CNC machine tools programming with HEIDENHAIN control Workshop CNC MACHINE OPERATOR

Objective mastering the skills of operating the control, setting the workpiece datum, preparing the tools and safety running the NC program

Duration 1 day x 8 hours

Contents First steps
- switching on the machine
- operation of the control interface
- coordinate systems on machine tools
- error messages and help system

Workpiece setup
- touch probe cycles in the manual modes of operation
- application of touch probe cycles on the machine without the touch probe
- preset table: setting and datum management

Setting up tools
- tool table: tools parameters definition
- pocket table: tools management in the magazine
- tools measurement: tool probe cycles

Test run operating mode
- graphic settings
- activation of the tool table
- workpiece blank form in the working space
- conditional functions: M1, skipping NC blocks

Program run operating mode
- block scan function: start NC program at any desired block
- program pause and continuation
- conditional functions: M1, skipping NC blocks
- program interruption

NC programs transfer
- USB device
- TNCremo software

Target group CNC milling machines operators

Requirements CNC fundamentals

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