

SIEMENS

Ingenuity for life

Online Webinars at DEX

www.siemens.co.in/DEX

We are running webinars on a daily basis on a range of topics to help you learn all about our flexible and innovative CNC solutions with SINUMERIK for the shopfloor. These webinars will inform you in a single glance about the current trends, topics, and tips regarding CNC manufacturing. These are One hour sessions including Q&A. Register on www.siemens.co.in/DEX to attend our webinars.

Webinar on ShopMill

Overview

When programming the SINUMERIK control utilizing the ShopMill conversational system, it's very helpful to take advantage of the vast selection of standard cycles in Milling

- Work-step programming without DIN-ISO-knowledge
- Benefits of ShopMill vs G-code programming
- ShopMill programming structure for Milling technology
- Milling Cycles – Facing, Pocket, Spigot, Slot, Thread milling and engraving
- Drilling Cycles – Centering, Drilling, Deep hole, Boring and Tapping
- Positions patterns – Rows, Columns, PCD, Random, Obstacle, Position repetition
- A Brief session on Block search, overstore, single blocks (SBL1 & SBL3) and basic block options

Webinar on ShopTurn

Overview

Intended to show a machine tool user how to begin applying strategies that will make their part programs even more efficient. This is ShopTurn-based programming utilizing advanced functions like gouge detection, residual material, and irregular stock machining to name a few. Next, we will begin to explore feed optimization, as well as different cutting strategies. This course will take your ShopTurn programming to the next level.

- Benefits of ShopTurn vs G-code programming
- Introduction of part programs & program structure in shopturn
- Simple turning cycle programming using shopturn
 - Simple straight/taper turning
 - Grooving
 - Threading
 - Undercut
- Introduction to free contour programming in shopturn
- Tool creation with full geometry description
- Introduction of standard DRILLING cycles in Shopturn
 - Centering
 - Drilling
 - Deep hole drilling
 - Tapping
- Advance Contour turning cycles:
 - Profile turning

Webinar on ShopTurn - TurnMill

Overview

As mixed technology becomes more mainstream in today's manufacturing, it is critical to understand all of the capabilities your SINUMERIK CNC has to offer. When looking in from the outside, these complex turning centers can appear intimidating. However, once you see how simple it is in ShopTurn to utilize these advanced capabilities, you will be applying them on a daily basis.

- Face-drilling operations with either C- or Y-axis
- Peripheral drilling on-center and off-center
- Applying the milling cycles for face and peripheral machining
- Key challenges when setting-up tools and offsets

Webinar on ShopMill 3+2

Overview

Programming with Cycle800 is designed to show a machine tool user on how to leverage 5-axis machining technologies in the demanding markets of the job shop. We will first look at the key functions within the Siemens control that aid in dramatically simplifying the tasks of setting up a 5-axis machine. We will explore how to handle 5-axis (3+2) part programming directly on the control.

- Different type of 3+2 and 5 axis machine kinematics
- SINUMERIK features required to handle 3+2 & 5 axis applications
- SINUMERIK 3+2 machining using Cycle800
- Milling Cycles – Facing, Pocket, Spigot, Slot
- Drilling Cycles – Centering, Drilling, Deep hole, Boring and Tapping
- Positions patterns – Rows, Columns, PCD, Random & Position repetition

Webinar on Milling 828D - Basic

Overview

When programming the SINUMERIK CNC in G-code, it's very helpful to take advantage of the vast selection of standard cycles. This is even more true when using a CAM system and trying to take advantage of the full power of the control.

- SINUMERIK programming structure in ProgramGuide
- Basic G codes and M codes
- SINUMERIK Functions comparison with ISO
- Tool offset and work offset terminology
- Milling cycles – facing, pocket, spigot, slot, thread milling, engraving
- Drilling cycles – centering, drilling, deep hole, boring, tapping
- Positions – random, pattern, bolt hole, obstacle, position repetition
- Brief session on Block search, overstore, single blocks (SBL1 & SBL3) and basic block options

Webinar on Milling 828D - Advance

Overview

When programming the SINUMERIK CNC in G-code, it's very helpful to take advantage of the vast selection of standard cycles. This is even more true when using a CAM system and trying to take advantage of the full power of the control.

- Advanced irregular contour milling cycles
- Introduction to free contour programming
- Introduction of Advanced technology milling cycles
 - Path mill
 - Profile pocket
 - Residual metal removal
- Island milling with SINUMERIK

Webinar on Milling 828D - Highlevel

Overview

It's a natural progression when beginning to master the standard programming of a CNC to start exploring more advanced techniques. Inevitably, this will lead you to variable-based programming, an advanced method of programming that utilizes instructional statements with definable variables to quickly change the result of a program.

- Benefits of high-level programming
- Unconditional statements with variables
- Comparative operands with variables
- Complex programming with R variables

Webinar on Turning 828D - Basic

Overview

Basic setup and programming principles of the SINUMERIK in turning applications. First, we'll begin with the creation and setup of tools. Next, we'll methodize a part program through the program guide user interface.

- Creation and part program management through the program manager function
- Turning using standard cycles
- Drilling Cycles – Centering, Drilling, Deep hole
- Editing, simulation, and running of programs
- Using standard cycles applied to a specific part

Webinar on Turning 828D - Advance

Overview

Programming with program guide for turning is intended to step a user through the basics of G-code programming in the Siemens SINUMERIK Operate graphical user interface. We will review a more complex part program that shows examples of Complex Turning.

- Introduction to free contour programming
- Introduction of Advanced technology Turning cycles
- Advanced feed strategies for chip handling and reducing insert notching
- Working with the cast or pre-machined blanks
- Contour grooving and plunge turning applications

