708**
COUPDEL	**cycle709**
COUPRES	**cycle710**

Adaptive cycle command	
EGDEF	**cycle711**
EGON	**cycle712**
EGONSYN	**cycle713**
EGONSYNE	**cycle714**
EGOFC	**cycle715**
EGOFS	**cycle716**
EGDEL	**cycle717**

SITRAIN © Siemens AG 2017

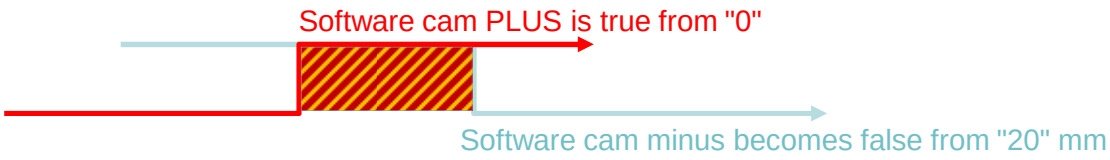
CYCLE700 …717 are restored in the standard cycle directory whenever an NC run-up is performed.

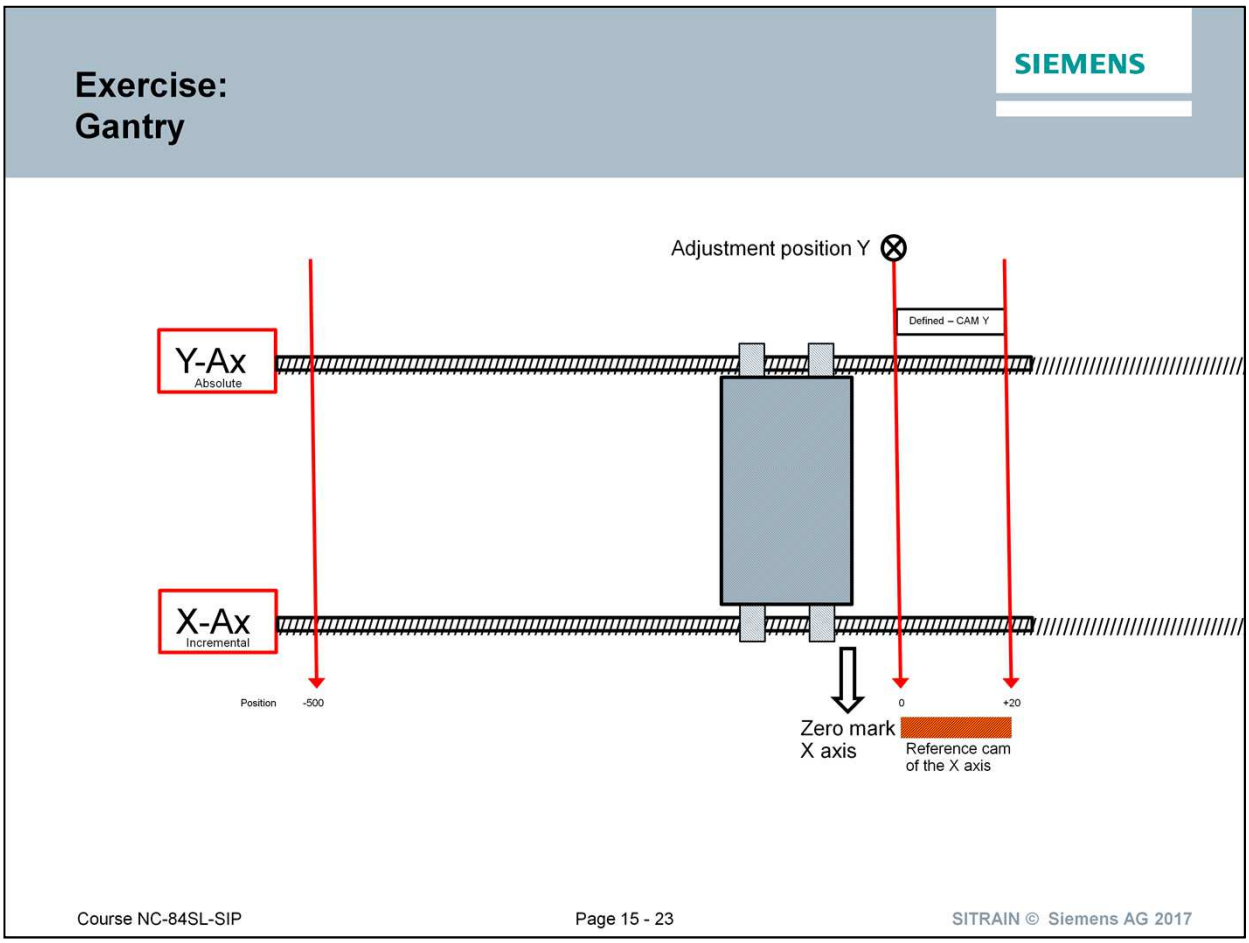
Exercise: Cam controller (perform prior to gantry exercise)

For the Y axis (absolute encoder) of the training device, create a cam that switches on at 0 mm and switches off at 20 mm.

Perform the following steps:

1. Set the position switching signals/cam controller option.
2. Assign the Y axis to the first cam in machine data 10450.
3. Set the position values in the setting data 41500 and 41501.
   SD41500 [0] $SN_SW_CAM_MINUS_POS_TAB_1 = 20
   SD41501 [0] $SN_SW_CAM_PLUS_POS_TAB_1 = 0
   In this way, both cam signals between 0 and 20 mm are true, above this and below this, only one cam is true.
4. Activate the cam switching signals on the PLC interface of the Y axis.
   DB32.DBX2.0 for Y = 2nd machine axis.
5. Make sure the absolute encoder is correctly adjusted.
   MD34100[0] $MA_REFP_SET_POS = 0
   MD 34210[0]$MA_ENC_REFP_STATE = 2
6. Check in the interface of the DB 10 that both cams are true in the range between Y=0 and Y=20.
   DB10.DBX110.0 -> software cam 1 minus
   DB10.DBX114.0 -> software cam 1 plus





Exercise: Gantry

The X and the Y axis of the training rack are to be coupled in the gantry grouping 1.
Adjusting the encoder in advance improves handling on the training racks, but has nothing to do with the functionality as such.

Please note that the Y axis on virtually every training rack features an absolute encoder, which means that the reference point approach is shorter.

Option

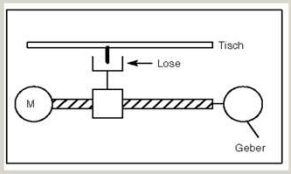
Activate the pair of synchronous axes option (gantry axes)
6FC5800-0AM02-0YB0

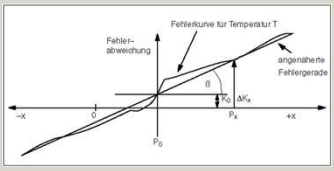
Exercise: Gantry

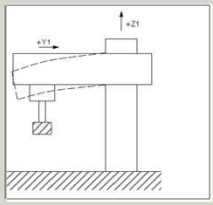
1. Adjust the (absolute) Y axis with MD 34100 = 0.
2. Set MD 34100 for the X axis to 0.
3. Define X-axis as leading axis: MD 37100[X] = 01
4. Define Y-axis as following axis: MD 37100[Y] = 11
5. Remove Y from list of the geometric axis: MD 20050[0] = 0
6. Before referencing and synchronizing the following axis, set the error limit to a value greater than zero to prevent errors prior to synchronization;
   MD 37130[X,Y] = 1mm
7. Set gantry warning limit: MD 37110[X,Y] =0.5mm
8. Set the gantry error limit to either
   zero (= trip) or a higher value. MD 37120[X,Y] = 10mm
9. Hide the Y axis from the MCS/WCS display,
   leave it in the ref. point display: MD 20098[1] = 2H
10. As the cam signal for the X axis, use the software cam of the absolute encoder for the Y axis.
    To do this, in the PLC program, set the bit DB32.DBX 2.0,
    and DB10.DBX114.0
    to DB31.DBX12.7.
11. NC RESET
12. Start axis-specific referencing for the X axis. For improved visualization, the reference point speeds can be reduced.
    MD 34070[X] = 1000 mm/min
    MD 34020[X] = 1000 mm/min
    MD 34040[X] = 1000 mm/min
13. The X axis is referenced. The Y axis moves in synchronism. The result is the reference position of the X axis with the position "0" at another position of the Y axis. You see the message "10654 Waiting for synchr. start".
14. Transfer the value of the Y axis into MD34100 of the X axis and reference it again.

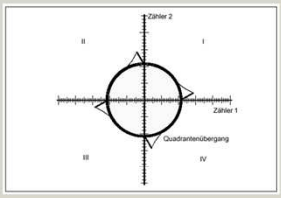
After the reference point approach, the same value is entered in the X axis as the one adopted by the Y axis. A gantry synchronization run is therefore unnecessary.

SIEMENS









Chapter 16

Compensations

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Overview 3
Backlash compensation 4
Temperature compensation 5
Leadscrew error/measuring system 8
Friction compensation 15
Friction compensation 16
Vertical axes 17
Exercise: Leadscrew and measuring system error compensation 18

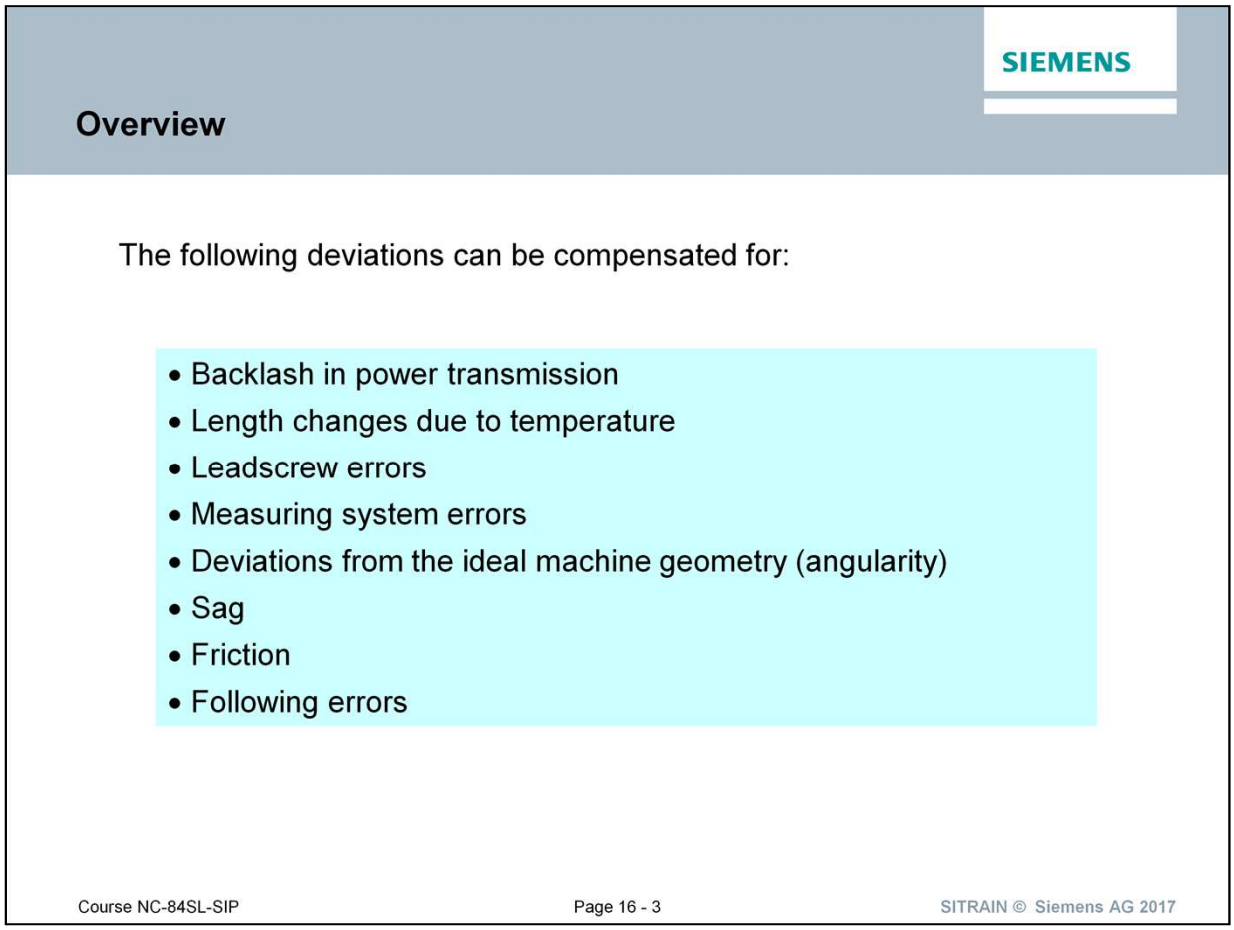
Learning objectives

- You will be familiar with leadscrew error compensation
- You will be familiar with selected compensation types and how to apply them



Course NC-84SL-SIP Page 16 - 2 SITRAIN © Siemens AG 2017

Notes



Compensations

The accuracy of MTs is impaired by deviations from the ideal geometry, power transmission faults and measuring system errors, physical properties, and NC-related errors.

Temperature differences and mechanical forces often result in great reductions in precision when large workpieces are machined.

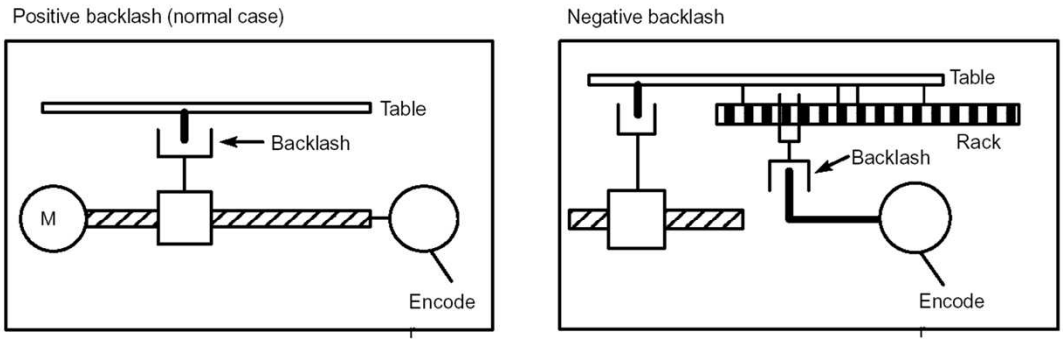
These impairments can be compensated for using the CNC. A slight residual error will remain because of idealized assumptions about the machine behavior (e.g. an even response to temperature changes) or because of approximated error curves. Despite this, the residual error will generally be significantly below the permissible workpiece tolerances.

Some of the compensations listed above are already included in the basic version of the CNC, but highly complex compensation functions have to be ordered as a separate option. All compensations can be deactivated in cases where, for example, the machine accuracy is already very high and the intention is to remove the burden of additional calculation tasks from the CNC to achieve the shortest possible cycle times.

Activation

Compensations relate to axis positions. This means that they do not become active until the reference point for the axes concerned is approached.

Backlash compensation



Encoder actual value is ahead of the real actual value (table): the table does not travel far enough

Real actual value (table) is ahead of the encoder actual value:the table travels too far

MD 32450: Backlash

Course NC-84SL-SIP Page 16 - 4 SITRAIN © Siemens AG 2017

Backlash In the case of axes/spindles with indirect measuring systems, mechanical backlash results in corruption of the traverse path. For example, when the direction of movement is reversed, an axis will travel too much or too little by the amount of the backlash.

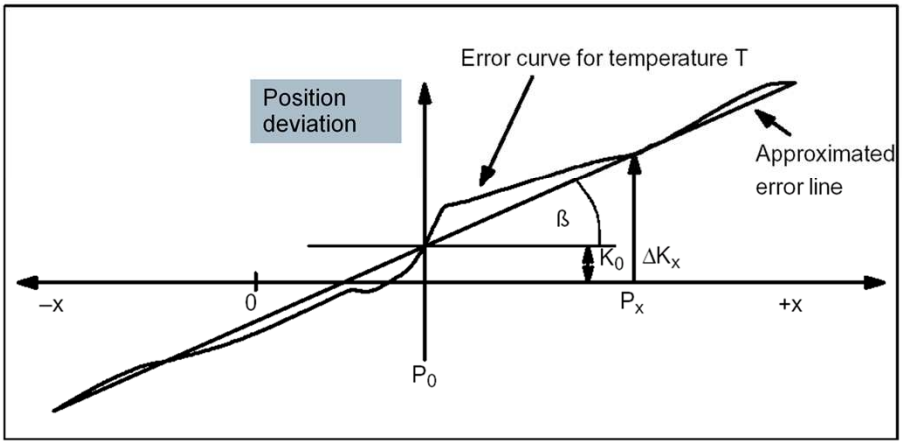
Compensation To compensate for backlash, the axis-specific actual value is corrected by the amount of backlash every time the axis/spindle changes direction. This amount can be entered for each axis/spindle at the start-up phase in MD 32450: BACKLASH. If a 2nd measuring system is available, the relevant backlash must be entered for each of the two measuring systems.

Activation Backlash compensation is always active in all operating modes after reference point approach.

Positive backlash The encoder is ahead of the machine part (e.g. table). Since the actual position acquired by the encoder is also ahead of the true actual position of the table, the table travels too short a distance. The backlash offset value must be entered as a positive value here (normal case).

Negative backlash The encoder lags behind the machine part (e.g. table); the table then travels too far. The correction value entered is negative.

Temperature compensation



Course NC-84SL-SIP Page 16 - 5 SITRAIN © Siemens AG 2017

Temperature-dependent parameters

Error curves for different temperatures can be defined for each axis, as illustrated in the diagram above. The curves are obtained by measuring the position change in the event of temperature changes at various points. This results in an error curve like the one shown above.

For each error curve the following parameters must be determined and then entered in the setting data:

K0 SD 43900: $SA_TEMP_COMP_ABS_VALUE
Temperature compensation value independent of the position

P0 SD 43920: $SA_TEMP_COMP_REF_POSITION
Reference position for position-dependent temperature compensation

tan ß SD 43910: $SA_TEMP_COMP_SLOPE
Lead angle for position-dependent temperature compensation

Note If the setting data cannot be viewed in the 'Parameters' operating area, you can still read/write the corresponding variables.

Temperature changes

Since the approximated error line applies only to the instantaneous temperature value, the parameters of the error lines that are newly generated when the temperature rises or falls must be sent to the NCK again. Only in this way can expansion due to heat always be correctly compensated.

Temperature compensation
Compensation straight line

SIEMENS

The compensation value K_x is calculated based on the current actual position P_x of this axis and temperature T according to the following equation:

$$K_x = K_0 (T) + \tan ß (T) * (P_x - P_0)$$

Coefficient comparison with a linear equation:

$$y = b + m * x$$

where:

$K_0 (T) = b$	Non-position-dependent component (= compensation value at reference point)
$\tan ß (T) = m$	Gradient of the straight line
$(P_x - P_0) = x$	Distance from reference point

Course NC-84SL-SIP Page 16 - 6 SITRAIN © Siemens AG 2017

Kx	Temperature compensation value of axis at position P x
K0	Non-position-dependent temperature compensation value of axis (reference point offset due to temperature expansion)
Px	Actual position of axis
P0	Reference position of axis
tanß	Coefficient for position-dependent temperature compensation (corresponds to the gradient of the approximated error line)

Note The compensation values are acquired in interpolation cycles. If the compensation value Kx is positive, the axis moves in the negative direction.

Temperature compensation Configuration

SIEMENS

MD 32750: TEMP_COMP_TYPE	Meaning	Associated parameters
0	No temperature compensation active	_____
1	Position-independent temperature compensation active	SD 43900: TEMP_COMP_ABS_VALUE
2	Position-dependent temperature compensation active	SD 43920: TEMP_COMP_REF_POSITION SD 43910: TEMP_COMP_SLOPE
3	Position-independent and position-dependent temperature compensation active	SD 43900: TEMP_COMP_ABS_VALUE SD 43920: TEMP_COMP_REF_POSITION SD 43910: TEMP_COMP_SLOPE

Course NC-84SL-SIP Page 16 - 7 SITRAIN © Siemens AG 2017

Activation

The following conditions must be fulfilled so that temperature compensation can be activated:

The compensation type is selected (MD 32750: TEMP_COMP_TYPE).

The parameters for the compensation type are defined.

The axis is referenced (IS "Referenced/synchronized 1 or 2" DB31...48, DBX60.4 or 60.5 = 1).

Effect

Following this, the temperature compensation value for the current position actual value is added to the setpoint in all operating modes and the machine axis traverses this distance.

If the reference position is lost, e.g. because the encoder limit frequency has been exceeded (IS "Referenced/synchronized 1 or 2" = 0), compensation processing is deactivated.

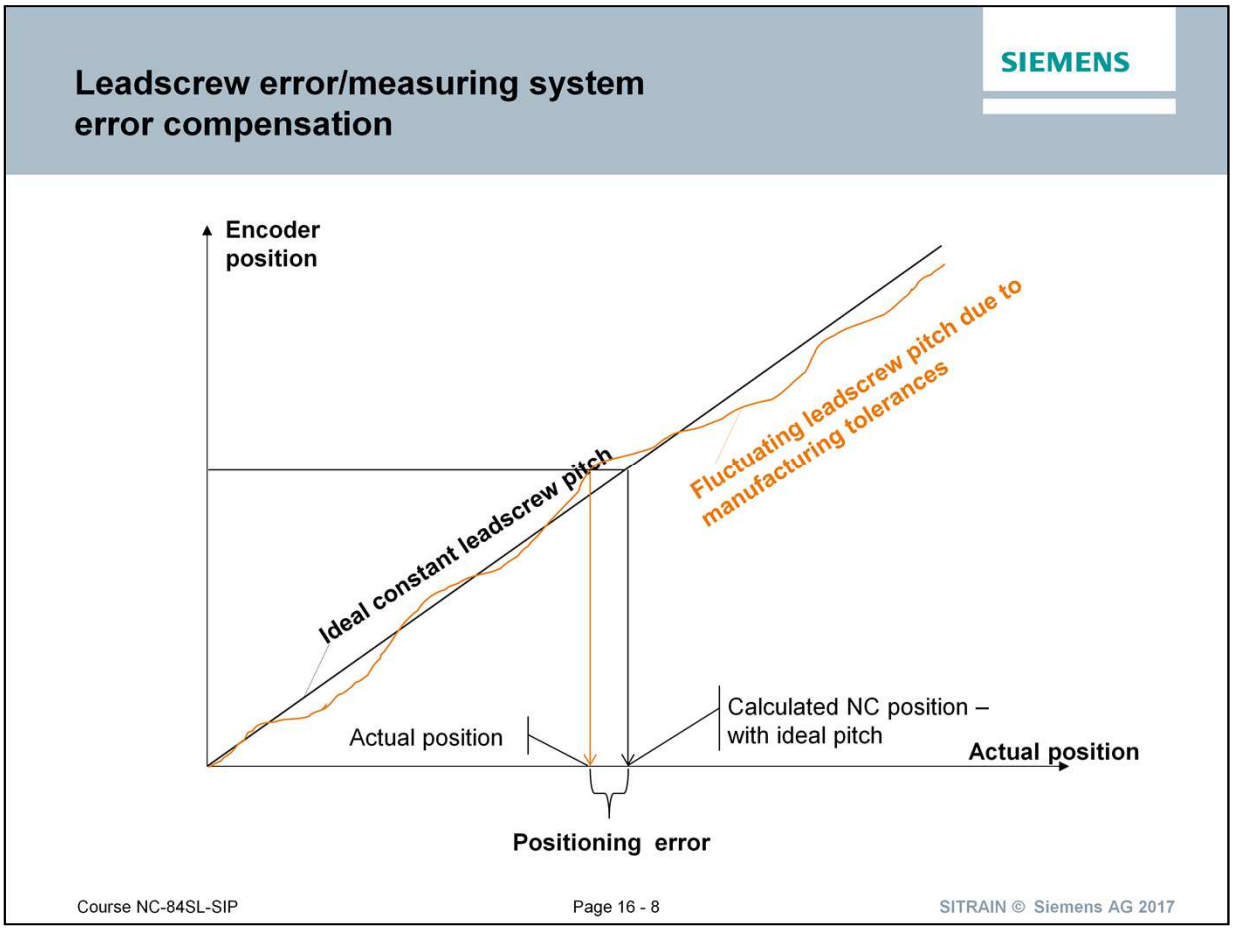
Update

When temperature T changes, the parameters which are temperature-dependent (K0, tan ß, and P0) also change and can thus always be overwritten by the PLC using the service FB3 (PUT) "Write data".

This means it is possible for the machine manufacturer to emulate the mathematical and technological relationships between the axis positions and temperature values in a PLC user program and thus calculate the relevant parameters.

MD 32760:

COMP_ADD_VELO_FACTOR
Maximum compensation value per IPO cycle



Positioning error

The measuring principle of "indirect measurement" on NC-controlled machines is based on the assumption that the pitch of the ball screw is constant at any point within the traversing range, so that the actual position of the axis can be derived from the position of the drive spindle (ideal case).

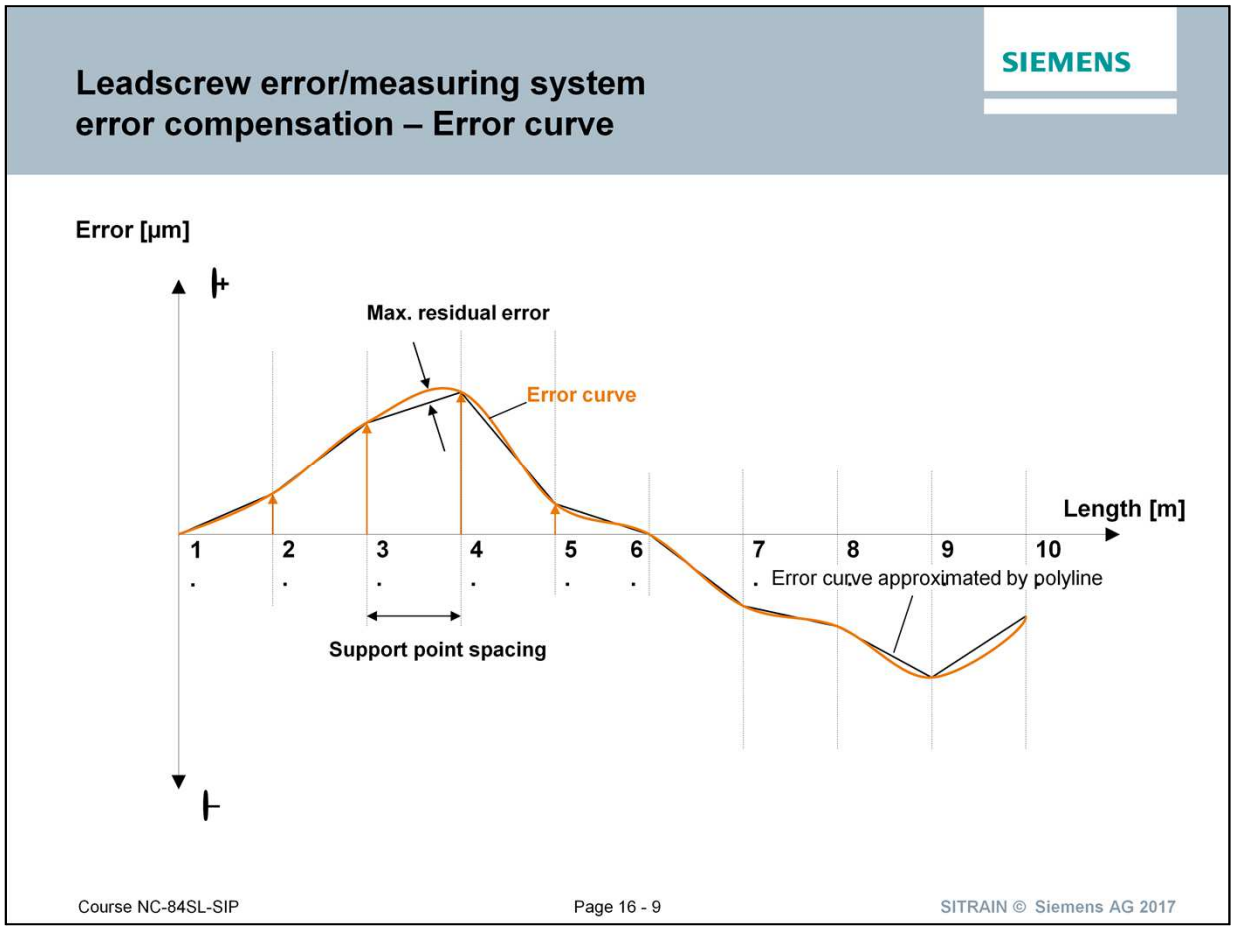
Manufacturing tolerances for position-defining machine elements result in more or less significant deviations between the measured and actual axis positions.

Added to this are the dimensional deviations (differences in reference division) caused by the measuring system as well as its mounting on the machine (so-called measuring system errors), plus any machine-specific sources of error.

Activating an offset value makes it possible to reduce the overall deviation significantly.

Simple or bidirectional

In addition to the simple compensation described above, the leadscrew error can also be compensated for direction-dependently. This can also be used to compensate for position-dependent backlash.



Obtaining the error curve

To determine the offset value, a highly precise measuring instrument (usually a laser interferometer) is used to identify the actual position. The difference between this and the NC position provides the offset value for this point. You obtain the error curve for this axis by measuring as many different points of this deviation as possible and applying them over the measured positions.

Determine the support point spacing

Any distance between the points (intermediate points of the polyline that describes the error curve) can be chosen, as long as it remains constant within a single axis. This support point spacing is used to calculate the residual error, which must be determined at the point where the error curve exhibits the greatest curvature. If the residual error is too large at this point, the support point spacing must be reduced.

Start position, end position

You can freely select the positions between which compensation can be performed. If this area is exceeded, the last compensation value remains active until the compensation area is reached again.

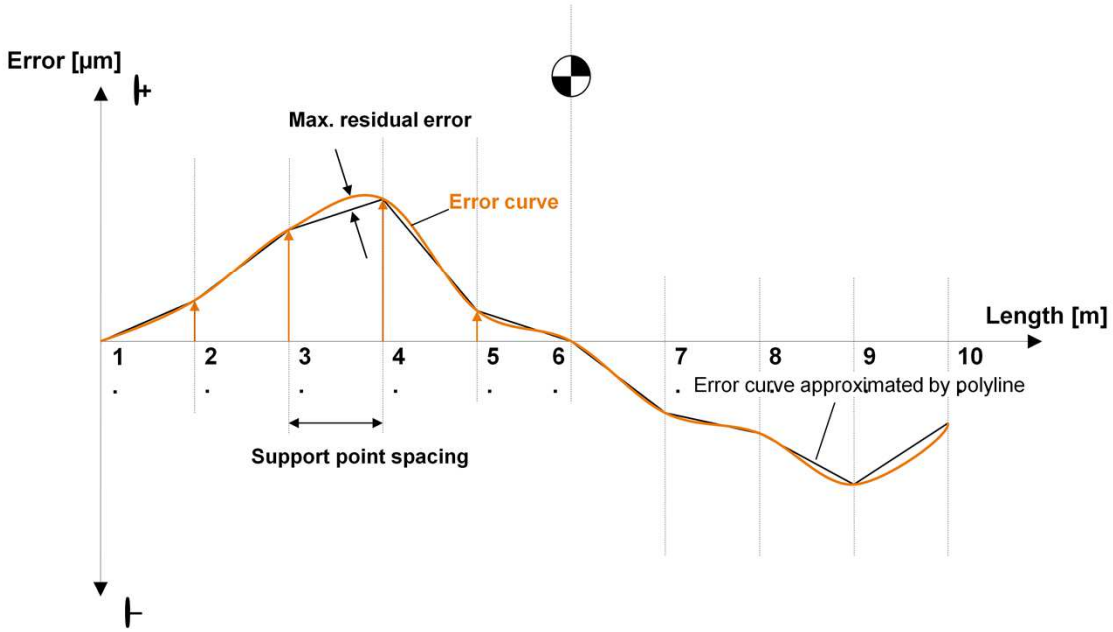
Calculating the number of intermediate points

The number of intermediate points is calculated on the basis of the axis length to be compensated and the support point spacing.

2 measuring systems

Where 2 measuring systems are used, each can be compensated for separately in its own table. The table that is active is selected automatically when you switch between the measuring systems.

Leadscrew error/measuring system error compensation – Reference point



Course NC-84SL-SIP Page 16 - 10 SITRAIN © Siemens AG 2017

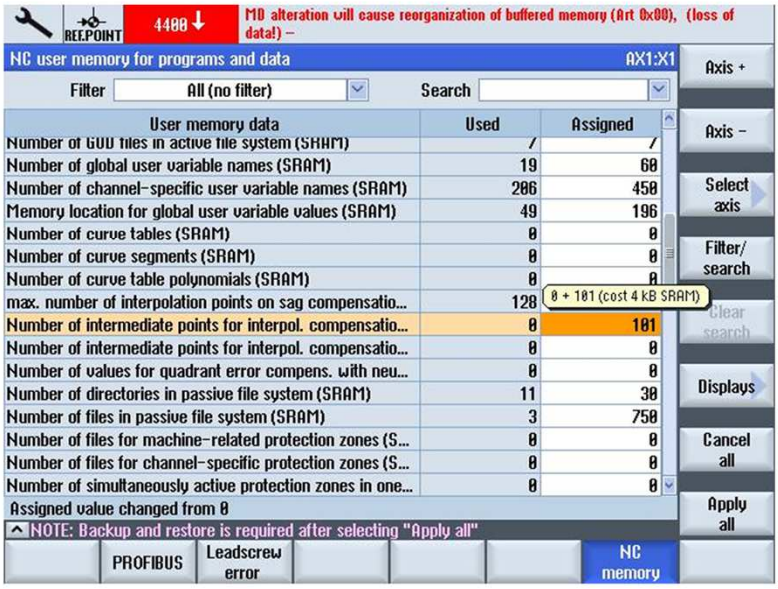
Interpolation

For axis positions between 2 intermediate points, the NC linearly interpolates the two adjacent compensation values ("interpolatory compensation"). Each table may contain up to 5,000 offset values. The modulation factors are calculated in the interpolation cycle, with the offset value applied to the axis-specific position actual value. Positive offset values result in axis motion in the negative travel direction. It is possible to check the current compensation value in the "Diagnostics" operating area, softkey "Service displays", "Service axis".

Reference point

Compensation is activated once the reference point has been approached. At this point, the offset value should be zero to prevent axis jerk after referencing. (The compensation is taken into account in the position controller, which is why "soft" acceleration is not performed.) Once the error curve has been measured, the zero line must be moved so that it is intersecting the error curve at the reference point.

Leadscrew error/measuring system error compensation – Number of intermediate points SRAM storage location



 SITRAIN © Siemens AG 2017

Number of intermediate points

This must be entered in → **softkey Setup** → **softkey NC** → **softkey NC memory**. It can be larger, but not smaller than the value actually required.

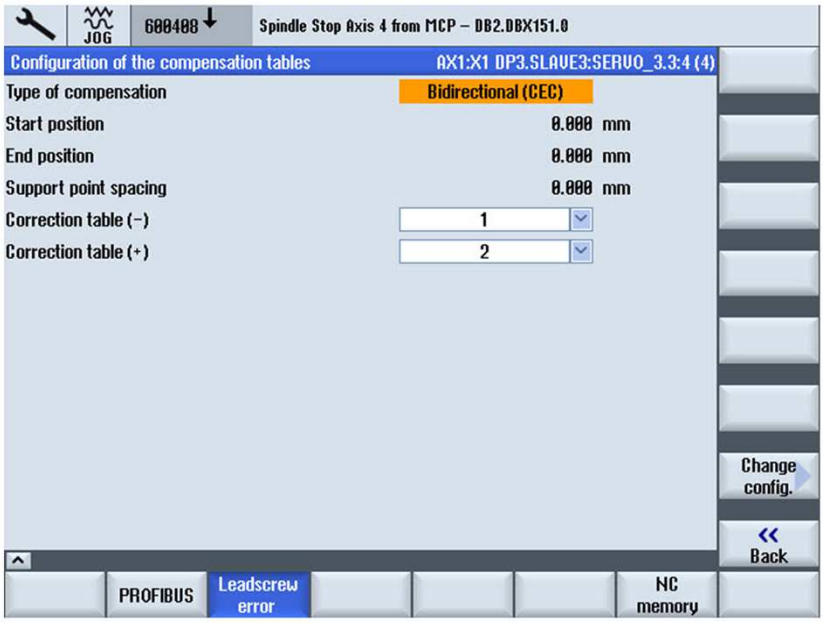
Variable name

Simple (EEC)

Depending on the software version, the value is called either
Measuring system/leadscrew error compensation or
Number of intermediate points in interpolatory compensation (SRAM)

Bidirectional (CEC)

Depending on the software version, the value is called either
Sag compensation or
Maximum number of interpolation points in sag compensation (SRAM)
Because it is a memory-configuring machine data item, after changing and before activation, an NC data backup must be generated and imported again. The control points this out and then automatically displays the data backup screen.



 SITRAIN © Siemens AG 2017

Configuration

In → **softkey Setup** → **softkey NC** → **softkey Leadscrew error** → **softkey Configuration**, it is now possible to choose between
Simple (EEC) and
Bidirectional (CEC).

Commissioning

With → **softkey Change config.,**
the axis positions between which the axis will compensated and the support point spacing are defined first.
With → **softkey Activate**,
after Power On Reset with
→ **softkey Back**, you get to the list of compensation values that has now been created and you can enter the compensation values determined at the intermediate points in mm with
→ **softkey Change values**.
With
→ **softkey Activate** the values are reactivated.
After referencing, the values are active and can be checked in
→ **softkey Diagnostics** → **softkey Axis diag.** → **softkey Service axis** under **Abs. compensation value measuring system 1 (EEC)** or
Compensation sag + temperature (CEC).

Bidirectional (CEC)

For this compensation, a separate table is used for compensation for each direction of movement. Tables are available for all axes in the default value 8.
By setting the
MD 18342 [8]-[61] (number of intermediate points),
a total of up to 62 tables can be created.

Leadscrew error/measuring system error compensation – System variable EEC

Offset values:	**$AA_ENC_COMP[e, n, AXi]**	**= <offset value>**
Support point spacing:	**$AA_ENC_COMP_STEP[e, Ax_i]**	**= <interval>**
Start position:	**$AA_ENC_COMP_MIN[e, Axi]**	**= <axis actual value>**
End position:	**$AA_ENC_COMP_MAX[e, Axi]**	**= <axis actual value>**

NOTE: It is only possible to write to the table values when compensation is deactivated

Course NC-84SL-SIP Page 16 - 13 SITRAIN © Siemens AG 2017

Offset values $AA_ENC_COMP[e, n, AXi]

For each intermediate point, the absolute error must be entered as an offset value. The values are not subject to any limits, although higher offset values may result in instability as a result of stepped activation in the position control loop for the purpose of activating velocity monitoring.

Support point spacing $AA_ENC_COMP_STEP[e, Axi]

The maximum support point spacing (s max) is calculated based on the position of the ball screw that demonstrates the most significant error change; i.e. where the curvature of the error curve is at its maximum. The support point spacing selected should be at least small enough to prevent the positioning error from exceeding the permissible tolerance because of error curve simulation by a polyline.

Start position $AA_ENC_COMP_MIN[e, Axi]

The start position is the 1st intermediate point (intermediate point index n = 0). The offset value of this intermediate point is assigned to all position values that are lower than the starting position.

End position $AA_ENC_COMP_MAX[e, Axi]

The end position determines the end of the compensation table. The offset value of the last intermediate point is used for this position and all higher-value positions.

Leadscrew error/measuring system error compensation – Alternative commissioning

SIEMENS

```
$MA_CEC_ENABLE[AX1]=0
$SN_CEC_TABLE_ENABLE[0]=0
$SN_CEC_TABLE_ENABLE[1]=0
NEWCONF
CHANDATA(1)
$AN_CEC[0,1]=1
$AN_CEC[0,2]=2
....
$AN_CEC[0,7]=3
$AN_CEC[0,8]=2
$AN_CEC_INPUT_NCU[0]=1
$AN_CEC_INPUT_AXIS[0]=(AX1)
$AN_CEC_OUTPUT_NCU[0]=1
$AN_CEC_OUTPUT_AXIS[0]=(AX1)
$AN_CEC_STEP[0]=10
$AN_CEC_MIN[0]=10
$AN_CEC_MAX[0]=100
$AN_CEC_DIRECTION[0]=-1
$AN_CEC_MULT_BY_TABLE[0]=0
$AN_CEC_IS_MODULO[0]=0
$MA_CEC_ENABLE[AX1]=1
$SN_CEC_TABLE_ENABLE[0]=1
$SN_CEC_TABLE_ENABLE[1]=1
NEWCONF
M17
```

```
$MA_CEC_ENABLE[AX1]=0
$SN_CEC_TABLE_ENABLE[0]=0
$SN_CEC_TABLE_ENABLE[1]=0
NEWCONF
CHANDATA(1)
$AN_CEC[1,1]=1
$AN_CEC[1,2]=2
....
$AN_CEC[1,7]=1
$AN_CEC[1,8]=0.01
$AN_CEC_INPUT_NCU[1]=1
$AN_CEC_INPUT_AXIS[1]=(AX1)
$AN_CEC_OUTPUT_NCU[1]=1
$AN_CEC_OUTPUT_AXIS[1]=(AX1)
$AN_CEC_STEP[1]=10
$AN_CEC_MIN[1]=10
$AN_CEC_MAX[1]=100
$AN_CEC_DIRECTION[1]=1
$AN_CEC_MULT_BY_TABLE[1]=0
$AN_CEC_IS_MODULO[1]=0
$MA_CEC_ENABLE[AX1]=1
$SN_CEC_TABLE_ENABLE[0]=1
$SN_CEC_TABLE_ENABLE[1]=1
NEWCONF
M17
```


SITRAIN © Siemens AG 2017

Offset values Instead of being able to enter the offset values via the user interface, they can be read in via a part program.
You can see one example of such a program above.
With these two programs, a bidirectional leadscrew error compensation is commissioned.
The errors occurring during execution must be acknowledged with a power-on reset.

$MA_CEC_ENABLE =MD32710: Values can only be read in while the compensation is deactivated.

$SN_CEC_TABLE_ENABLE =SD41300: Values can only be read in while the table to be written to is deactivated.

NEWCONF = softkey Set MD active

$AN_CEC[m.n] Compensation value, m: Table index, n:consecutive number of the value.

$AN_CEC_INPUT_NCU Only relevant for compensation across NCUs

$AN_CEC_INPUT_AXIS Only relevant if the table is used for sag compensation.

$AN_CEC_OUTPUT_NCU Only relevant for compensation across NCUs

$AN_CEC_OUTPUT_AXIS Only relevant if the table is used for sag compensation.

$AN_CEC_STEP Support point spacing

$AN_CEC_MIN Axis actual value of the start position

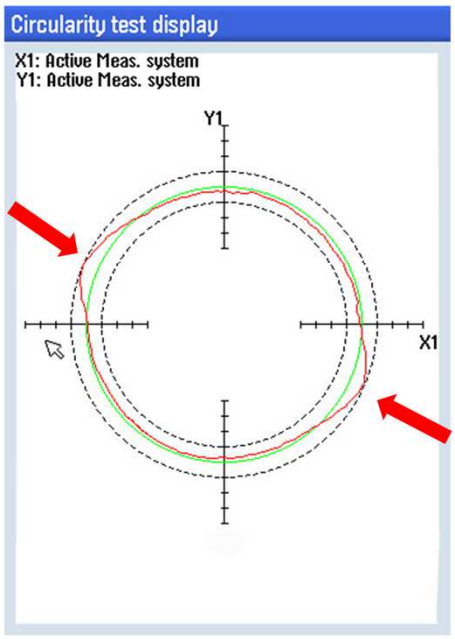
$AN_CEC_MAX Axis actual value of the end position

$AN_CEC_DIRECTION Traversing direction in which the compensation value is active; -1 or 1.

$AN_CEC_MULT_BY_TABLE Only relevant if the table is used for the sag compensation.

$AN_CEC_IS_MODULO Only relevant if the table is used for the sag compensation.

Friction compensation



 SITRAIN © Siemens AG 2017

Overview

In addition to the mass inertia and the machining forces, the friction forces in the gearing and guideways of the machine influence the dynamic behavior of a machine axis. In particular, when a machine axis is accelerated from standstill, the transition from static to sliding friction has an adverse effect. The resulting sudden change in the friction force results in a briefly increased following error. With interpolating axes, this results in significant contour violations. For circles, these occur especially at the quadrant transitions where one of the participating axes is stationary as its direction is reversed.
To improve the contour accuracy at these points, an additional setpoint pulse (velocity or torque setpoint pulse) is applied to the friction compensation when a machine axis is accelerated from standstill.

Constant compensation values

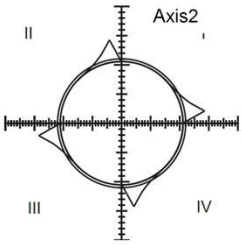
Depending on the acceleration of the machine axis, the same pulse is always applied to the velocity setpoint.
The amplitude and the decay time of the velocity setpoint pulse can be set.

Adaptive characteristic

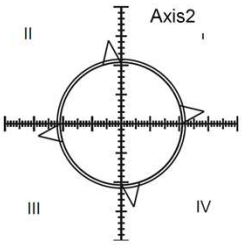
Depending on the acceleration of the machine axis, the velocity setpoint pulse is determined from one or more parameterizable characteristics.
Three different acceleration values can be set as well as the minimum and maximum amplitude and the decay time of the velocity setpoint pulse.

Friction compensation
With a constant compensation value

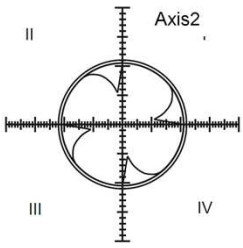
SIEMENS



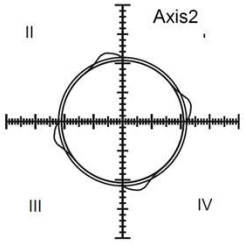
Uncompensated



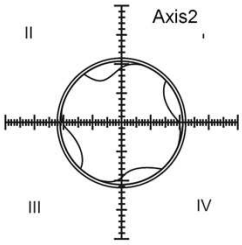
Amplitude too small



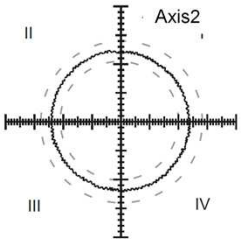
Amplitude too large



Decay time too short



Decay time too long

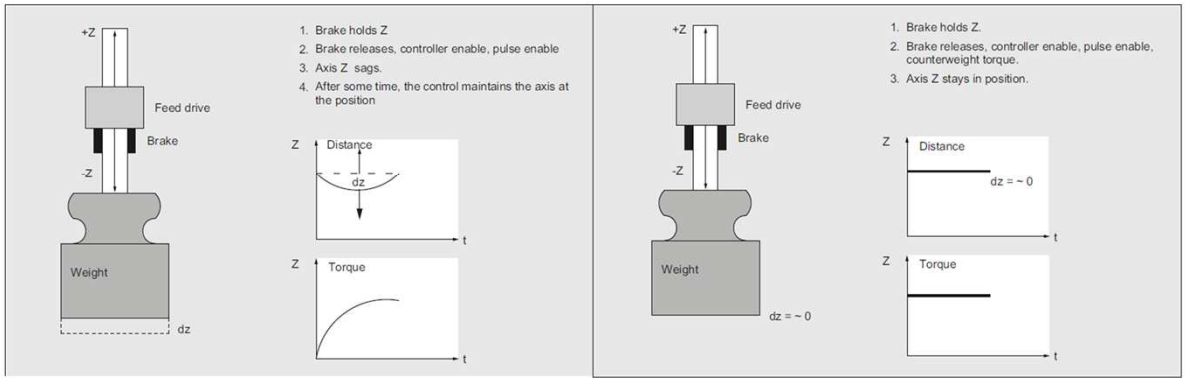


Well compensated

SITRAIN © Siemens AG 2017

MD32500=1	Friction compensation activated
MD32490=1	Friction compensation with a constant compensation value
MD32510=0	Without adaptation
MD32520	Amplitude of the velocity setpoint pulse
MD32540	Decay time of the velocity setpoint pulse

Vertical axes



Course NC-84SL-SIP Page 16 - 17 SITRAIN © Siemens AG 2017

Overview

When the brake of a vertical axis is released without mechanical weight compensation, this can make a movement downward because the torque that the axis needs to stay in position first has to be built up by the controller. The smaller the servo gain factor of this axis, the stronger the effect will be.
The electronic weight compensation ensures that the necessary torque is available immediately.
To achieve this, the necessary holding torque is measured with the brake released and manually entered in an additive torque setpoint

Torque actual value

Sinamics parameter r31. The current holding torque can be read off here

Torque limit, offset

Sinamics parameter p1532. The value of r31 is entered here to balance the torque limits.

Additive torque

Sinamics parameter p1513. The value of r31 is entered in this value with a sign. It produces the necessary holding torque as soon as the brake is released.

Delayed reboot

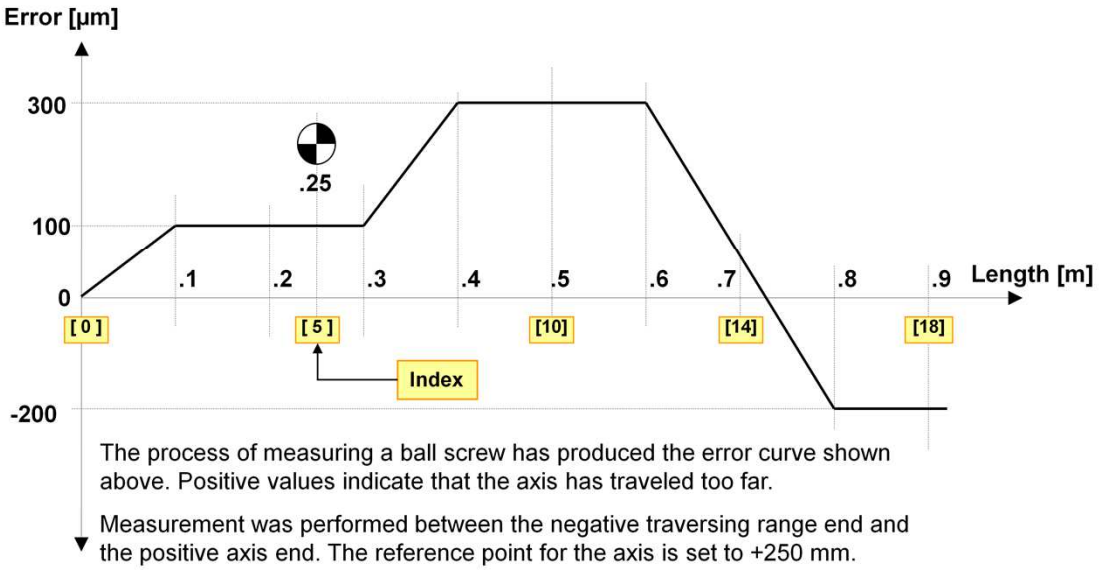
This is a measure to prevent sagging of vertical axes on a reset via the operator interface.
If there are vertical axes on the machine, failure of the closed-loop control while the control is starting will result in the axes failing to maintain height. With the "reboot delay" function, reset of the NC is delayed by a parameterizable time. During this time, for example, engaging the holding brakes of the vertical axes, can be performed.
Alarm 2900 "Reboot performed after delay" is output

MD10088

$MN_REBOOT_DELAY_TIME = Reboot delay time

Exercise: Leadscrew and measuring system error compensation

SIEMENS



The process of measuring a ball screw has produced the error curve shown above. Positive values indicate that the axis has traveled too far.

Measurement was performed between the negative traversing range end and the positive axis end. The reference point for the axis is set to +250 mm.

The compensation range should be between 0 and 900 mm.

 SITRAIN © Siemens AG 2017

Task:

The error curve shown in the diagram has been determined for the X axis. Perform error compensation with simple compensation.
Check its effectiveness using the axes service display in the "Diagnostics" operating area.

Solution:

The shape of the error curve permits a support point spacing of 100 mm. However, since the reference point is at the 250 mm position and compensation cannot start until referencing has been performed, the support point spacing must be reduced to 50 mm. This results in 18 intervals; i.e. 19 intermediate points.

Measuring system/leadscrew error compensation or
number of intermediate points for interpolatory compensation (SRAM) = 19

The compensation values start to be included after the reference point approach has taken place. According to the error curve, however, there is a pitch error requiring compensation at this point. To prevent any compensation from having to be incorporated at the reference point, the 0 line of the error curve needs to be moved up by 100 µm.

The offset values ENC_COMP[0,n,AXj] (n=0 .. 18) for compensation points 1 to 19 after the 0 line has been moved are as follows:

-0.1 / -0.05 / 0 / 0 / 0 / 0 / 0 / 0.1 / 0.2 / 0.2 / 0.2 / 0.2 / 0.2 / 0.075 / -0.05 / -0.175 / -0.3 / -0.3 / -0.3

SIEMENS



Chapter 17

Training and support

Course NC-84SL-SIP

Training documentation SITRAIN © Siemens AG 2017

Learning objectives 2
Do you still have any questions 3
www.siemens.com/sitrain 4
Learning path: SINUMERIK Fundamentals 5
Learning path: 840D sl service and commissioning 6
Learning path: 840D sl 7
Industry Online Support – 8
The navigation principle 9
The complete product information 10
mySupport – overview 11
Support Request 12
Industry Online Support Internet Portal – 14
Checkpoint 18
Forum in the Internet: CNC Arena 19
Siemens Service&Support Forum and CNC Arena are interlinked 20
CNC4YOU Portal of SINUMERIK 21
CNC4YOU Journal of SINUMERIK 22

Learning objectives

- You will be familiar with the Sinumerik course DVD
- You will be familiar with the learning paths for Sinumerik
- You will be familiar with the Sitrain website
- You will be familiar with Siemens online support



SITRAIN © Siemens AG 2017

Notes

Do you still have any questions regarding our training courses?

SIEMENS

We can help you!

... on the Internet:

www.siemens.com/sitrain

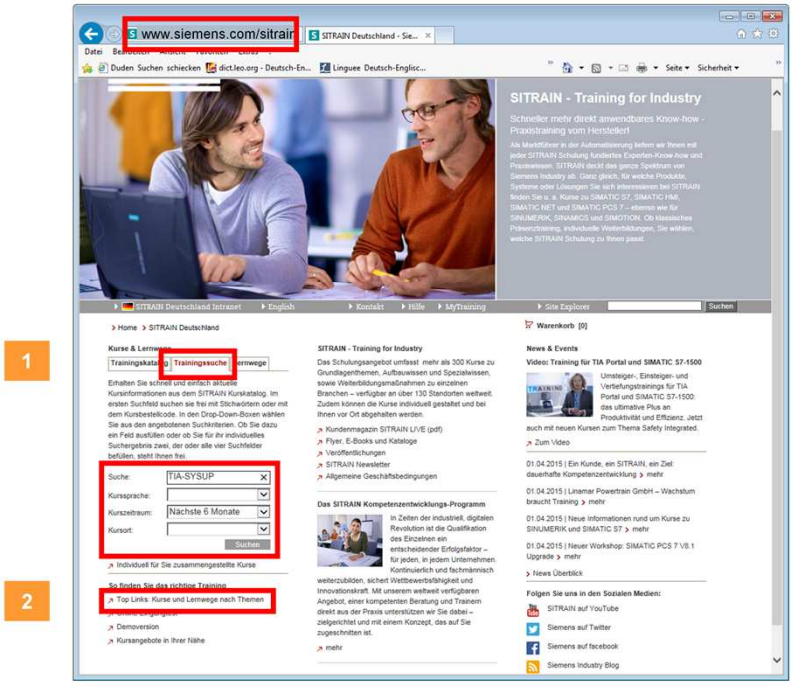
or by e-mail:

info@sitrain.com



SITRAIN © Siemens AG 2017

Notes



 SITRAIN © Siemens AG 2017

The full range of courses offered can be accessed using the following link:

www.siemens.de/sitrain or

www.siemens.com/sitrain

Search for a course The search function allows you to set various search filters such as keyword, target group, etc. to find the desired course. The filters can also be combined.

Top links Top links provide direct access to a variety of courses, for example SIMATIC S7, SINUMERIK solution line, etc.

Training news Here you will find the latest courses offered and SITRAIN activities at a glance.

Course catalog The course catalog enables you to find the desired course using training methods or via the organizational structure of the Siemens Mall.

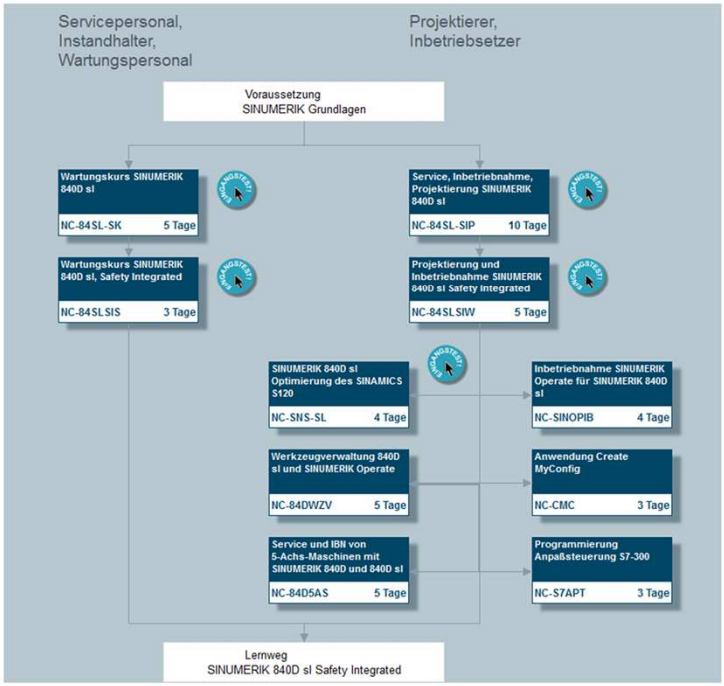
Learning path: SINUMERIK Fundamentals



SITRAIN © Siemens AG 2017

Notes

Learning path: 840D sl service and commissioning

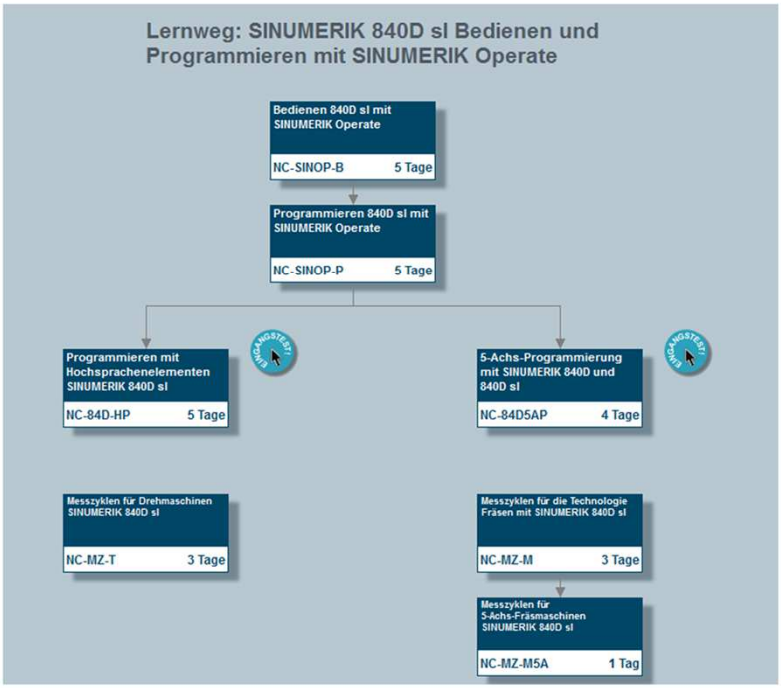


SITRAIN © Siemens AG 2017

Notes

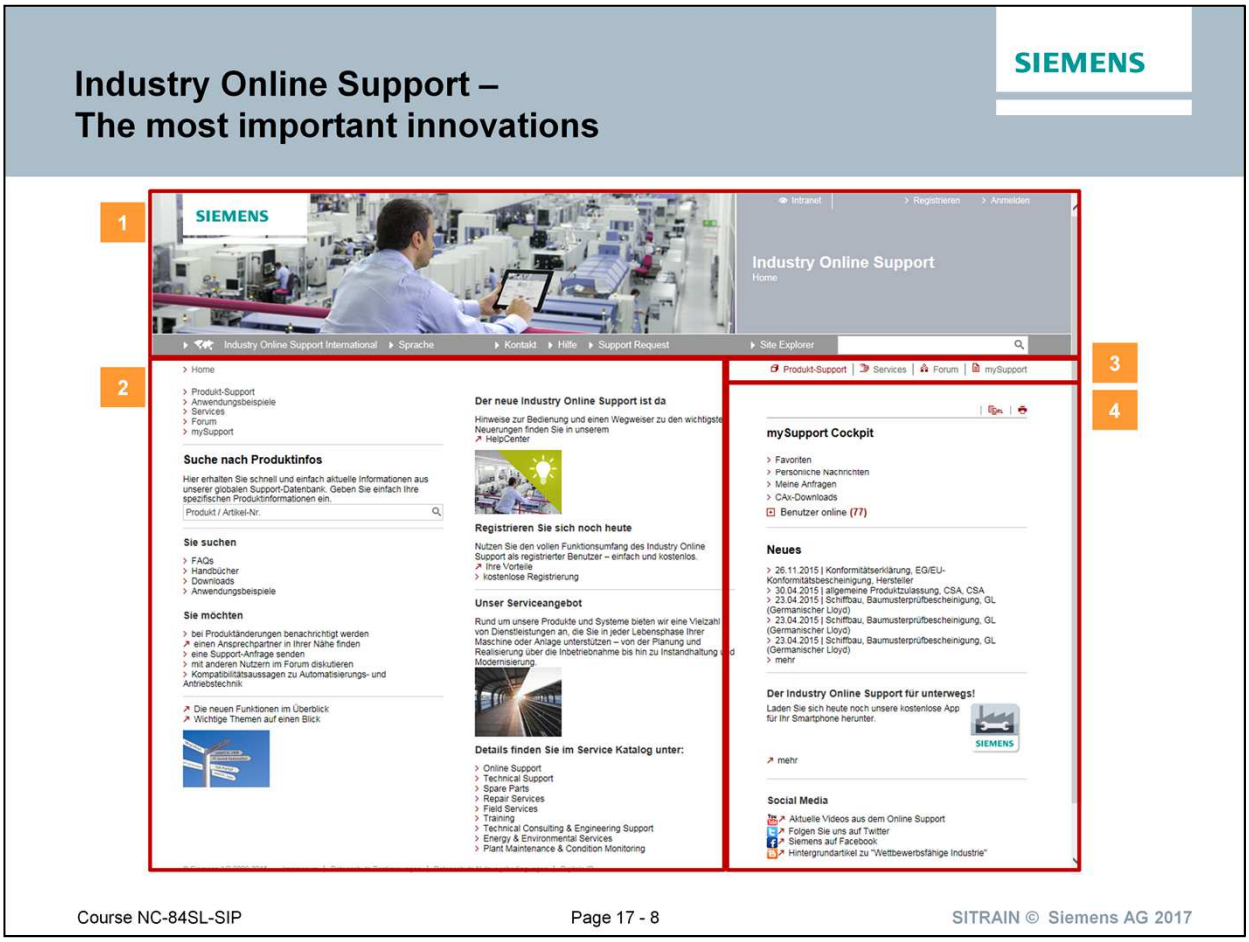
Learning path: 840D sl
Operating/programming SINUMERIK Operate

SIEMENS



SITRAIN © Siemens AG 2017

Notes



You will always find the most important functions in the same place on all pages:

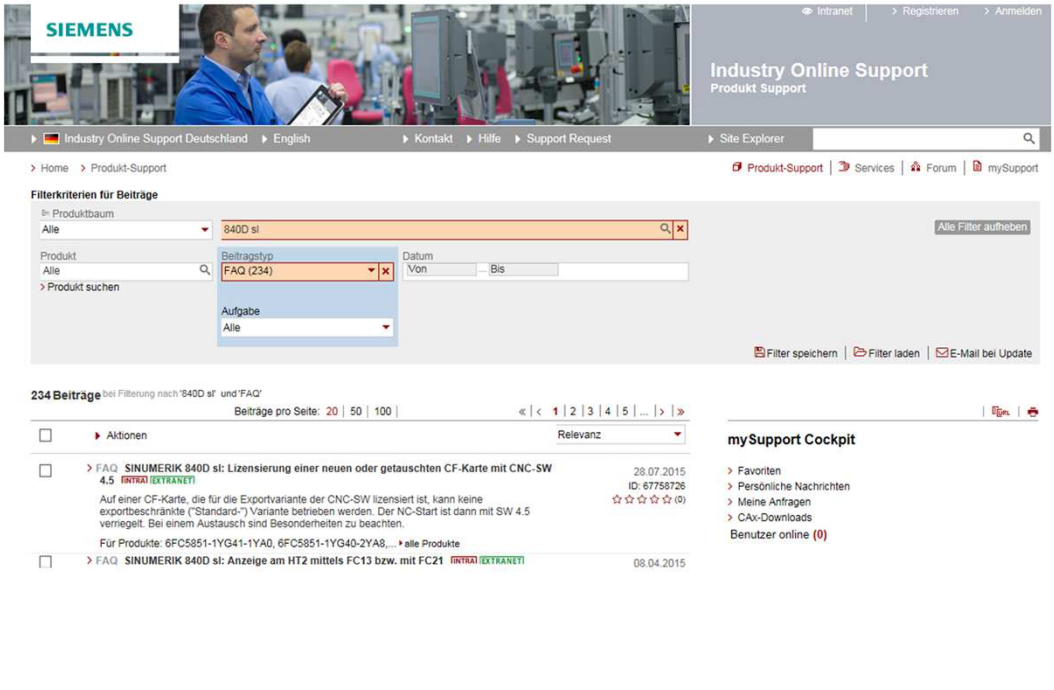
1. The menu bar contains the most important standard functions. And you can register and log on at any time to benefit from the new features of the personal area mySupport.

2. The contents are in the center. On the home page, you will find up-to-date information and links with which you can very quickly reach destinations in other areas of Online Support.

3. You can switch between the four central area of Online Support at any time: Product Support, Services, Forum, and mySupport.

4. And you will find your personal mySupport cockpit on every page. There you can see, for example, whether the status of your support queries has changed.

The navigation principle



Here you will find information on all current and also discontinued products, such as:

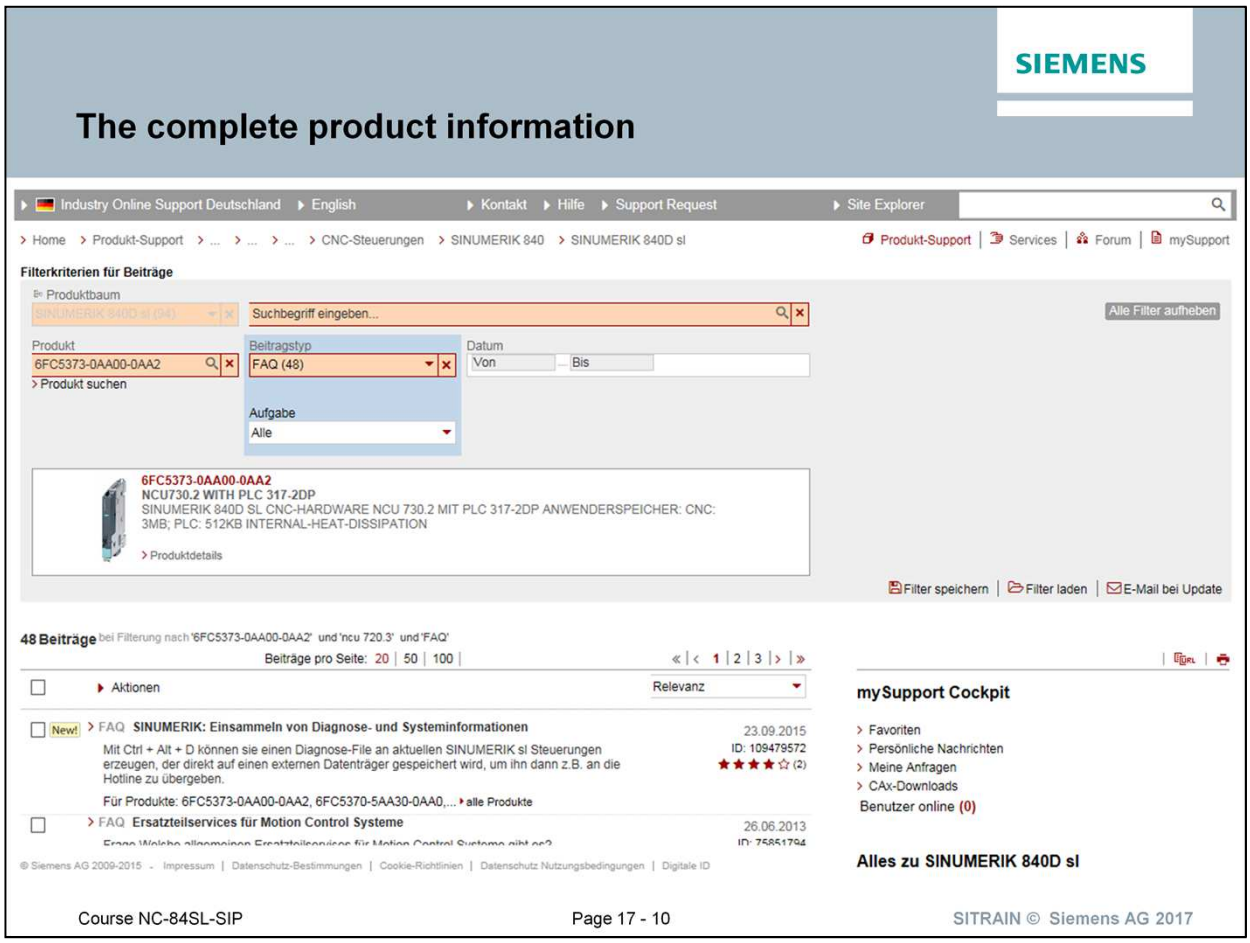
- Frequently asked questions (FAQ)
- Manuals and operating instructions
- Downloads
- Product notes (product announcements, product discontinuation, etc.)
- Certificates
- Characteristics
- Application examples

Access to these articles is now not just offered through the product tree but also via a central filter bar. Using the different filters, you can get to the relevant content with just a few clicks. The product tree has been moved into an equivalent filter, which means multiple filter steps can be combined in a clear and understandable way.

Preview numbers show you how many hits to expect before you apply the filter. This makes finding the relevant information must simpler and more efficient.

For your search you can, for example, combine the product tree, a search term, and a document type.

There are no hidden search parameters: All settings and results are displayed to you clearly.



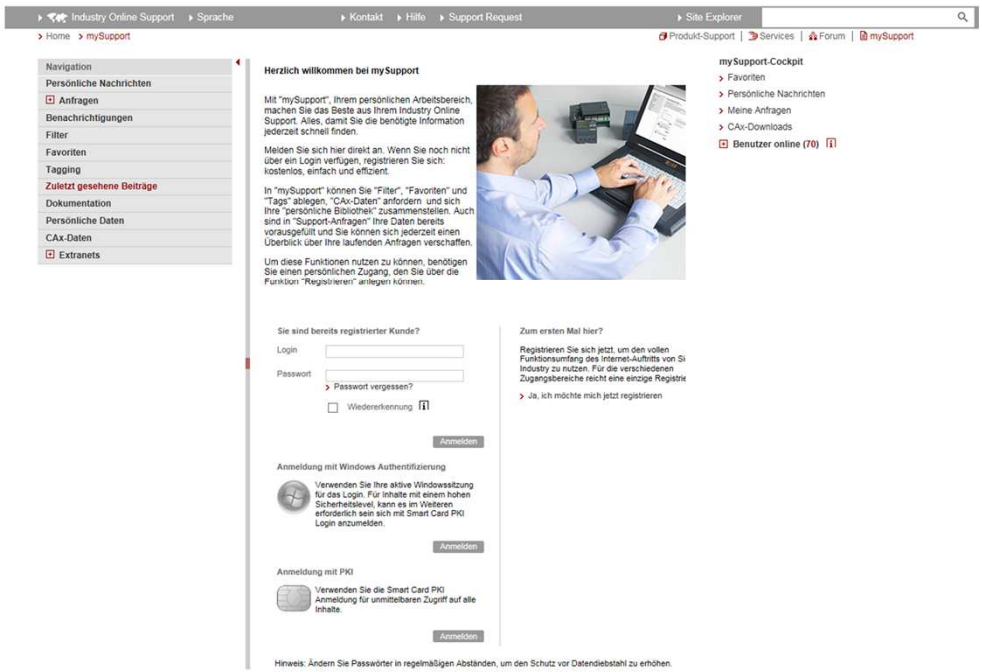
One powerful function of the new Industry Online Support is the direct path to the complete product information. Use this if you are searching for all technical information about a Siemens product simply and quickly, for example, to compare products when you are expanding your plant or replacing individual components. This is how it works:

In the Product Support area, you will find the central navigation bar.
To select a product, simply choose the "Product" filter. Here, you enter an article number or a product name. You are supported by a dynamic display of matching products (suggestion list).

With one click, you can obtain the details of the selected product - always up to date:

- Product life cycle info, comprising a milestone with a date (e.g. delivery release, product discontinuation, ...) You will learn whether the selected product is a current product or whether the product is already being phased out.
- Information about a successor product for phase-out products and further developments. If there is a successor product, you will have a direct link to the product information on this product.
- Technical data, clear, compact, and complete. This gives you all available technical data about the selected product: dimensions, operating voltages, or number of inputs/outputs, etc.

mySupport – overview



The mySupport area is and remains your personal work space. Use it to get the very best out of your Industry Online Support experience.
The most important aspect first: If you are already working with mySupport today, we will transfer all of your personal data that you have entered and the stored information into the new Industry Online Support.
In this area, you compile the information that is important for you to perform your daily tasks and we provide you with the suitable tools. Create your folder structures and store the information there, such as Favorites. This can be project-oriented or product-oriented, or however you want it.
You can now also add notes, comments, and tags (descriptors). With the so-called "tag cloud", that the system automatically creates based on your inputs, you can very simply and quickly access the information via your own descriptors. The operation of mySupport is standardized so that you can find your way around easily. You can also use "drag & drop", just as you are used to in your operating system.

As soon as you have logged on, the mySupport cockpit will always be at hand. For example, there you can see immediately when the status of a support request changes or you have a new personal message. There, you also have direct access to your personal tags in the tag cloud, to the articles you last visited and you can see which users are online at that instant.

What else can you expect in the new mySupport? Here are a few further points:

- The previous MyDocumentationManager is now integrated completely into mySupport with the name "mySupport Documentation". This new function category "Documentation" contains all functions of the MyDocumentationManager and provides some additional new features.
- The Service & Support Newsletter has been completely revised: An individual messaging system will provide all that it ever did and more.

Support Request



 SITRAIN © Siemens AG 2017

Support Request

The new Support Request is part of mySupport and is your most important input channel when you need support in your work with Siemens Industry products.

The Support Request offers you

- direct access to technical experts
- suggested solutions for your questions
- status tracking of your requests

So that we can answer your request as fast and competently as possible, we require

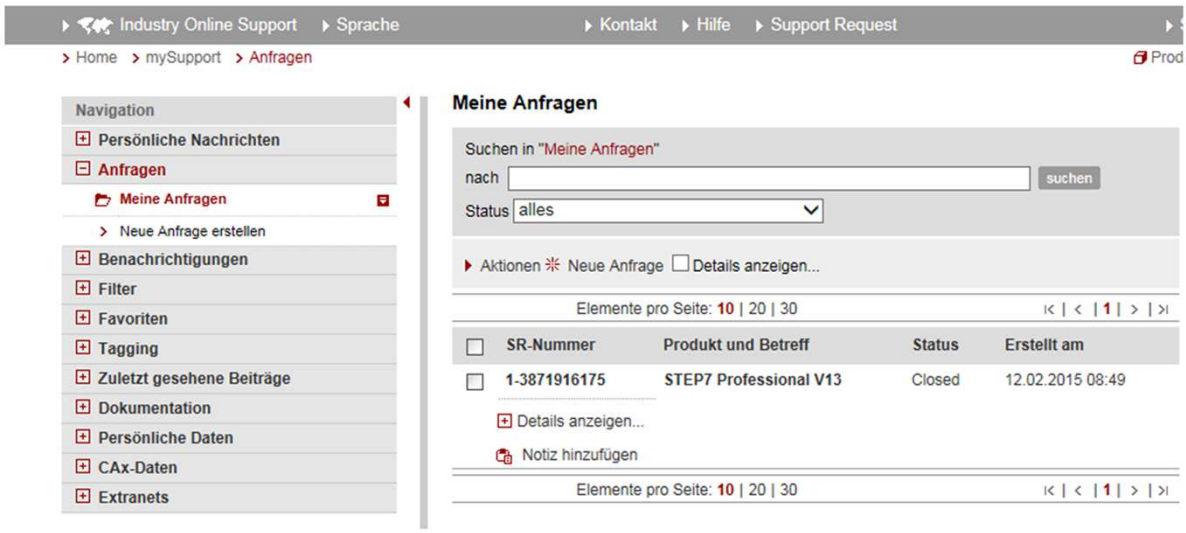
- precise details about the product used
- precise details relating to your question (e.g. version number, system environment, etc.)
- and, of course, your contact data.

So that you do not forget anything, the software guides you through all the necessary steps. While you are creating a request, you are already offered suggested solutions in the form of

- FAQs, application examples
- manuals or from current
- product announcements and
- solutions from our community (technical forum).

The suggestions provided are solutions from our Product Support. They were created by our technical experts and have been especially evaluated as being helpful for your or similar issues. Maybe this support will already solve your problem.
If not, an expert from our "Technical Support" will contact your immediately and answer your inquiry competently – in a direct and straightforward way.

Support Request

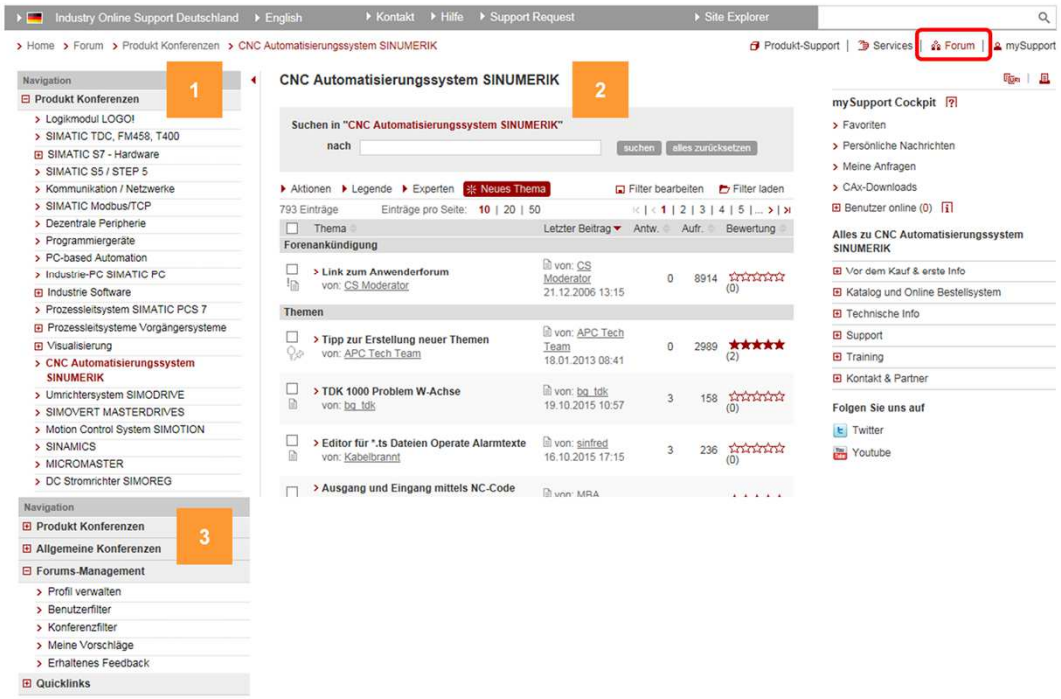


SITRAIN © Siemens AG 2017

Notes

Industry Online Support Internet Portal – Forum

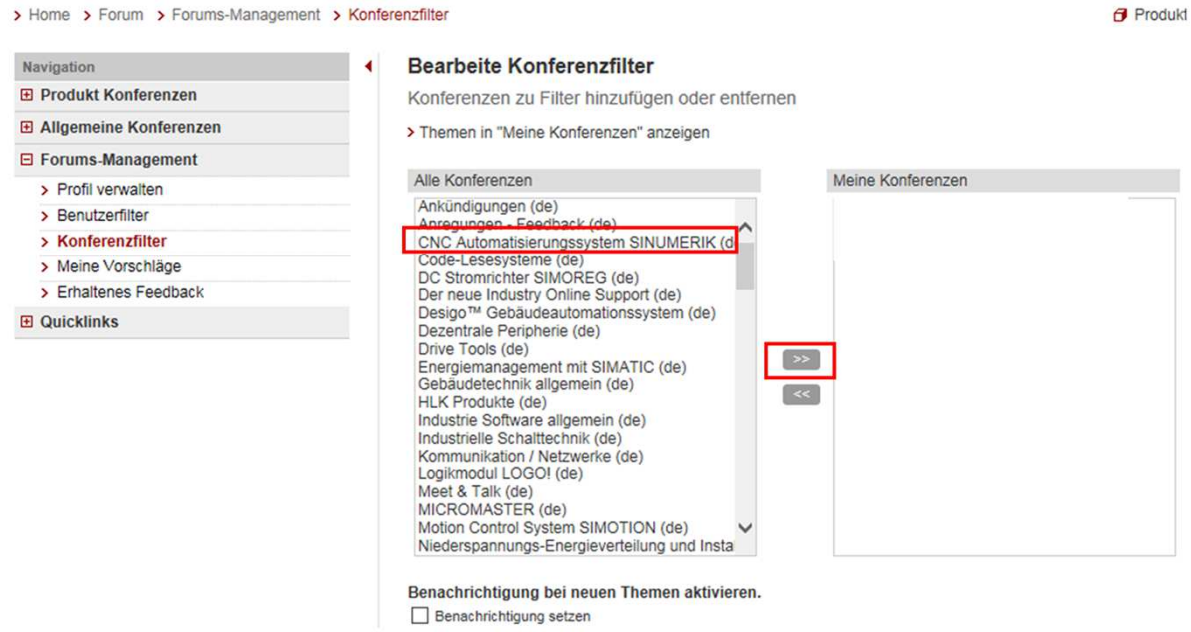
SIEMENS



SITRAIN © Siemens AG 2017

Notes

Industry Online Support Internet Portal – Conference filter

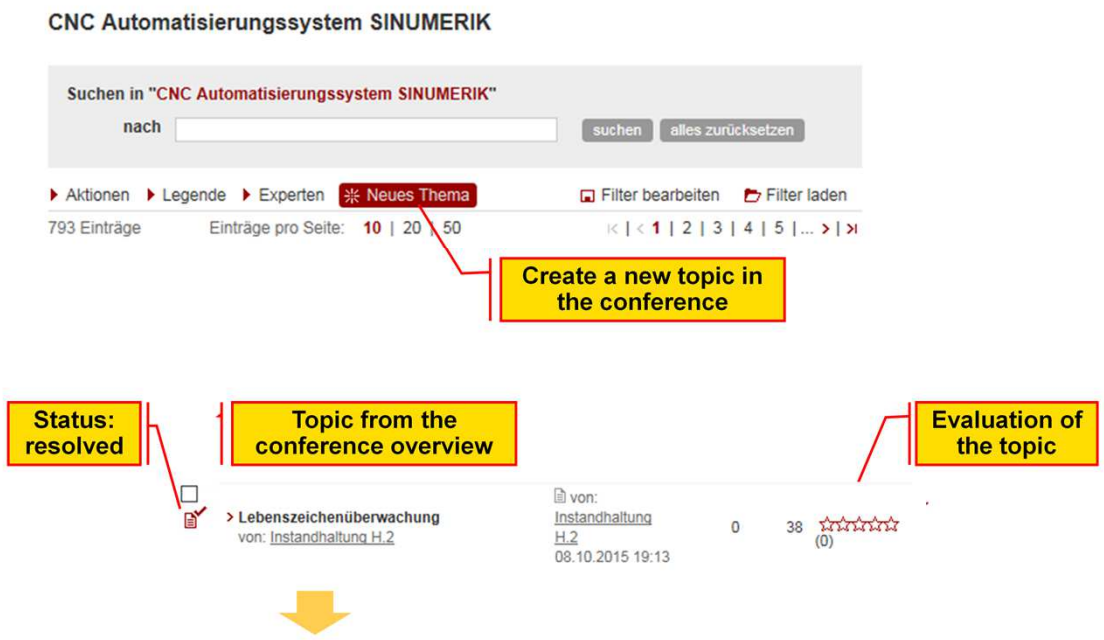


SITRAIN © Siemens AG 2017

Notes

Industry Online Support Internet Portal – Interaction in the conference view

SIEMENS

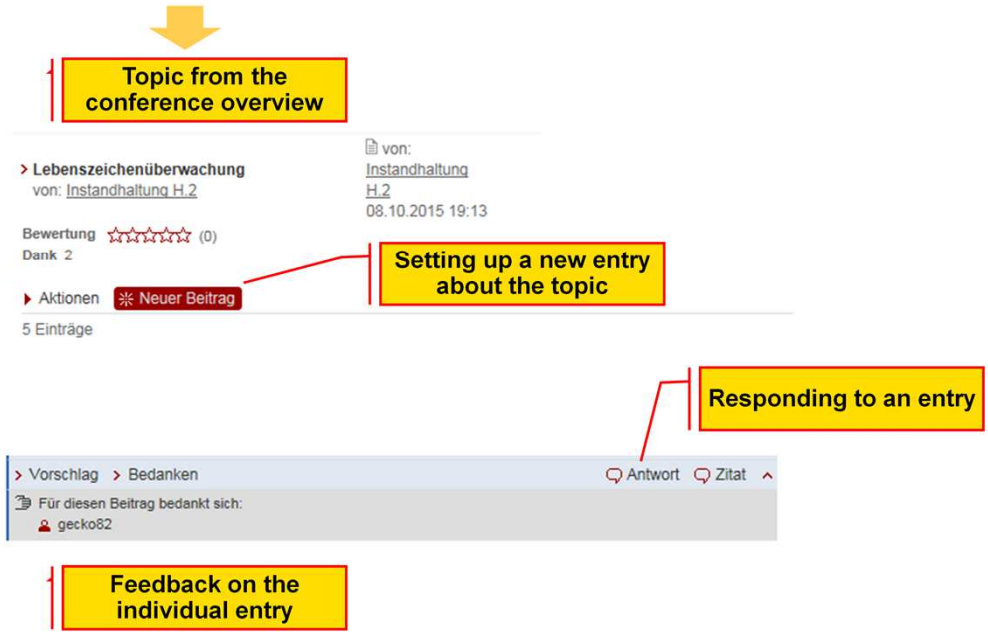


 SITRAIN © Siemens AG 2017

Notes

Industry Online Support Internet Portal – Interaction in the Forum

SIEMENS



 SITRAIN © Siemens AG 2017

Notes

Checkpoint



Denken Sie über folgendes nach:

- **Nennen Sie ein paar Gründe, warum eine Registrierung bei mySupport sinnvoll ist.**
- **Wie können Sie die erforderlichen Handbücher stets auf dem aktuellen Stand halten?**

SITRAIN © Siemens AG 2017

Notes

Forum in the Internet: CNC Arena



- Communication platform for users and application engineers
- Available in German and English
- Rapidly growing number of members, approx. 50,000
- Trade visitors approx. 50,000 per month
- Ranking No. 1 in Google for the term "CNC"

www.siemens.cnc-arena.com

Application engineers, users, service technicians, high-school and university students use the Internet media

SITRAIN © Siemens AG 2017

Notes

Siemens Service&Support Forum and CNC Arena are interlinked

SIEMENS

Central active moderation of both forums



CNC Arena: Only application questions

Link commissioning/service for Siemens&Support

www.siemens.cnc-arena.com



Siemens Service&Support: Only commissioning/service/ application

Link to application questions to the CNC Arena

www.automation.siemens.com

Important communication platforms for beginners and advanced users (e.g. commissioning OEM, service, maintenance)

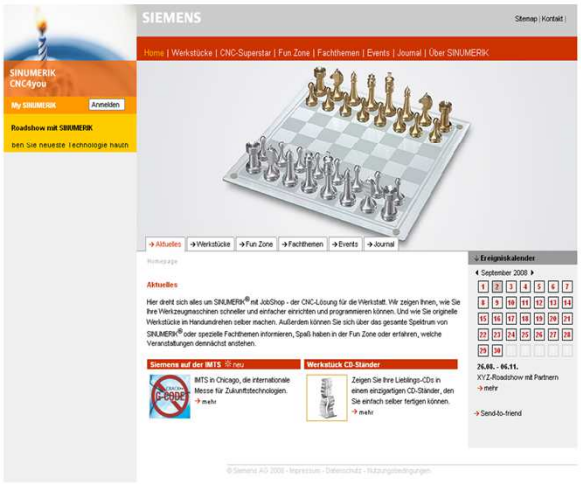
SITRAIN © Siemens AG 2017

Notes

CNC4YOU Portal of SINUMERIK

SIEMENS

Up-to-date information at any time!



www.siemens.com/cnc4you

- Workpieces
- Up-to-date information
- Technical topics
- Tips & Tricks
- Event calendar
- Fun Zone
- …

SITRAIN © Siemens AG 2017

Notes

CNC4YOU Journal of SINUMERIK

SIEMENS

Compact information to take away and read!



- Tips & Tricks
- Technical topics
- From practical applications
- News
- …

SITRAIN © Siemens AG 2017

Notes

SIEMENS TEST PROTOCOL

Machine Test Protocol Sinumerik 840D sl

Product / Version	Company: Address:
Sinumerik 840D sl SW4.7	

Machine Range	Machine Type - Features	Spindle Config. MLFB	KW	Axis Conf MLFB	N/M

Electrical Cabinet Protocol attached	☐	Safety Integrated Protocol Attached	☐

	Distribution list				
Name	Company / Department	e-mail	Name	Company / Department	e-mail

Title **Machine Test Protocol Sinumerik 840D sl**	
Document no.	Date
File name:	**Protocol version 1.2**

Copyright © Siemens AG 2013 All Rights Reserved.

Contents

1 RESPONSIBLE PERSONS: 5

1.1 Schedule 5

1.2 Participants carrying-out the test 5

2 SUMMARY AND RESULT 6

2.1 Summary 6

2.2 Result 6

2.2.1 Exception List 6

2.3 Release Siemens 6

2.4 Release from Machine Tool Builder 6

3 BACK UP 7

3.1 Modified data 7

3.2 Data Backup 7

4 CHANGE LOG 8

4.1 Changes to the machine. 8

5 SYSTEM DESCRIPTION 9

5.1 Machine 9

5.2 Control 9

Copyright © Siemens AG 2012 All Rights Reserved.

6 TEST OBJECTIVE 10

6.1 Note 10

7 FUNCTIONS TESTED 11

8 DETAILED TEST 13

8.1 Machine area 13
8.1.1 Jog → Ref 13
8.1.2 Jog → MCP 15
8.1.3 Jog → HMI Softkeys 18
8.1.4 MDA → Teachin 20
8.1.5 MDA → HMI Softkeys 21
8.1.6 MDA → MDA → Teachin 23
8.1.7 AUTO 24

8.2 Technology 26
8.2.1 Milling General 26
8.2.2 Milling 3+2 27
8.2.3 Milling 5 Axis 28
8.2.4 Turning Basic 29
8.2.5 Turning Advanced 30

8.3 Parameter 32
8.3.1 Machine Builder Options 32
8.3.2 Measure tool 32
8.3.3 Setting data 32
8.3.4 Magazine 33
8.3.5 Tool List 34

8.4 Geometry 35
8.4.1 Circularity test HMI 35
8.4.2 Renishaw circularity test 35

8.5 Safety Integrated 35
8.5.1 Safety related test 35

Copyright © Siemens AG 2012 All Rights Reserved.

8.6 Password protection....36
8.6.1 Securing data areas....36

8.7 Various....37

Copyright © Siemens AG 2012 All Rights Reserved.

1 Responsible persons:

1.1 Schedule

Test carried out at (**OEM Name**):

From :

To:

1.2 Participants carrying-out the test

Name	Company / Department

Copyright © Siemens AG 2012 All Rights Reserved.

2 Summary and result

2.1 Summary

2.2 Result

Release of the machine recommended (YES/NO/Reason): _____

2.2.1 Exception List

The machine has been released for sale with the following exceptions:

Function:	Test Number	Remark

2.3 Release Siemens

Machine can be released for Sale: Confirm with Signature

Siemens			
Date	Author	Company / Department	Signature

2.4 Release from Machine Tool Builder

Machine Manufacturer			
Date	Name	Company / Department	Signature

Copyright © Siemens AG 2012 All Rights Reserved.

3 Back up

3.1 Modified data

Machine data number	Original Value	New Value

3.2 Data Backup

Create data backup after completing the test and attach it to the protocol.

Appendices:

- _____ (NC Archive)
- _____ (PLC Archive)
- _____ (Drive Archive)
- _____ (HMI Archive)
- _____ (Complete Archive)
- _____ (Clone TGZ file for series production)
- _____ (Image for restore)
-

Copyright © Siemens AG 2012 All Rights Reserved.

4 Change Log

4.1 Changes to the machine.

Date	Change	Reason	Tested	Responsible person	Signature
			☐		
			☐		
			☐		
			☐		
			☐		
			☐		
			☐		
			☐		
			☐		
			☐		
			☐		

Copyright © Siemens AG 2012 All Rights Reserved.

5 System description

5.1 Machine

Type	
Serial No.	
Configuration	
Axes	
Spindle	
Measuring devices	
Miscellaneous	

5.2 Control

NC type	Sinumerik 840D
Serial No.	
Machine control panel	
Software version:	
User software	
PLC Application	

Copyright © Siemens AG 2012 All Rights Reserved.

6 Test objective

6.1 Note

The test plan extends the commissioning documentation of 840D sl with Sinumerik Operate.

The application is tested with the current test plan, which helps to achieve for the most part an error-free interaction of SINUMERIK with the machine. No guarantee for fault-free machine application can be guaranteed.

Siemens is not responsible for the PLC safety related functions which are implemented on the Machine. The Machine tool builder is responsible in satisfying the receiving countries “Safety regulations regarding CNC machine tools” regulations.

Basic knowledge of operation and programming of 840D is necessary for carrying out the test. In case of some test tasks, it is necessary to start an automatic program or to trigger the machine functions. It is, therefore, explicitly indicated that all operations that trigger machine functions should only be carried out by the employee of the machine manufacturer, who is authorized to do this.

The necessary programs are to be created directly at the machine.

ALL RELEVANT TESTS HAVE TO BE CARRIED OUT WITH THE FOLLOWING CONDITIONS:

1)	**Siemens Code(DIN) and ShopMill/ShopTurn with →**	**Metric system of units**
2)	**Siemens Code(DIN) and ShopMill/ShopTurn with →**	**Inch System of Units**
3)	**ISO / dialect with →**	**Metric system of units**
4)	**ISO / dialect with →**	**Inch system of units**

Copyright © Siemens AG 2012 All Rights Reserved.

7 Functions tested

Confirm the functions as per the test plan

Test .	Function	Result / Accepted/To Do
8.1	**Machine**	
8.1.1	Jog → Ref	
8.1.2	Jog → MCP	
8.1.3	Jog → HMI Soft keys	
8.1.4	MDA → Teachin	
8.1.5	MDA → HMI Soft keys	
8.1.6	MDA → MDA → Teachin	
8.1.7	AUTO → DRF, Jog	
8.2	**Technology**	
8.2.1	Milling General	
8.2.2	Milling 3+2	
8.2.3	Milling 5 Axis	
8.2.4	Turning Basic	
8.2.5	Turning Advanced	
8.3	Parameter	
8.3.1	Machine Builder Options	
8.3.2	Measure Tool	
8.3.3	Setting data	

Copyright © Siemens AG 2012 All Rights Reserved.

Test .	Function	Result / Accepted/To Do
8.3.4	Magazine	
8.3.5	Tool List	
8.4	**Geometry**	
8.4.1	Circularity test HMI	
8.4.2	Renishaw circularity test	
8.5	**Safety integrated**	
8.5.1	Safety related test	
8.6	**Password protection**	
8.6.1	Securing data areas	
8.7	**Various**	
8.7.1	General tests	

Copyright © Siemens AG 2012 All Rights Reserved.

8 Detailed Test

8.1 Machine area

8.1.1 Jog → Ref

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Machine Start-up in Jog-REF	☐		☐		
T+M	2	Automatic referencing (Jog - REF Cycle start)	☐		☐		
T+M	3	Reference not necessary (Absolute measuring system)(Start-up in JOG)	☐		☐		
T+M	4	Prevention of Axis faults, when starting in axis operation, the Spindles with axis functionality should be referenced during the referencing procedure.	☐		☐		
T+M	5	Additional equipment requiring reference point approach should be referenced prior to the first NC Start (Turret). “Alarm prevention”	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

T+M	6	Prior to referencing it should be possible to change to JOG and manually move the axis to a safe position. If the axis is resting on the Hardware limit switch, it should be possible to JOG the axis away in the opposite direction. The message "Axis lies on Hardware end limit" should remain even after powering the machine off and on.	☐		☐			
T+M	7	Restarting the reference procedure (interruption / E-Stop) should be possible without having to switch the machine off and on.	☐		☐			
T+M	8	Repeating the referencing procedure should be possible, after reaching reference point - repeating the process without a fault message.	☐		☐			
T+M	9	The feedrate and Spindle enable signals should not be active immediately after power-on of the machine. The operator must enable these signals with the respective key presses on the MCP	☐		☐			

Copyright © Siemens AG 2012 All Rights Reserved.

8.1.2 Jog → MCP

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	If the axis is resting on the Hardware limit switch, it should be possible to JOG the axis away in the opposite direction.	☐		☐		
T+M	2	It should be possible in the RESET state to change to JOG mode	☐		☐		
T+M	3	When changing from REF to Jog - Operator must select again the required axis. There should be no pre selected axis.	☐		☐		
T+M	4	Select axis - the axis should travel in the defined direction when the (+- Pushbutton) is pressed. The axis should travel with the feedrate defined in the NC machine data.	☐		☐		
T	5	With Turning machines it should also be possible to move more axes at the same time. The machine builder has to realize this in the PLC.	☐		☐		
M	6	With Milling machines it is also possible to Jog axes simultaneously; however the machine builder has to realize this in the PLC.	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

T+M	7	The machine builder should also ensure that the rotary axis can be jogged, including rotary axis functionality with Spindles. C+ button = spindle direction M4 C- button = spindle direction M3	☐		☐			
T+M	8	The feedrate override should be effective on all axes which can be Jogged. All axes should come to standstill when the feed override is set to zero.	☐		☐			
T+M	9	When the feed stop button is pressed the Jog plus and minus buttons should have no effect on any axis motion. The axis should all come to a stop.	☐		☐			
T+M	10	When the spindle stop button is pressed the Jog plus and minus buttons should have no effect on the spindle motion. The spindles should all come to a stop.	☐		☐			
T	11	When Spindle direction buttons are available, the spindle should move clockwise or anti-clockwise dependent upon which button is press. The default speed is defined in the machine data.	☐		☐			
T+M	12	The Spindle override should correct the currently controlled spindle/s.	☐		☐			
T+M	13	Do all axis move according to the work plane, defined in the DIN standard.	☐		☐			

Copyright © Siemens AG 2012 All Rights Reserved.

T+M	14	When direction keys are pressed simultaneously, the axis in motion should stop immediately. Further motion can only be achieved when both buttons are once again released, and the desired direction button pressed again	☐		☐			
T+M	15	WCS >><< MCS With key not pressed (MCS) the axis should be moved using the Machine axis interface, and the axis will move on an individual basis, without active rotations. With key pressed (WCS) the axis should be moved using the Channel axis interface, and the axis will move on an individual basis, with active rotations.	☐		☐			
T+M	16	Repos With interruption in Automatic, it should be possible to switch to Jog, move axis away from the contour, subsequently return to Automatic / Repos and Jog the axis back to the previous position.	☐		☐			

Copyright © Siemens AG 2012 All Rights Reserved.

8.1.3 Jog → HMI Soft keys

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	T,S,M → T-D Select a tool from the tool list, using SK TOOL and IN MANUAL. Press cycle start tool should be transferred into the spindle or positioned before the work piece. The tool must not rotate if already present in the spindle or present before the work piece. Repeat the process with other tools from the magazine. Prior to changing from a driven tool, the rotating tool should be stopped prior to changing.	☐		☐		
T+M	2	Spindle → Set a spindle speed and activate "Cycle Start" (together with spindle M function) Repeat the procedure with further spindles.	☐		☐		
T+M	3	Spindle gear change→ Change gear, if a gear box is available on the machine.	☐		☐		
T+M	4	Spindle direction → Program M3 and M4 on all spindles which are available on the machine and ensure for the programmed code that spindle rotates in the correct direction.	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	5		☐		☐		
T+M	6	Measure Tool → Manual → Test with all types of possible tools Automatic → Test with all types of possible tools	☐		☐		
T+M	7	Position → All axis must be available, ensure that it is possible to move the spindle (C-Axis function) without first running in spindle operation	☐		☐		
T+M	8	MCS / WCS → Should change only the display from MCS/WCS	☐		☐		
T+M	9	Hand wheel → Test in both MCS and WCS Syncron-actions→. show currently active SYNACT Settings → Test and acknowledge functionality of the Setting data's	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.1.4 MDA → Teachin

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Teach-in → It should be possible to Jog the axis in MDA - Teach-in	☐		☐		
	2		☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.1.5 MDA → HMI Softkeys

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Program Control → When the PRT checkbox is set the axis should work internally, and not physically move the axis. The signal should work only in Automatic mode - when changing to Jog/MDA the signal should not be active except when an external switch is used.	☐		☐		
T+M	2	Program Control → DRY - When selected, the axis should move with the feedrate which is defined in Settings. This should only be available in Automatic.	☐		☐		
T+M	3	Program Control → M01 - When selected, the next occurrence of M01 in the NC program should cause the NC program to stop and wait for the next NC start.	☐		☐		
T+M	4	Program Control → When DRF is selected it should be possible in the Automatic mode of operation to move the axis with the Hand wheel, the position will be stored in the DRF offset.	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	5	Program Control → SKP - When selected, the next NC block with the Skip block instruction will not be executed.	☐		☐		
T+M	6	G functions → G - Functions for the reset state of the machine - To be checked for: Turning Milling	☐		☐		
T+M	7	Auxiliary functions → Machine tool builder should check that the functions which he does not allow in MDA are not processed.	☐		☐		
T+M	8	>> Extend SK → All G functions G - Functions for the reset state of the machine - To be checked for: Turning Milling	☐		☐		
T+M	9	> Extend menu → Hand wheel → Move axis using Hand wheel in the machine axis interface or using Geoax interface in connection with Teach-in mode	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.1.6 MDA → MDA → Teachin

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	MDA → Tool change in MDI window - T01 + Cycle start for Turning machines T01 M06 + Cycle start for Milling machines.	☐		☐		
T+M	2	MDA → It should not be possible to Jog the axis in MDA - exception : Teach-in is active	☐		☐		
T+M	3	Teach-in → Hand wheel → Move axis using Hand wheel in the machine axis interface or using Geoax interface in connection with Teach-in mode	☐		☐		
	4		☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.1.7 AUTO

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Move axis using Hand wheel/Jog/Inc in the machine axis interface to generate a DRF offset	☐		☐		
T+M	2	Jog in Automatic → Jog Mode Mask is set to 0 as standard, which means you are not able to Jog the axis in Automatic, with the exception of Teach-in. Jog Mode Mask is set to 1, means you are able to Jog the axis in Automatic. The machine builder should explicitly set this machine data when he requires this functionality.	☐		☐		
T+M	3	AUTO → Cycle start	☐		☐		
T+M	4	AUTO → Cycle Stop	☐		☐		
T+M	5	AUTO → Cycle single block	☐		☐		
T+M	6	AUTO → Program reset	☐		☐		
T+M	7	AUTO → Feed Stop in Program effect	☐		☐		
T+M	8	AUTO → Spindle stop in Program - effect	☐		☐		
T+M	9	AUTO → Feed override = 0	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

T+M	10	AUTO → Spindle override = 50	☐		☐		
T+M	11	AUTO → Effect of E-Stop	☐		☐		
T+M	12	Program control → PRT - Program Test	☐		☐		
T+M	13	Program control → DRY - Dry run feedrate	☐		☐		
T+M	14	Program control → M01 - Program stop	☐		☐		
T+M	15	Program control → DRF - Differential hand wheel offset	☐		☐		
T+M	16	Program control → SKP - Block skip	☐		☐		
T+M	17	Block search → M,S,T,H - Effect Search to Tool	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.2 Technology

8.2.1 Milling General

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
M	1	ShopMill Licensed	☐		☐		
M	2	Customer Cycles CUST800 etc	☐		☐		
M	3	Settings for Technology and measuring cycles	☐		☐		
M	4	Settings for Tool management views	☐		☐		
M	5	Cycle settings when multitasking	☐		☐		
M	6	Specific customer cycle CUST832	☐		☐		
M	7	3/5 Axis M Dynamics MD Test	☐		☐		
M	8	Tool change with DRF Deactivation M6	☐		☐		
M	9		☐		☐		
M	10		☐		☐		
M	11		☐		☐		
M	12		☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.2.2 Milling 3+2

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
M	1	Settings for Swivelling	☐		☐		
M	2	Tool change with active CYCLE800 / Transmit / Tracyl	☐		☐		
M	3	Settings Transmit / Tracyl	☐		☐		
M	4	Reset State with Head/ Table / Mixed kinematic	☐		☐		
M	5	Power off state / Head / Table / Mixed kinematic	☐		☐		
M	6		☐		☐		
M	7		☐		☐		
M	8		☐		☐		
M	9		☐		☐		
M	10		☐		☐		
M	11		☐		☐		
M	12		☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.2.3 Milling 5 Axis

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
M	1	Settings for Traori	☐		☐		
M	2	Settings for CUST832	☐		☐		
M	3	Correct Trafo Type set	☐		☐		
M	4	Tool Change with active Traori	☐		☐		
M	5	Reset State with Head/ Table / Mixed kinematic	☐		☐		
M	6	Power off state / Head / Table / Mixed kinematic	☐		☐		
M	7	Settings for Orientation / ORIAXES reset ORIEULER ORIMKS ORIWKS reset	☐		☐		
M	8		☐		☐		
M	9		☐		☐		
M	10		☐		☐		
M	11		☐		☐		
M	12		☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.2.4 Turning Basic

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T	1	Is the coordinate system displayed correctly in the editor?	☐		☐		
T	2	Are the softkeys for the standard cycles displayed in the editor?	☐		☐		
T	3	Is the “plunge-cutting” and “plunge-turning” softkey displayed in the editor?	☐		☐		
T	4	Are the softkey for the remaining-material machining displayed in the editor?	☐		☐		
T	5	Is the “Tailstock” parameter displayed in the program header? (only in ShopTurn)	☐		☐		
T	6	Is the “parts gripper” parameter displayed in the cut-off cycle?	☐		☐		
T	7	Does the parts gripper in the cut-off cycle function correctly?	☐		☐		
T	8	Is a program simulated correctly?	☐		☐		
T	9	Is a simulated program represented correctly in the 3D view?	☐		☐		
T	10	Is a program simulated correctly with a open working-area door?	☐		☐		
T	11	Is the simulation aborted when the “Reset” key is pressed?	☐		☐		
T	12	Is the calculated program run time updated during the simulation?	☐		☐		
T	13	Is a program executed correctly when you press “Cycle Start”?	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

T	14	Is the “Base block” softkey displayed?	☐		☐		
T	15	Is the ”Simultaneous recording” softkey displayed?	☐		☐		
T	16	Does “Block search” work right?	☐		☐		
T	17	Does ”REPOS” work right?	☐		☐		

8.2.5 Turning Advanced

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T	1	Is a Face milling contour milled correctly on main spindle?	☐		☐		
T	2	Is a Face milling contour milled correctly on counterspindle?	☐		☐		
T	3	Is a Peripheral milling contour milled correctly on main spindle?	☐		☐		
T	4	Is a Peripheral milling contour milled correctly on counterspindle?	☐		☐		
T	5	Is a Y-Axis milling contour milled correctly on main spindle?	☐		☐		
T	6	Is a Y-Axis milling contour milled correctly on counterspindle?	☐		☐		
T	7	Is a B-Axis milling contour milled correctly on main spindle?	☐		☐		
T	8	Is a B-Axis milling contour milled correctly on counterspindle?	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

T	9	Is a position pattern drilled/tapped correctly on main spindle?	☐		☐		
T	10	Is a position pattern drilled/tapped correctly on counterspindle?	☐		☐		
T	11	Works the counterspindle cycle correctly? (only in ShopTurn)	☐		☐		
T	12	Works turning with B axis on the main spindel correctly?	☐		☐		
T	13	Works turning with B axis on the counterspindel correctly?	☐		☐		
T	14	Are the softkeys for the measuring cycles displayed in the editor?	☐		☐		
T	15	Works a measurement program with zero-offset correctly?	☐		☐		
T	16	Works a measurement program with tool-offset correctly?	☐		☐		
T	17	Works a workpiece measurement program correctly?	☐		☐		
T	18	Works the measurement result pictures correctly?	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.3 Parameter

8.3.1 Machine Builder Options

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Test the commissioning of machine builder options	☐		☐		
T+M	2	Test the activation / deactivation of machine builder options	☐		☐		
	3		☐		☐		

8.3.2 Measure tool

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Measure the length of the Tool	☐		☐		
T+M	2	Measure the Diameter / radius of the tool	☐		☐		

8.3.3 Setting data

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Spindle data - minimum spindle speed should be set to a value not equal to 0	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.3.4 Magazine

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Relocate a tool in the Magazine	☐		☐		
T+M	2	Position the Magazine	☐		☐		
T+M	3	Rename a tool	☐		☐		
T+M	4	Disable magazine locations -	☐		☐		
T+M	5	Test oversize tools	☐		☐		
T+M	6	Test tools with fixed locations	☐		☐		
T+M	7	Test interruption scenarios in tool changer ■ Reset the channel ■ Emergency stop ■ Isolator switch	☐		☐		
M	8	Test Hand loaded Tools in program	☐		☐		
T+M	9	L6 Optimised, Spindle Tool = Pre-selected Tool	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.3.5 Tool List

_Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Enter a new tool directly in the spindle	☐		☐		
T+M	2	Enter a new tool directly in the Magazine	☐		☐		
T+M	3	Enter a new tool directly in the Buffer	☐		☐		
T+M	4	Delete a tool directly in Spindle	☐		☐		
T+M	5	Delete a tool from the Magazine	☐		☐		
T+M	6	Delete a tool from the buffer	☐		☐		
T+M	7	Unload the tool in the spindle	☐		☐		
T+M	8	Machine with sub-spindle should be possible to create 2 tools for 1 turret position (different tool types !!!)	☐		☐		
T+M	9	It must be possible to program/call a tool with tool name (e.g. T="name") and also turret position T1	☐		☐		
T+M	10	Relocate a tool in the Spindle	☐		☐		
T+M	11	Tool Change in Inch and Metric	☐				

Copyright © Siemens AG 2012 All Rights Reserved.

8.4 Geometry

8.4.1 Circularity test HMI

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Geometry axis	☐		☐		
T+M	2	Thread boring	☐		☐		

8.4.2 Renishaw circularity test

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Machine axis	☐		☐		
T+M	2	Geometry axis	☐		☐		

8.5 Safety Integrated

8.5.1 Safety related test

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Safety integrated protocol attached.	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.6 Password protection

8.6.1 Securing data areas

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	Key switch → Tools Tool wear length and radius Tool length and radius	☐		☐		
T+M	2	Key switch → Create a tool	☐		☐		
T+M	3	Key switch → Delete a tool	☐		☐		
T+M	4	Programming → Open program	☐		☐		
T+M	5	Programming → Edit program	☐		☐		
T+M	6	Programming → Select program (NC)	☐		☐		
T+M	7	Offsets → fine coarse	☐		☐		
	8		☐		☐		
	9		☐		☐		
	10		☐		☐		
	11		☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

8.7 Various

Technology	Test No	Description	Tested	OEM Alternative / or Comments / not relevant	Tested	Reason for OEM Alternative	To Do
T+M	1	GSP iBase data input in Prototype	☐		☐		
T+M	2	GSP iBase data Upload Test	☐		☐		
T+M	3	Clone backup (TGZ) available for Series machines.	☐		☐		
T+M	4	Country specific safety regulations included in PLC program + door interlocks etc.	☐		☐		
T+M	5	NCK, HMI, PLC and Drive archives tested.	☐		☐		
T+M	6	TGZ file for series production tested	☐		☐		
T+M	7	Adaptive Friction compensation active	☐		☐		
T+M	8	Advanced Surface Machine date set and tested	☐		☐		
T+M	9	Customer Cycles optimised and tested	☐		☐		

Copyright © Siemens AG 2012 All Rights Reserved.

Copyright © Siemens AG 2012 All Rights Reserved.

Copyright © Siemens AG 2012 All Rights Reserved.

Copyright © Siemens AG 2012 All Rights Reserved.

Copyright © Siemens AG 2012 All Rights Reserved.

Copyright © Siemens AG 2012 All Rights Reserved.