CNC Milling Course 1 - Controller - Basic 3 Axis (X , Y & Z) Milling

Dates: 6th - 8th of September

Location: IMR training facilities, Mullingar

Duration: 3 Day Course

Delivery: Face to Face Classroom and Workshop (Facilitator Led)

Contact: marketing@imr.ie

Register Interest

IRISH MANUFACTURING RESEARCH

MILLS CNC TRAINING ACADEMY

Course Aim

This is a basic Fanuc milling course designed for people with some knowledge and experience of the machining process and CNC controls but who want to acquire the necessary skills to program 3-axis Mills.

The course is aimed at operators and setters looking to acquire new and upgrade existing skills, and at adult learners with knowledge and experience of the machining process looking to make the step up to CNC Programming. Designed for operators in engineering already experienced using the machine but want that extra bit of knowledge to understand what the programs are doing and how to do simple editing and adjustments of offsets etc.

The course is suitable for customers with either Doosan 3-axis mills or other manufacturers' Fanuc controlled machines with a similar configuration.

Day 1 - Module 1 - Fanuc Milling ISO Code

1. Introduction, Explanation & Purpose of the Course
2. Machine Layout, work datums, G54~G59 & G28 Reference
3. Explanation of Absolute G90 & Incremental G91 Programming
4. Tool Offsets. D&H Codes & Applying G43 Height & Radius Offsets
5. Movement Using the Rapid, Feed & Circular Commands ie: G0, G1, G2 & G3
6. Explanation of Feed-rates G94 mm/min & G95 mm/rev
7. Applying Cutter Comp G41 & G42 with Auto Corner Chamfer Function
8. Using a Sub-Program to Repeat Shapes etc.
9. Using Rounding & Chamfer Function & Facing Off & Adjusting Offsets

Day 2 - Module 2 - Fanuc Milling ISO Code

1. G73, G74, G80~G89 Canned Cycles for Drilling
2. G84 & G74 Rigid Tapping
3. G10 Data Input for Offsets
4. G68 Co-Ordinate Rotation
5. G16 & G15 Polar Command
6. Examples of using the above
7. Thread Milling and G2 & G3 using I, J & K Function
8. Review of Examples done at Home/Work

Day 3 - Module 3 - Fanuc Milling NC Guide Conversational

1. Fixed Forms, Creating, Editing & Deleting
2. Tool Data Base, Creating and Setting
3. Billet Size, Start Menu, Cycles & End Menus
4. Pocketing & Facing Cycles
5. Hole Machining & Thread Milling menus
6. Pocketing, Grooving Contouring Irregular Shapes
7. Copy, Rotate, & Mirror Shapes, Engraving
8. Time Studies, NC Conversion, Review of Examples done, Back Up Data.

Learning Outcomes

At the end of the course the participants will have acquired basic knowledge for CNC programming and operation of CNC Precision Mills. They will have gained an understanding of ISO G codes and cycles relevant to CNC precision Milling, giving them the confidence to make understand CNC programs and make minor edits.

Participants Profile

This course is designed for operators currently operating or beginning to operate CNC Mills at an operator level i.e. part loading, basic tool changing, wear adjustments etc

Certification / Awarding Body

CPD Engineers Ireland

Register Interest

*Limited to 8 participants