

SIEMENS

Ingenuity for life

DEX Course Content

ASO - Advanced Servo Optimization

CourseName : 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

C840D - Commissioning Maintenance

Course Name : C840 - SINUMERIK 840DSL Commissioning & Maintenance

Course ID : C840

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
- 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 PCU50.x
- Fault finding exercises
- Open session

CMH - CreateMyHmi_ 828D

Course Name : Create My HMI - 828D

Course ID : CMH

Duration : 2 Days

Timings : 09.30 to 17.30

Requirements : Min 2 years working experience on CNC electrical maintenance/servicing.

Day 1:

- Introduction to SINUMERIK systems
- Menu structure of 828D
- Fundamentals of Run My Screens
- Basic configuration information
- Structure and elements of a dialog
- Defining soft key menus & keys
- Run My Screen User files
- Example exercises

Day 2:

- Overview / Summary of the previous day
- Parameters and Variables descriptions
- Read/Write NC/PLC/USER variables through custom screens.
- Custom cycle creation
- Image display at custom cycles/screens.
- Progress bar display at custom cycles/screens.
- Custom operating area configuration & usage.
- Example exercises

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 : Basic knowledge of Robotics.

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 **cycle832** 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 its 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/840DsI
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 – Mynamics + 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**
 - **Mould 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 developemnt 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

CourseName : 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 non1:1 tool
- Tool recovery functions
- Tool Change with turret
- Practical Exercises.
- Question and Answers.

STM - 840-SINUMERIK 840Dsl Tool Management

CourseName : 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

T101 - Basic turning SINUMERIK 828D - Gany type

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

T105 - Basic Turning PPU161

Course Name : Basic turning on– SINUMERIK PPU161/808D advance.

Course ID : T105

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.
- Basic G codes – G0, G1, G2, G3, G04, G18, G90, G91, G95, G96, G33, and 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.
- Introduction to the keyboard layout and Screen layout in NEW SINUMERIK operate.
- Main menu – machine, program, program manager, diagnosis, tool offset parameter.
- Main modes – Jog, Ref, Inc1, 10, 100, 1000, 10000, MDA, Auto
- Tool offset & work offsets with SINUMERIK.
- Practical training on machine and Sinutrain.

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.

TTM - Train the trainer Milling

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.

TTT - Train the trainer turning SINUMERIK 828D

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.

