NCTRENER.PL

HEIDENHAIN PARTNER TRAINING

CNC machine tools programming with HEIDENHAIN control
Transfer course – TNC 320/620/640

Objective getting knowledge about the differences between the iTNC 530 control type and the new series TNC 320/620/640

Duration 2 days x 8 hours

Contents

Basic knowledge
- defining the blank form
- NC program layout
- tool table vs. tool management
- status display for Q parameters
- user parameters iTNC vs. TNC

Programming
- review of new cycles
- fitting programs with SL cycles
- cutting data calculator
- freely definable tables and FN26 - FN28 functions
- data import from DXF / CAD files

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

Workpiece touch probe
- touch probe table
- touch probe cycles in the manual modes of operation
- preset table iTNC vs. TNC

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

Target group CNC milling machines operators, 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