NCTRENER.PL

HEIDENHAIN PARTNER TRAINING

CNC machine tools programming with HEIDENHAIN control
Basic course – iTNC 530, TNC 320/620/640

Objective the course participants can create NC programs from workpiece drawings with HEIDENHAIN conversational programming

Duration 5 days x 8 hours

Contents

Basic knowledge
- operation of the control interface
- coordinate systems on machine tools
- tool table: tools parameters definition
- pocket table: tools management in the magazine
- preset table: setting and datum management
- absolute and incremental data input
- programs management

Contours programming
- cartesian contour description
- polar contour description

Cycles programming
- face milling
- drilling, milling pockets, studs and slots
- SL cycles: free shapes pockets and studs
- polar and linear points patterns
- coordinate transformation cycles
- trochoidal slot milling

Programming techniques
- program section repeats
- subprogramming
- nesting
- templates

Data import from DXF / CAD files
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 CNC fundamentals, ability to read technical drawings

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