### ASO - Advanced Servo Optimization

**Course Name**: SINUMERIK Advanced Servo Optimization  
**Course ID**: ASO  
**Duration**: 2 Days  
**Timings**: 09.30 to 17.30  
**Requirements**: Min 2 years working experience on CNC commissioning/retrofitting.

**Day 1:**
- Introduction to SINUMERIK systems – A brief history on controls.  
- Fundamentals & Prerequisites of Automatic Servo Tuning  
- 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 / Summary of the previous day  
- 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**: C828 - SINUMERIK 828D Commissioning & Maintenance  
**Course ID**: C828  
**Duration**: 5 Days  
**Timings**: 09.30 to 17.30  
**Requirements**: Min 2-3 years working experience on CNC electrical - commissioning/servicing.

**Target group**: CNC commissioning and maintenance engineers.

**Day 1:**
- System Overview  
- Line & Motor Modules (Combi & S120)  
- PPU Connections & Diagnostics  
- Tool Box Installation  
- Time & Date Settings  
- Machine Control Panel

**Day 2:**
- Remote diagnosis (AMM)  
- License & Option Commissioning  
- Machine and Setting Data  
- MCP & PP Module PN Configuration  
- Commissioning sequence

**Day 3:**
- 828D Plc Instructions  
- PLC Interface  
- Creating PLC Program  
- PLC Alarm & Message structure  
- User Alarms & Help files

**Day 4:**
- Axis & Drive diagnostics  
- Referencing of Encoders  
- Maintenance Planner  
- Easy Extend  
- Servo Optimization (AST)

**Day 5:**
- Electronic Log book  
- Electrical Cabinet Design  
- System Restore  
- Data Management (NC, PLC, DRIVE, HMI, System Software backups)  
- Data Admin
<table>
<thead>
<tr>
<th><strong>C840D - Commissioning Maintenance</strong></th>
<th><strong>CMH - CreateMyHmi_ 828D</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Course Name</strong> : C840 - SINUMERIK 840DSL Commissioning &amp; Maintenance</td>
<td><strong>Course Name</strong> : Create My HMI - 828D</td>
</tr>
<tr>
<td><strong>Course ID</strong> : C840</td>
<td><strong>Course ID</strong> : CMH</td>
</tr>
<tr>
<td><strong>Duration</strong> : 5 Days</td>
<td><strong>Duration</strong> : 2 Days</td>
</tr>
<tr>
<td><strong>Timings</strong> : 09.30 to 17.30</td>
<td><strong>Timings</strong> : 09.30 to 17.30</td>
</tr>
<tr>
<td><strong>Requirements</strong> : Min 2-3 years working experience on CNC electrical - commissioning/servicing.</td>
<td><strong>Requirements</strong> : Min 2 years working experience on CNC electrical maintenance/servicing.</td>
</tr>
<tr>
<td><strong>Target group</strong> : CNC commissioning and maintenance engineers.</td>
<td></td>
</tr>
</tbody>
</table>

**Day 1:**
- System Overview
- Service & Commissioning Tools
- Menu Structure
- License & Option Management
- PCU, TCU and OP Diagnostics and Interfaces

**Day 2:**
- Machine and setting Data
- NCU Diagnostics Displays and Switches
- NCU Digital Inputs & Outputs
- Creating a PLC Program
- Basic PLC Function
- Machine Control Panel and IO Module Configuration

**Day 3:**
- Commissioning Sequence.
- Encoder Adjustment & Referencing.
- Alarm Structure
- PLC Alarms & Messages - Basic
- PLC User Alarms & Help files

**Day 4:**
- Axis & Drive Diagnostics
- Servo Optimization (AST)
- NX Module Configuration
- Adding / Removing Module

**Day 5:**
- Data Management (NC, PLC, DRIVE, HMI, System Software backups)
- Ghost Backup on PCUS0.x
- Fault finding exercises
- Open session
### KC2A - Kinematic chain and Collision Avoidance

**Application and Commissioning engineers**

**Course Name**: Kinematic chain and Collision avoidance on SINUMERIK 840Dsl

**Course ID**: KC2A

**Duration**: 3 Days

**Timings**: 09.30 to 17.30

**Requirements**:

**Target group**: Application and Commissioning engineers

**Day 1**:

- Introduction to SINUMERIK systems – A brief on latest 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 3-axis machine with mountable A axis.
  - Identifying the kinematic of the machine.
  - Creating root, linear axis and rotary axis elements.
  - Testing the chain.
- Creating Collision avoidance with the HMI for 3 & 3+1 axes.
  - Protection areas
  - Protection area elements.
  - Collision pairs.
- Activation and deactivation of collision avoidance.
- Kinematic chain for 5-axis table kinematics.
  - Identifying the kinematic of the machine.
  - Creating root, linear axis and rotary axis elements.
  - Testing the chain.
- Swivel configuration.
- Introduction to Transformations with kinematic chains.
- Practical training on sinutrain software and machine.
- Open session & Feedback followed by certificate distribution.

### KRT - Basic Robotics training

**Course Name**: Basic Robotics training

**Course ID**: KRT

**Duration**: 2 Days

**Timings**: 09.30 to 17.30

**Requirements**:

**Target group**: Training co-ordinates.

**Day 1**:

- Introduction to robotics
- Introduction to parts of robots
- Introduction to teach pendent
- Axis and their movements
- Robot handling in manual and auto modes
- Coordinate systems in conjunction with robots
- Alarms and messages
- Mastering principle
- Practical training on KUKA robot

**Day 2**:

- Overview / summary of the previous day
- Introduction to robotic programs
- Creating and modifying programmed motions
- Practical training on KUKA robot
- Demo-robot application with SINUMERIK CNC
- Open session and feedback.
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
Requirements: Basic knowledge of milling programming
Target group: 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 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, 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 on sinutrain software and machine.

Day 2:
• Overview / Summary of the previous day.
• 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 Superb optical display clear sight with 2D/3D simulation.
• Practical part cutting on machine.

Day 3:
• Overview / Summary of the previous day.
• 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 cycle 832 for CAM programming.
• Program execution from USB and Local drive
• Mould make view & Point distribution in SINUMERIK.
• Tap retraction while power failure
• Conversion drawing to NC output – DXF Reader
• Unconditional conditional statements.
• Details session Block search, Basic block, overstore & Program control features.
• Practical training on sinutrain software and machine.
• Open session & 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
Requirements : Basic knowledge of milling programming
Target group : 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 it purpose.
  - Care to be taken during calibration and Measurement.
  - Different type of Probing Systems available like, Optical, Radio, Hardwired 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.
  - Load and unloading tools to 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

Day 2:
- Overview / Summary of the previous day.
- 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 & Feedback followed by certificate distribution.

M103 - Mould & Die with SINUMERIK

Course Name : Mold and Die SINUMERIK 828D/840Dsl
Course ID : M103
Duration : 3 Days
Timings : 09.30 to 17.30
Requirements : Basic knowledge of design & manufacturing
Target group : Application and manufacturing engineers

Day 1:
- Introduction to SINUMERIK systems – A brief history on controls.
- Introduction basic modeling.
  - Sketching (line, circle, fillet, chamfer, offset curve, pattern curve and mirror curve.)
  - Modeling (extrude, Revolve, Unite, Subtract, fillet and chamfer)
- Practical training on software.

Day 2:
- Overview / Summary of the previous day.
- Introduction basic manufacturing.
  - Analyzing model
  - Work piece setting (MCS, Blank and part defining)
  - 2.5 Axis Milling- Prismatic Shapes, Flat face milling, Drilling, Hole Milling, Thread Milling
  - 3 Axis Milling- Mill Contour; Roughing, Rest Milling, Z-Level, Roughing Strategies
  - 3 Axis Advanced-Streamline, Area Milling, 3D Engraving, Finishing Strategies
  - Point distribution in CAM software.
  - Tool path verification and simulation
  - Practical training on software.

Day 3:
- Overview / Summary of the previous day.
- Latest features of SINUMERIK V4.7 for Die mould application
- Technology package for High speed milling – Mdynamics + Top surface
- Control features
  - High speed setting (cycle832)
  - Mold make view & Point distribution in SINUMERIK.
  - Details session extended Block search, Basic block, overstore & Program control features.
  - Tap retraction while power fail.
  - Real-time simulation with Superb optical display clear sight with 2D/3D Simulation.
  - Program execution from USB and Local drive
  - Mold make view & Point distribution in SINUMERIK.
  - Tap retraction while power failure
  - Conversion of drawing to NC output – DXF Reader
- Unconditional and conditional statements.
- Mold manufacturing using CAM software and SINUMERIK controller
- Practical cutting on machine.
- Open session & Feedback
M104 - High level program milling
828D_840DSL - 3D

Course Name : High level milling program 828D/840DSL.
Course ID : M104
Duration : 03 Days
Timings : 09.30 to 17.30
Requirements : Min 2-3 years working experience on cnc milling machine or trained on M100 in Siemens TAC.
Target group : 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 on sinutrain software and machine.

Day 2:
• Overview / Summary of the previous day.
• High level programming concepts with Local user data in NEW SINUMERIK operate.
• Programming using Polar coordinate system.
• Usage of $ commands in tool data management – Read/ writing the tool & work offsets.
• Real-time simulation with Superb optical display clear sight with 2D/3D simulation.
• Practical training on sinutrain software and machine.

Day 3:
• Overview / Summary of the previous day.
• Details session on Frame concepts.
• Mould views in NEW SINUMERIK operate.
• Use of High speed setting cycle832 for CAM programming.
• Tap retraction while power failure
• Finding NC interruption block in CAM programming when power failures.
• Converting drawing to program - DXF reader
• External execution of CAM program – EXTCALL
• Open session & Feedback followed by certificate distribution.

M105 - Postprocessor development with SINUMERIK

Course Name : Postprocessor Development with SINUMERIK
Course ID : M105
Duration : 2 Days
Timings : 09.00 to 17.00
Target group : Application engineers and post build engineers

Day 1:
• TAC Agenda and orientation of TAC
• Introduction to SINUMERIK systems – A brief history on controls.
• Installation and setup of Sinutrain in Laptops
• Automation license manager - usage
• Controller main menu – Screen area, machine, program, program manager
• Keyboard layout and Screen layout in New SINUMERIK operate
  - Setup operating area
  - Diagnostic
  - Tool list management
• Zero point or Settable work offsets – Read and writing methodology
• Blank information or defining the raw material stock in program
• Brief session on Build-group function - to Reduce the program blocks
• Testing of SINUMERIK function in participants post developing tools.
• Practical training on SINUTRAIN software.

Day 2:
• Overview / Summary of the previous day.
• Easy-start – Install, usage and benefits
• High level programming concepts using R variables, LUD and GUD
• Swivel cycle - CYCLE800 For transformation of Coordinate system in 5 axes positioning milling
• High speed setting - CYCLE832 for Mould & Die programming to achieve best surface finish with SINUMERIK.
• Transformation for 5 axes simultaneous machining with - TRAORI
• Testing of SINUMERIK function in participants post developing tools.
• Practical training on sinutrain software.
• Open session - Q&A
• Feedback
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
Requirements: Basic knowledge of milling programming
Target group: 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 of 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 Superb optical display clear sight with 2D/3D Simulation
  • Practical training on sinutrain software & machine.

Day 2:
• Overview / Summary of the previous day.
• Introduction of standard drilling cycles with position patterns.
  – Centering
  – Deep hole drilling
  – Tapping
  – Reaming
  – Positions patterns – Rows & columns, PCD & Random
• Introduction 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 – cycle 832
• Introduction to sub-routine method of 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
• Details session Block search, Basic block, overstore & Program control features.
• Practical training on sinutrain software & machine.
• Open session & 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
Requirements: Min 2 years working 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 Answers.

STM - 840 SINUMERIK 840Dsl Tool Management

Course Name: SINUMERIK 840Dsl Tool Management
Course ID: STM-840
Duration: 2 Days
Timings: 09.30 to 17.30
Requirements: Min 2 years working experience on CNC commissioning/retrofitting.

Day 1:
- Introduction to SINUMERIK systems – A brief history on controls.
- Tool Management Fundamentals
- User Interfaces PLC-NCK
- Tool management Interface data blocks
- Machine data for Tool Management
- Basic Magazine configuration
- Tool Load and Unload sequence
- Tool Relocate and Position Magazine sequence
- Practical Exercises

Day 2:
- Overview / Summary of the previous day
- Tool Change preparation
- Tool Change Execution
- Hand tool sequence
- Tool change interruption
- Tool Recovery functions
- Turret tool change
- Practical Exercises
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
Requirements: Basic knowledge of turning programming
Target group: 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 of part programs & program structure in Program guide G code.
• Basic new functions – RND, CHF, CHR, ANG & RNDM.
• Practical training on sinutrain software and machine.

Day 2:
• Overview / Summary of the previous day.
• Tool movement and nose radius compensation – G40, G41 & G42
• Introduction to standard TURNING cycles.
  – Simple turning cycle951
  – Grooving cycle930
  – Threading cycle99
  – Undercut cycle94
• Introduction of standard DRILLING cycles.
  – Centering cycle 81
  – Drilling cycle 82
  – Deep hole drilling cycle83
  – Tapping cycle84
• Real-time simulation with Superb optical display clear sight with 2D/3D simulation.
• Practical part cutting on machine.

Day 3:
• Overview / Summary of the previous day.
• Introduction to free contour programming.
• Introduction of Advanced technology cycles
• Profile turning cycle 952
• Residual metal removal
• Plunge turning
• Introduction to sub-routine method of programming
• Unconditional conditional statements.
• Tap retraction while power failure
• Conversion of drawing to NC output – DXF Reader
• Details session Block search, Basic block, overstore & Program control features.
• Practical training on sinutrain software and machine.
• Open session & Feedback followed by certificate distribution
Course Name : Basic turning – SINUMERIK 828D/840DSL operate.
Course ID : T101
Duration : 3 Days
Timings : 09.30 to 17.30
Requirements : Basic knowledge of turning programming
Target group : 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 & workoffsets with New HMI SINUMERIK operate.
• Tool offset methods for Gang type lathes
• 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 of part programs & program structure in Program guide G code.
• Basic new functions – RND, CHF, CHR, ANG & RNDM.
• Practical training on sinutrain software and machine.

Day 2:
• Overview / Summary of the previous day.
• Tool movement and nose radius compensation – G40, G41 & G42
• Introduction to standard TURNING cycles.
  – Simple turning cycle951
  – Grooving cycle930
  – Threading cycle99
  – Undercut cycle94
• Introduction of standard DRILLING cycles.
  – Centering cycle 81
  – Drilling cycle 82
  – Deep hole drilling cycle83
  – Tapping cycle84
• Programming concepts for Gangtype tools application (X is negative direction)
• Real-time simulation with Superb optical display clear sight with 2D/3D simulation.
• Practical part cutting on machine.

Day 3:
• Overview / Summary of the previous day.
• Introduction to free contour programming.
• Introduction of Advanced technology cycles
• Profile turning cycle 952
• Residual metal removal
• Plunge turning
• Introduction to sub-routine method of programming
• Unconditional conditional statements.
• Tap retraction while power failure
• Conversion of drawing to NC output – DXF Reader
• Details session Block search, Basic block, overstore & Program control features.
• Practical training on sinutrain software and machine.
• Open session & Feedback followed by certificate distribution
T103 - Turn-mill with Y - SINUMERIK 828D_840Dsl operate

Course Name: Turn-mill with Y - SINUMERIK 828D/840DSL operate.

Course ID: T103

Duration: 2 Days

Timings: 09.30 to 17.30

Requirements: Basic knowledge of turning programming

Target group: 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 of 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 on sinutrain software & machine.

Day 2:
- Overview / Summary of the previous day.
- Introduction of standard DRILLING cycles
  - Centering cycle 81
  - Drilling cycle 82
  - 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
- Details session Block search, Basic block, overstore & Program control features.
- Open session & Feedback followed by certificate distribution
Day 2:
- Overview / Summary of the previous day.
- Introduction to basic programming concepts.
- Simple examples writing manual program.
- Introduction to standard turning cycles:
  - stock removal
  - Grooving
  - Threading
  - Undercut
  - Cutoff.
- Details session on Block search, interrupt point and Program control features.
- Real-time and offline with 2D simulation.
- Practical with component cutting on the machine.

Day 3:
- Overview / Summary of the previous day.
- Introduction to free contour programming (attach contour method)
- Introduction to standard drilling cycles:
  - Centering
  - Drilling
  - Deep hole drilling
  - Tapping
- Introduction to sub-routine method of programming.
- Practical training on sinutrain software and machine.
- Brief session on MM+ programming.
- Open session & Feedback followed by certificate distribution.
Course Name: Train the trainer milling SINUMERIK 828D/840Dsl Operate

Course ID: TTM

Duration: 5 Days

Requirements: Min 2-3 years working experience as faculty or Trained on M100 in Siemens TAC.

Target group: 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.
- Practical training on sinutrain software and machine.

Day 2:
- Overview / Summary of the previous day.
- Basic G codes – G0, G1, G2, G3, G04, 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 of 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
- Details session Block search, Basic block, overstore & Program control features.
- Practical training on sinutrain software and machine.

Day 3:
- Overview / Summary of the previous day.
- 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
- Introduction to sub-routine method of programming.
- Real-time simulation with Superb optical display clear sight with 2D/3D simulation.
- Practical training on sinutrain software and machine.

Day 4:
- Overview / Summary of the previous day.
- Introduction to free contour programming.
- Introduction of Advanced technology milling cycles
  - Path mill
  - Profile mill pocket & Spigot
  - Residual metal removal
- Mould view with New HMI SINUMERIK operate.
- High speed setting cycle 832
- Practical training on sinutrain software and machine.

Day 5:
- Overview / Summary of the previous day.
- Unconditional conditional statements.
- Basic concept to high level language with R-variable (Macro).
- Explanation of FRAME commands.
- Mould views in new SINUMERIK operate.
- Tap retraction while power failure
- Finding NC interruption block in CAM programming when power failures.
- Converting drawing to program – DXF reader
- External execution of CAM program – EXTCALL
- Introduction to Systems variables ($ Variables).
- Open session & feedback followed by certificate distribution.
**Course Name**: Train the trainer Turning – SINUMERIK 828D/840DSL operate.

**Course ID**: TTT

**Duration**: 5 Days

**Requirements**: Min 2-3 years working experience as faculty or Trained on M100 in Siemens TAC.

**Target group**: 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 offset & work offsets with New HMI SINUMERIK operate.
- Special functions in jog mode.
- Practical training on sinutrain software and machine.

**Day 2**:
- Basic tooling concepts. (Different type 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 of 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 on sinutrain software and machine.

**Day 3**:
- Overview / Summary of the previous day.
- Introduction to standard TURNING cycles.
  - Simple turning cycle951
  - Grooving cycle930
  - Threading cycle99
  - Undercut cycle94
- Introduction of standard DRILLING cycles.
  - Centering cycle 81
  - Drilling cycle 82
  - Deep hole drilling cycle83
  - Tapping cycle84
- Real-time simulation with Superb optical display clear sight with 2D/3D simulation.
- Practical training on sinutrain software and machine.

**Day 4**:
- Overview / Summary of the previous day.
- Introduction to free contour programming.
- Introduction of Advanced technology cycles
  - Profile turning cycle 952
  - 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 / Summary of the previous day.
- Unconditional conditional statement & Logic comparators.
- High level language with R-variable (R parameter).
- Introduction to Systems variables ($ Variables).
- Explanation of FRAME commands.
- Tap retraction while power failure
- Conversion of drawing to NC output – DXF Reader
- Details session Block search, Basic block, overstore & Program control features.
- Practical training on sinutrain software and machine.
- Open session & 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

www.siemens.co.in/DEX

For more information call us on 1800 209 1800

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

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.