

SIEMENS

**DEX Course
Content**

ASO - Advanced Servo Optimization

Course Name : SINUMERIK Advanced Servo Optimization
Course ID : ASO
Duration : 2 Days
Timings : 09:30 to 17:30
Pre-requisite : Minimum 2 years of work experience on CNC commissioning/retrofitting

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Fundamentals & pre-requisites of Automatic Servo Tuning (AST)
- Optimization through different strategies & methods
- Measurement & optimization of speed plant in AST
- Frequency response study through Bode plots
- Amplitude and phase responses in Bode plots
- Pole and zero identification in Bode plots
- Standard and extended current setpoint filters
- Amplitude margin and phase margin study
- Practical exercises

Day 2:

- Overview of Day 1
- Measurement of position controller loop
- Optimization techniques of position controller
- Following error and feed forward control
- Spindle optimization
- Interpolation path optimization
- Circularity test
- Practical exercises

C828 - Commissioning Maintenance

Course Name : SINUMERIK 828D Commissioning & Maintenance
Course ID : C828
Duration : 4 Days
Timings : 09:30 to 17:30
Pre-requisite : Minimum 2-3 years of work experience on CNC electrical-commissioning/servicing
Relevant for : CNC commissioning and maintenance engineers

Day 1:

- System overview
- Line & motor modules (Combi & S120)
- PPU connections & diagnostics
- Toolbox installation
- Time & date Settings
- Machine control panel
- License & option commissioning

Day 2:

- Machine and Setting data
- MCP & PP Module PN configuration
- Commissioning sequence
- 828D PLC instructions
- PLC interface
- Creating PLC program
- PLC alarm & message structure
- User alarms & support files

Day 3:

- Axis & drive diagnostics
- Referencing of encoders
- Maintenance planner
- Easy extension
- Servo optimization (AST)

Day 4:

- Electronic logbook
- Electrical cabinet design
- System restore
- Data management (NC, PLC, Drive, HMI, system software backups)
- Data admin

CMH - CreateMyHmi_ 828D

Course Name : Create My HMI –SINUMERIK 828D
Course ID : CMH
Duration : 2 Days
Timings : 09:30 to 17:30
Pre-requisite : Min 2 years of work experience on CNC electrical maintenance/servicing

Day 1:

- Introduction to SINUMERIK systems
- Menu structure of SIUNUMERIK 828D
- Fundamentals of Run MyScreens
- Basic configuration information
- Structure and elements of adialog
- Defining soft key menus & keys
- Run MyScreen user files
- Example exercises

Day 2:

- Overview of Day 1
- Parameters and variables descriptions
- Read/Write NC/PLC/USER variables through custom screens
- Custom cycle creation
- Image display on custom cycles/screens
- Progress bar display on custom cycles/screens
- Custom operating area configuration & usage
- Example exercises

M100 - Basic milling SINUMERIK 828D

Course Name : Basic Milling – SINUMERIK 828D/840DSL operate
Course ID : M100
Duration : 3 Days
Timings : 09:30 to 17:30
Pre-requisite : Basic knowledge of milling programming
Relevant for : Machine operators, CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Introduction to CNC Milling machines & details about machine parts
- Keyboard layout and screen layout in **new SINUMERIK operate**
- Main menu – machine, services, program, program manager, diagnosis, tool management
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, 10000, MDA, Auto
- Tool offsets & work offsets with **new HMI SINUMERIK operate**
- Introduction to tool management feature with new SINUMERIK operate
- Basic G codes – G0, G1, G2, G3, G04, G90, G91, G95, G96, G33, G75
- Basic M codes – M00, M01, M02, M03, M04, M05, M17, M30 & SPOS
- Concept of federates – F mm/min, F mm/rev. (G94 G95)
- Introduction of part programs & program structure in **Program guide G code**
- ISO dialect mode with SINUMERIK
- Practical training with SINUTRAIN software and machine

M100 - Basic milling SINUMERIK 828D (Contd.)

Day 2:

- Overview of Day 1
- Tool movement and nose radius compensation – G40, G41 & G42
- Introduction to standard milling cycles
 - Face mill
 - Standard pocket & spigot milling
 - Slot mill
 - Thread mill
 - Engrave
- Introduction of standard drilling cycles with position patterns
 - Centering
 - Drilling
 - Deep hole drilling
 - Tapping
 - Reaming
 - Boring
 - Positions patterns – Rows & columns, PCD & Random
- Real-time simulation with advanced optical display clear sight with **2D/3D simulation**
- Practical part cutting on machine

Day 3:

- Overview of Day 2
- Introduction to free contour programming
- Introduction of Advanced technology milling cycles
 - Path mill
 - Profile mill pocket
 - Residual metal removal
- Introduction to sub-routine method of programming
- Use of High speed setting **cycle832** for CAM programming
- Program execution from **USB and Local drive**
- **Mould make view** & Point distribution in SINUMERIK
- Tap retraction while power failure
- Conversion of drawing to NC output – **DXF Reader**
- Unconditional conditional statements
- Detailed session on block search, basic block, over store & program control features
- Practical training with SINUTRAIN software and machine
- Open session with feedback followed by certificate distribution

M101 - Shopmill 828D_840Dsl operate

Course Name : ShopMill – SINUMERIK 828D/840DSL operate

Course ID : M103

Duration : 2 Days

Timings : 09:30 to 17:30

Pre-requisite : Minimum 2-3 years of work experience on Milling machine or completed the M100 course at Siemens DEX

Relevant for : Machine operators, CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on control
- Key benefits of ShopMill compared to G-code programming
- Keyboard layout and screen layout in **new SINUMERIK operate**
- Main menu – machine, services, program, program manager, diagnosis, tool management
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, 10000, MDA, Auto
- Tool offsets & work offsets with **new HMI SINUMERIK operate**
- Introduction to tool management feature with **new SINUMERIK operate**
- Special functions in jog mode
- Introduction to part programs & program structure in ShopMill
- Introduction to standard Milling cycles
 - Face milling
 - Pocket mill
 - Spigot mill
 - Thread mill
 - Slot mill
 - Engraving
- Practical training with SINUTRAIN software and machine

M101 - Shopmill 828D_840Dsl operate (Contd.)

Day 2:

- Overview of Day 1
- Introduction to standard drilling cycles with position patterns
 - Centering
 - Drilling
 - Deep hole drilling
 - Tapping
 - Reaming
 - Boring
 - Positions patterns – Rows & columns, PCD & Random
- Hole masking method with position patterns
- Introduction to free contour programming
- Introduction to Advanced milling cycles
 - Profile pocket milling
 - Profile spigot milling
 - Island milling
 - Residual material removal with pocket/spigot mill
 - Rough drill with profile pocket mill
- Introduction to sub-routine method of programming
- Briefing session on block search, overstore, single blocks (SBL1 & SBL3) and basic block options
- Real-time simulation with advanced optical display clear sight with **2D/3D simulation**
- Practical training with SINUTRAIN software and machine
- Open session with feedback followed by certificate distribution

M102 - Measuring with SINUMERIK

Course Name : Measuring with SINUMERIK

Course ID : M102

Duration : 2 Days

Timings : 09:30 to 17:30

Pre-requisite : Basic knowledge of Milling programming

Relevant for : Machine operators, CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Introduction to online probing
 - Need for probing, probe calibration and its purpose
 - Care to be taken during calibration and measurement
 - Different types of probing systems like Optical, Radio, Hard wired etc.
- Keyboard layout and screen layout in **new SINUMERIK operate**
- Brief session on tool management
 - Different types of tools
 - New tool creation
 - New tool edge creation
 - Loading and unloading tools in the magazine
- Manual measurement in Jog mode – tool & work piece
 - Tool length offset
 - Work piece measurement
 - > Set edge
 - > Align edge
 - > Spacing two edges
 - > Rectangular corner
 - > Rectangular pocket / spigot
 - > Circular Pocket /Spigot Practical demo on machine
- Practical demo on machine
- Question and Answer session

Day 2:

- Overview of Day 1
- Measurement of work piece after machining
 - Single point
 - Two point corner
 - Three point corner
 - Centre of bore
 - Centre of shaft
 - Centre of block
 - Centre of slot
- Practical demo on machine
- Open session with feedback followed by certificate distribution

M104 - High level program milling 828D_840Dsl

Course Name : High level milling program 828D/840DSL
Course ID : M104
Duration : 3 Days
Timings : 09:30 to 17:30
Pre-requisite : Minimum 2-3 years of work experience on CNC milling machine or completed the M100 course at Siemens DEX
Relevant for : Application engineers, CNC programmers and faculties

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Benefits of Advance programming over normal NC programming
- Keyboard layout and Screen layout in **new SINUMERIK operate**
- Basic overview on G codes and M codes
- Unconditional statements with block number & labels
- Logic comparators for conditional statements
- Advance programming concept with **R_variables**
- More exercise on R_variables
- Practical training with SINUTRAIN software and machine

Day 2:

- Overview of Day 1
- High level programming concepts with Local User Data (LUD) in **new SINUMERIK operate**
- Programming using **polar coordinate** system
- Usage of \$ commands in tool data management – Read/write the tool & work offsets
- Real-time simulation with advanced optical display clear sight with **2D/3D simulation**
- Practical training with SINUTRAIN software and machine

Day 3:

- Overview of Day 2
- Detailed session on **frame** concepts
- **Mould views** in new SINUMERIK operate
- Use of High speed setting **cycle832** for CAM programming
- **Finding NC interruption block** in CAM programming during power failures
- Converting drawing to program - **DXF reader**
- External execution of CAM program – **EXTCALL**
- Open session with feedback followed by certificate distribution

M106 - Milling 3+1 Rotary axis - SINUMERIK 828D

Course Name : Milling with Rotary axis 3+1– SINUMERIK 828D/840DSL operate
Course ID : M106
Duration : 3 Days
Timings : 09:30 to 17:30
Pre-requisite : Basic knowledge of Milling programming
Relevant for : Machine operators, CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Basic G codes – G0, G1, G2, G3, G04, G90, G91, G94, G95, G75 and G500
- Basic M codes – M00, M01, M02, M03, M04, M05, M17, M30 & SPOS
- Keyboard layout and screen layout in **new SINUMERIK operate**
- Main menu – machine, services, program, program manager, diagnosis, tool Management
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, MDA, Auto
- Tool offset & work offsets with **new HMI SINUMERIK operate**
- Introduction to part programs & program structure in **Program guide G code**
- Basic programming concepts with **TRACYL**
- Introduction to **standard milling** cycles
 - Standard pocket
 - Slot mill
 - Engrave
- Real-time simulation with advanced optical display clear sight with **2D/3D Simulation**
- Practical training with SINUTRAIN software & machine

M106 - Milling 3+1 Rotary axis - SINUMERIK 828D (Contd.)

Day 2:

- Overview of Day 1
- Introduction to standard **drilling cycles** with position patterns
 - Centering
 - Deep hole drilling
 - Tapping
 - Reaming
 - Positions patterns – Rows & columns, PCD & Random
- Introduction to 4th rotary axis with perfect CAM postprocessor
 - Slot milling with rotary axes
 - PCD drilling on OD
 - Key way milling on OD
- Tangential feed-rate using FGROUP, FL, FGREF
- Introduction to **high speed setting – Cycle832**
- Introduction to **sub-routine method** of programming
- Program execution from **USB and Local drive**
- **Mould make view** & point distribution in SINUMERIK
- Tap retraction during power failure
- Conversion of drawing to NC output – **DXF Reader**
- Detailed session on block search, basic block, overstore & program control features
- Practical training with SINUTRAIN software & machine
- Open session with feedback followed by certificate distribution

STM-828 SINUMERIK 828D Tool Management

Course Name : SINUMERIK 828D Tool Management

Course ID : STM-828

Duration : 2 Days

Timings : 09:30 to 17:30

Pre-requisite : Minimum 2 years of work experience on CNC commissioning/retrofitting

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Tool management fundamentals
- Basic magazine configuration
- User interfaces
- Transfer tables
- Acknowledgement tables
- Load / Unload sequence
- Practical exercises

Day 2:

- Relocate tool / Position magazine sequence
- Machine data related to tool management
- Tool preparation
- Tool change sequence for 1:1 tool
- Tool change sequence for non 1:1 tool
- Tool recovery functions
- Tool change with turret
- Practical exercises
- Question and Answer session

T100 - Basic turning SINUMERIK 828D

Course Name : Basic Turning – SINUMERIK 828D/840DSL operate

Course ID : T100

Duration : 3 Days

Timings : 09:30 to 17:30

Pre-requisite : Basic knowledge of turning programming

Relevant for : Machine operators, CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Introduction to CNC turning machines & details about machine parts
- Keyboard layout and screen layout in new SINUMERIK operate
- Main menu – machine, services, program, program manager, diagnosis, tool management
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, 10000, MDA, Auto
- Tool offset & work offsets with new HMI SINUMERIK operate
- Special functions in jog mode
- Basic G codes – G0, G1, G2, G3, G04, G90, G91, G95, G96, G33, G75
- Basic M codes – M00, M01, M02, M03, M04, M05, M17, M30 & SPOS
- Introduction to part programs & program structure in Program guide G code
- Basic new functions – RND, CHF, CHR, ANG & RNDM
- Practical training with SINUTRAIN software and machine

Day 2:

- Overview of Day 1
- Tool movement and nose radius compensation – G40, G41 & G42
- Introduction to standard Turning cycle
 - Simple turning Cycle951
 - Grooving Cycle930
 - Threading Cycle99
 - Undercut Cycle94
- Introduction of standard DRILLING cycles
 - Centering Cycle81
 - Drilling Cycle82
 - Deep hole drilling Cycle83
 - Tapping Cycle84
- Real-time simulation with advanced optical display clear sight with 2D/3D simulation
- Practical part cutting on machine

Day 3:

- Overview of Day 2
- Introduction to free contour programming
- Introduction to Advanced technology cycles
- Profile turning Cycle952
- Residual metal removal
- Plunge turning
- Introduction to sub-routine method of programming
- Unconditional conditional statements
- Detailed session on block search, basic block, overstore & program control features
- Practical training with SINUTRAIN software and machine
- Open session with feedback followed by certificate distribution

T103 - Turn-mill with Y - SINUMERIK 828D_840Dsl operate

Course Name : TurnMill with Y– SINUMERIK 828D/840DSL operate

Course ID : T103

Duration : 2 Days

Timings : 09:30 to 17:30

Pre-requisite : Basic knowledge of turning programming

Relevant for : Machine operators, CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Basic G codes – G0, G1, G2, G3, G04, G90, G91, G94, G95, G75 and G500
- Basic M codes – M00, M01, M02, M03, M04, M05, M17, M30 & SPOS
- Keyboard layout and screen layout in **new SINUMERIK operate**
- Main menu – machine, services, program, program manager, diagnosis, tool Management
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, MDA, Auto
- Tool offset & work offsets with **new HMI SINUMERIK operate**
- Introduction to part programs & program structure in Program guide G code
- Introduction to co-ordinate system in turn-mill application, different planes, Face Transformation (**TRANSMIT**) and Cylinder transformation (**TRACYL**)
- Basic programming concepts with **TRANSMIT & TRACYL**
- Introduction to standard **MILLING** cycles
 - Rectangular & circular pocket
 - Rectangular & circular spigot
 - Long & circumferential slot
 - Across flat milling
- Practical training with SINUTRAIN software & machine

Day 2:

- Overview of Day 1
- Introduction to standard **Drilling** cycles
 - Centering Cycle81
 - Drilling Cycle82
 - Deep hole drilling Cycle83
 - Tapping cycle84
- Introduction to **free contour** programming
- Introduction to Basic programming concepts with Y axis
 - Key way milling on OD and Face
 - Rectangular and circular pocket milling on OD and face
 - PCD drilling on OD and face
 - Across flat milling
- Introduction to **Advance Milling** cycles
 - Profile mill pocket & spigot
 - Residual metal removal
- Introduction to **sub-routine method** of programming
- Component cutting on the machine
- Detailed session on block search, basic block, overstore & program control features
- Open session with feedback followed by certificate distribution

TTM - Train the trainer Milling

Course Name : Train the trainer Milling

Course ID : TTM

Duration : 3 Days

Pre-requisite : Basic knowledge of milling programming

Relevant for : Training faculties, CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Introduction to CNC Milling machines & Details about machine parts
- Keyboard layout and Screen layout in **new SINUMERIK operate**
- Main menu – machine, services, program, program manager, diagnosis, tool management
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, 10000, MDA, Auto
- Tool offset & work offsets with **new HMI SINUMERIK operate**
- Introduction to tool management feature with new SINUMERIK operate
- Basic G codes – G0, G1, G2, G3, G04, G17, G90, G91, G331, G332, G75
- Basic M codes – M00, M01, M02, M03, M04, M05, M17, M30 & SPOS
- Concept of federates – F mm/min, F mm/rev. (G94, G95)
- Introduction to part programs & program structure in **Program guide G code**
- Practical training with SINUTRAIN software and machine

Day 2:

- Overview of Day 1
- Tool movement and nose radius compensation – G40, G41 & G42
- Introduction to standard Milling cycles
 - Face mill
 - Rectangular & circular pocket
 - Rectangular & circular spigot
 - Long & circumferential slot
- Introduction of standard drilling cycles with position patterns
 - Centering Cycle81
 - Drilling Cycle82
 - Deep hole drilling Cycle83
 - Tapping
 - Position patterns – Row & Columns, PCD & Random
- Real-time simulation with advanced optical display clear sight with **2D/3D simulation**
- Practical part cutting on machine

Day 3:

- Overview of Day 2
- Introduction to free contour programming
- Introduction to Advanced technology milling cycles
 - Path mill
 - Profile pocket
 - Residual metal removal
- Introduction to sub-routine method of programming
- Unconditional conditional statements
- Basic concept of high level language with R-variable (R parameter)
- Practical training with SINUTRAIN software and machine
- Open session with feedback

TTT - Train the trainer turning SINUMERIK 828D

Course Name : Train the trainer Turning – SINUMERIK 828D/840DSL operate

Course ID : TTT

Duration : 5 Days

Pre-requisite : Minimum 2-3 years of work experience as faculty or completed the M100 course at Siemens DEX

Relevant for : Training institute faculties, Application engineers and trainers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Introduction to CNC turning machines & details about machine parts
- Keyboard layout and screen layout in **new SINUMERIK operate**
- Main menu – machine, services, program, program manager, diagnosis, tool management
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, 10000, MDA, Auto
- Tool offsets & work offsets with **new HMI SINUMERIK operate**
- Special functions in jog mode
- Practical training with SINUTRAIN software and machine

Day 2:

- Basic tooling concepts (Different types of tools for various profiles)
- Concept of federates – F mm/min, F mm/rev. (**G94 G95 G96 G97**)
- Basic G codes – G0, G1, G2, G3, G90, G91, G95, G33, G04, G75
- Basic M codes – M00, M01, M02, M03, M04, M05, M17, M30 & SPOS
- Introduction to part programs & program structure in **Program guide G code**
- Tool movement and nose radius compensation – G40, G41 & G42
- Basic new functions – RND, CHF, CHR, ANG & RNDM
- Practical training with SINUTRAIN software and machine

Day 3:

- Overview of Day 1 and 2
- Introduction to standard Turning cycles
 - Simple Turning Cycle951
 - Grooving Cycle930
 - Threading Cycle99
 - Undercut Cycle94
- Introduction of standard Drilling cycles
 - Centering Cycle81
 - Drilling Cycle82
 - Deep hole drilling Cycle83
 - Tapping Cycle84
- Real-time simulation with advanced optical display clear sight with **2D/3D simulation**
- Practical training with SINUTRAIN software and machine

Day 4:

- Overview of Day 3
- Introduction to free contour programming
- Introduction of Advanced technology cycles
 - Profile Turning Cycle952
 - Residual metal removal
 - Profile Plunge turning
 - Profile Groove turning
- Complex machining concept with above cycles
- Introduction to sub-routine method of programming
- Practical part cutting on machine

Day 5:

- Overview of Day 4
- Unconditional conditional statement & logic comparators
- High level language with R-variable (R parameter)
- Introduction to Systems variables (\$ variables)
- Explanation of FRAME commands
- Tap retraction during power failure
- Conversion of drawing to NC output – **DXF Reader**
- Detailed session on block search, basic block, overstore & program control features
- Practical training with SINUTRAIN software and machine
- Open session with feedback followed by certificate distribution

For online registrations visit www.siemens.co.in/DEX

Siemens Ltd.
Digital Experience & Application Center
Shed No. 1 & 2, SY No. 26 & 27
Plot No. 1 A, Systems Campus
Peenya Industrial Area, Phase 2
Chokkasandara Village, Yeshwantpur Hobli
Bengaluru - 560058
Karnataka, India
Tel.: 080-4097 5628
E-mail: tac.training.in@siemens.com

Article number: DI-MC-MTS-DEX-FY23

For more information call us on 1800 209 1800

For Life Cycle Support of Products, Systems and Solutions
call us on 1800 209 0987

www.siemens.co.in/DEX

Product upgradation is a continuous process. Hence, data in this brochure is subject to change without prior notice. For the latest information, please get in touch with our Sales Offices.