

SINUMERIK 5-Axis Programming (NC-PRO-5AX)

Ksa tanm

In this course you will learn how to program 5-axis machine tools in accordance with DIN 66025.

Hedefler

Your tasks will include programming 5-axis machine tools for complex components.

After the course, you will be able to create 5-axis part programs based on workpiece drawings. You will program in accordance with DIN 66025 and with selected high-level language commands.

Hedef Kitle

- CNC Programmer
- Machine operator
- Production Preparation
- CAM Programmer
- PP Developer

Çerik

- Mathematical and kinematic principles of 5-axis programming
- Function with introductory example of the CYCLE 800 swivel cycle
- Calculation and application of the swivel modes (solid angle, projection angle, direct and axis) in the CYCLE 800 on practice parts
- Introduction to the 5 axis transformation Traori
- Programming with TRAORI with round axis angles, Euler, direction vectors and lead/tilt
- Program part repetitions (lines)
- Traori programming with the tool roll side and with 3D tool radius compensation
- Cone shell interpolation on the full cone, partial cone, full and partial cone with rotation of the cone axis
- Program creation and examples with spline interpolation
- Programming in the time frame, i.e. cost-effective program creation with spline interpolation
- Mixed application of left and right mold halves on the deep-drawing tool
- Compendium for the contour line
- 1st case with dist and end mill and singular point machining
- 2nd case with Traori and ball end mill and singular point machining
- 3rd case with vertical ball end mill and single point machining
- 4th case with dist and end mill with corner radius and single point machining
- Overview of the 5 most important axis programming commands
- Short demonstration of NX CAM

Katımc önkoullar

Good knowledge of programming SINUMERIK controls and the SINUMERIK Operate user interface in accordance with the NC-PRO2 (formerly NC-SINOP-P) course.

Not

Included in the course price: Free access to the digital learning platform [SITRAIN access](#) – starting one week before the start of the course until two weeks after the end of the course.

With the Learning Membership, you can deepen or repeat the content of this Learning Event as well as continue your education on other interesting topics.

This course is suitable for users of the Siemens SINUMERIK ONE and SINUMERIK 840D sl control systems. The programming is the same in both cases.

This course is the successor to the NC-SINOPP2 course. The content is comparable and updated.

Tip

Snf eitimi

Süre

5 gün

Dil

en