CNC machine tools programming with HEIDENHAIN control
Workshop FK & Q – iTNC 530, TNC 320/620/640

Objective   learning the rules of FK free contour programming and parametric programming

Duration   2 days x 8 hours

Contents   FK Programming
- when is FK necessary
- requirements for FK programming
- overview of FK functions
- contours programming
- converting FK programs → H
- FK-SL combination
- Tips and tricks

Q-parameters programming
- applications
- entry and resetting Q / QL / QR
- numbers pre-assigned and reserved Q-parameters
- overview of the parametric FN functions
- formulas and calculation rules for mathematical formulas
- 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
- each participant receives a certificate of participation