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

KCM - Kinematic Chain and Measurement – SINUMERIK 828D/ SINUMERIK ONE

Course Name : Kinematic Chain and Measurement – SINUMERIK 828D/ SINUMERIK ONE

Course ID : KCM

Duration : 2 Days

Timings : 09:30 to 17:30

Pre-requisite : Minimum 3 to 5 years of work experience on Milling programming and operation

Target group : Application and commissioning engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Fundamentals of kinematic chain
- Introduction to kinematic chains
  - T Type
  - P Type
  - M Type
- Requirements for kinematic chain and collision avoidance
  - License details
  - Machine data setting for ECO collision avoidance
- Introduction to elements of kinematic chain
- Introduction to ROOT structure
- Geometric correction of the kinematic chain
- Creating chain with the HMI
- Kinematic chain for 5-axis table kinematics
  - Identifying kinematics of the machine
  - Creating root, linear axis and rotary axis elements
  - Testing the chain

Day 2:

- Introduction to Transformations with kinematic chains.
- Enable the Transformation
  - Understand the table and its meaning for mechanical dimension calculation
  - Setup the table as per AC kinematic
- Probe calibration using SINUMERIK measuring cycles
- Understand the cycle9960 function
  - Kinematic measurement with cycle9660
- Validate the Cycle800 function in Jog mode
- Practical training on Run My Virtual Machine (RMVM)and machine
- Open session with feedback followed by certificate distribution

M100 - Milling with SINUMERIK 828D/ SINUMERIK ONE

Course Name : Milling with SINUMERIK 828D/ SINUMERIK ONE

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 - Milling with SINUMERIK 828D/ SINUMERIK ONE (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, overstore & program control features
- Practical training with SINUTRAIN software and machine
- Open session with feedback followed by certificate distribution

M101 - 3+2 Milling with SINUMERIK 828D

Course Name : 3+2 Milling with SINUMERIK 828D

Course ID : M101

Duration : 2 Days

Timings : 09:30 to 17:30

Pre-requisite : Minimum 3 to 5 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 controls
- Overview of different types of 3+2 & 5-axis machine kinematics
- Introduction to SINUMERIK features required to handle 3+2 & 5-axis applications
- Overview of variations in CYCLE800
- Right hand thumb rule for 2 ½D Milling
- Procedure for swiveling in JOG mode
- Programming with cycle800 with Direct axis & Axis-by-Axis programming
- Practical demonstration of manual programming, drilling cycles, and milling cycles
- Tap retraction method during power failure
- Real-time simulation with advanced optical display clear sight with 2D/3D simulation
- Practical training on SINUTRAIN software and machine

Day 2:

- Overview of Day 1
- Introduction to TRAORI
  - ORIAXES
  - ORIVECT
  - ORIMKS
  - ORIWKS
- Programming with Direct programming
- Programming with Vector programming
- Programming with the Euler angle
- 3D Tool Radius Compensation with
  - CUT3DC
  - CUT3DCD
  - CUT3DFD
- Fgroup to maintain synchronized path feed
- Use of High Speed Setting cycle832 with multi-orientation using CAM
- Program execution from USB and Local drive
- Overview of mould make view & point distribution in SINUMERIK
- Briefing session on block search, basic block, overstore & program control features
- Practical training on SINUTRAIN software and machine
- Open session with feedback followed by certificate distribution

M102 - Measuring with Sinumerik 828D/ 840Dsl/ SINUMERIK ONE

Course Name : Measuring with Sinumerik 828D/ 840Dsl/ SINUMERIK ONE

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

M103 - 3+2 & 5-axis Milling with SINUMERIK ONE

Course Name : 3+2 & 5-axis Milling with SINUMERIK ONE

Course ID : M103

Duration : 2 Days

Timings : 09:30 to 17:30

Pre-requisite : Minimum 3 to 5 years of work experience on Milling machines or completed the M100 course at Siemens DEX

Target group : CNC programmers and application engineers

Day 1:

- Introduction to SINUMERIK systems – A brief history on controls
- Different types of 3+2 and 5-axis machine kinematics
- SINUMERIK features required to handle 3+2 & 5 axis applications.
- Function of cycle800 and its explanation in details
- Right hand thumb rule for 2½D Milling
- Programming with cycle800 with Direct axis programming
  - Example 1 with Manual Programming
  - Example 2 with Drilling cycles
    - > Centering
    - > Drilling
    - > Deep hole drilling
    - > Tapping
    - > Reaming
    - > Positions patterns – Rows & columns, PCD & Random
  - Example 3 with Milling cycles
    - > Face mill
    - > Standard pocket & spigot milling
    - > Slot mill
- Real-time simulation with advanced optical display clear sight with **2D/3D simulation**
- Practical training on SINUTRAIN software and machine

Day 2:

- Overview of Day 1
- Programming with cycle800 with axis-by-axis programming
  - Example 1 with Manual Programming
  - Example 2 with Drilling + Milling cycles
- 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
- 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

M104 - High level milling program 828D/ 840Dsl/ SINUMERIK ONE

Course Name : High level milling program 828D/ 840Dsl/ SINUMERIK ONE

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 Advanced 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

P828 - Preventive Maintenance 828D

Course Name : Preventive Maintenance 828D

Course ID : P828

Duration : 1 Day

Timings : 09:30 to 17:30

Pre-requisite : Minimum 1 year of work experience on CNC electrical maintenance/ servicing, basic knowledge of Electrical & Electronics stream

Relevant for : CNC service and maintenance engineers

Day 1:

- PPU and interfacing ports, pin details
- Hardware functions connected in 828D systems (Line filters, line reactors, line & motor modules, combi module, sensor modules, NX modules, PP72/48, MCP, PN/PN couplers)
- Architecture line diagram of SINUMERIK 828D
- Hardware part numbers identification
- 828D screen navigation – setup/diagnosis
- Access levels, Time & Date Settings
- Machine & setting data
- Creating & restoring archives
- Axis & drive diagnostics
- Topology diagnostics
- PLC & NC variable diagnostics
- PLC ladder logic viewing and editing (Status list, symbol table, cross references,window 1 and 2)
- Alarm structure
- PLC alarm configuration
- Absolute encoder referencing
- License & options management
- Generic do's and don'ts

S#1 - Commissioning & Maintenance training on SINUMERIK ONE

Course Name : Commissioning & Maintenance training on SINUMERIK ONE

Course ID : S#1

Duration : 5 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:

- SINUMERIK ONE Hardware
- NCU + PPU
- Modular MCP
- Create MyVirtual Machine (CMVM) interface and usage
- NC machine data (for eg. - course machine)

Day 2:

- PLC Base program
- PLC NC axis to drive connection
- PLC base example for milling machine
- TIA portal with programming recommendations for IEC1131-3

Day 3:

- Basics of tool management for chain magazine
- Basics of technology configuration including Kinematic chain
- Commission Virtual to Real:
  - NC Data form Virtual to Real
  - PLC Program from TIA portal to Real
  - PLC Program from TIA as dsf archive to Real
  - Drive new configuration and optimize
  - HMI from Virtual to Real

Day 4:

- Basic SW installation of IPC (Windows)
- Basic installation of ONE NCU:
  - SSD Drive
  - SD Card
- System management
- Backup Local
- Backup - IPC
- Restore Local
- Restore - IPC
- Update
- Settings and Architecture

Day 5:

- Backup and restore complete system to:
  - USB
  - IPC
- Overview of course from Day 1 to Day 4
- Open session with feedback followed by certificate distribution

T100 - Turning – SINUMERIK 828D/ 840Dsl operate

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

T100 - Turning – SINUMERIK 828D/ 840Dsl operate (Contd.)

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 - TurnMill 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

T103 - TurnMill with Y– SINUMERIK 828D/ 840Dsl operate (Contd.)

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 **Advanced 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

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

S19588