
CNC machine tools programming with HEIDENHAIN control Touch probe cycles – iTNC 530, TNC 320/620/640

Objective the participants can apply the touch probe cycles in manual and automatic mode and log the measured values

Duration 2 days x 8 hours (3-axis machine tool), 3 days x 8 hours (5-axis machine tool)

Contents Basic knowledge

- parameters settings and preparation for calibration
- touch probe table (only with TNC 320/620/640)
- preset table: setting and datum management

Touch probe cycles in the manual modes of operation

- calibrating the touch probe
- compensating workpiece misalignment
- workpiece datum settings
- application of touch probe cycles on the machine without the touch probe

Touch probe cycles in the automatic modes of operation

- compensating workpiece misalignment
- workpiece datum settings
- workpiece measurement
- measurement protocols
- automatic tool compensation
- tilted plane measurement (only with 5-axis machine tool)
- kinematicsOpt: measurement and optimization of machine kinematic

Parametric functions

- FN26 - FN28: freely definable tables
- FN18: reading system data

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