

System 800 Digital Programming Course (NC-800-PK)

Short Description

This course is aimed at personnel who wish to combine the theoretical aspects of CNC programming with a 'hands on' practical approach. The course covers ISO programming of the Sinumerik 810D, 840D and 840Di digital controls in Turning and Milling configurations.

Objectives

On completion of the course the student will be able to:-
Create NC programs with linear and circular motion
Create programs using R parameters and user variables
Create subprograms
Parameterise standard cycles
Testing of the NC programs using Simulation
Back up NC programs, Zero offsets and Tool offsets

Content

Programming according to DIN 66025 + enhancements
Control Overview
Directory Structure
Workpiece creation
Co - ordinate systems
Frame Concept - Zero offsets
Tool offsets
Program Jumps - Labels
Subprograms
R parameters and system variables
Global User Data
Macros
Programming of cycles
Positional Control
Spindle Control
Polar co - ordinates
Program co - ordination
Auxiliary codes

Prerequisites

Some exposure to CNC machining techniques and terminology would be of benefit but is not essential.

Type

Face-to-face training

Duration

5 days

Language

en