

---

## CNC machine tools programming with HEIDENHAIN control Workshop TILTING & PROBING – iTNC 530, TNC 320/620/640

**Objective** learning the principles of programming 5-axis machining (3+2) and applying touch probe cycles on a tilted plane

**Duration** 4 days x 8 hours

**Contents**

Tilted machining

- 3D-ROT function
- axis angles vs. spatial angles: definition and rules
- graphic simulation: functions and tricks
- tilting plane in the manual modes of operation
- safe positioning for 5-axis machining
- tilting plane in the NC program: cycle 19 vs. PLANE functions
- NC program structure for tilted machining
- miscellaneous functions for 5-axis machining

Workpiece touch probe

- preset table: setting and datum management
- touch probe cycles in the manual modes of operation
- touch probe cycles in the automatic modes of operation
- measurement and setting of the work plane
- automatic tool compensation
- kinematicsOpt: measurement and optimization of machine kinematic

Q-parameters programming

- Q-parameters applications with tilted machining
- FN9 - FN12 functions: if-then decisions with Q parameters
- FN16 function: formatted output of text and Q parameter values

**Target group** CNC milling machines operators, technologists, CNC programmers, teachers

**Requirements** completion of the *Basic course* or the equivalent knowledge

**Remarks**

- control type to choose: iTNC 530 or TNC 320/620/640
- training is carried out on programming station and on a machine tool
- each participant receives a certificate of participation