CNC machine tools programming with HEIDENHAIN control
Workshop CAD-CAM-TNC – iTNC 530, TNC 320/620/640

Objective   optimization of the postprocessor and NC program

Duration   1 day x 8 hours

Contents   Fundamentals
- process chain
- machining types

Postprocessor
- subprograms
- accuracy from the postprocessor
- useful postprocessor configurations
- calling any NC program as a subprogram
- output format of NC programs

Optimizing the NC program
- cycle 32 tolerance
- comments vs. sections
- miscellaneous functions
- examples of programs

Control configuration options
- overview of software options
- data import from DXF / CAD files
- kinematicsOpt
- users parameters

Target group   postprocessor developers, technologists, CNC programmers, teachers

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

Remarks
- training is carried out on programming station and on a machine tool
- each participant receives a certificate of participation