
CNC machine tools programming with HEIDENHAIN control Transfer course – TNC7

Objective mastering the skills of operating and programming the TNC7 control based on knowledge of the iTNC 530 / TNC 640/620/320 controls

Duration 2 days x 8 hours

Contents Basic knowledge

- operating the TNC7 control interface
- overview of operating modes
- screen layout and workspaces
- files and tables management

Programming

- selecting the Editor operating mode
- editor configuration
- defining the workpiece blank
- entering and editing NC functions
- saving and managing NC sequences

Simulation

- description of available functions
- center of rotation in the simulation
- simulating an NC program up to a certain NC block
- cutout view in the simulation
- measuring function
- exporting a simulated workpiece as STL file
- model comparison

Graphical programming

- first steps in graphical programming
- exporting contours from graphical programming
- importing contours into graphical programming

Cycles for optimized contour milling OCM

- OCM fundamentals
- program structure for OCM cycles
- roughing / finishing / chamfering

Datum settings with touch probe cycles in the manual modes of operation
NC programs transfer and safe program start in automatic mode of operation

Target group CNC milling machines operators, technologists, CNC programmers, teachers

Requirements completion of the *Basic course (iTNC / TNC)* or the equivalent knowledge

Remarks

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

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 iTNC 530* or the equivalent knowledge

Remarks

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