CNC machine tools programming with HEIDENHAIN control
Advanced course – iTNC 530, TNC 320/620/640

Objective   learning advanced functions and programming techniques

Duration   4 days x 8 hours

Contents
- applications of the point tables and pattern definitions
- miscellaneous functions for coordinate data and contouring behavior
- cycle 32 Tolerance, calculation of maximum feed rate
- SL cycles: free shapes pockets and islands
- data import from DXF / CAD files
- simple contour formula vs. complex contour formula
- rules of FK free contour programming
- converting FK programs → H
- rules of Q-parameter programming
- FN9 - FN12 functions: if-then decisions with Q parameters
- FN16 function: formatted output of text and Q parameter values
- touch probe cycles in the manual modes of operation
- touch probe cycles in the automatic modes of operation
- preset table: setting and datum management
- tool measurement and automatic tool compensation
- indexed tools: creation and application

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