Milling 4 & 5 Axis VMC with Fanuc Control

Dates: 21st March - 24th March

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

CNC TECHNOLOGY

Course Aim

This is an advanced 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 4 & 5 axis mills.

Designed for operators/programmers with a basic understanding the standard ISO codes for Milling but need to upskill and use the facility of the 4th Axis on a Vertical Milling Machine and B axis in a Vertical 5 axis machine.

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.

The course is suitable for customers with Fanuc controlled machines with a 4 or 5 axis configuration.

Day 1: Module 1

1. Introduction to Fanuc Milling
2. General Layout of Machine
3. Walk Through Control Panel
4. Memory and Library
5. Axis Configuration X, Y & Z
6. Making A Program
7. Reference Return G28/G30
8. End of Program
9. Absolute and Incremental Movements G90/G91
10. Rapid Traverse & Feed Rate G00/G01
11. Feed/Rev & Feed/Mm G94/G95
12. Using Codes, I, J & K G02//GG03
13. Plane Selection G17/G18/G19
14. Helical Interpolation
15. Programming Examples

Day 2: Module 2

1. Day 1 Overview
2. Work Co-Ordinate Setting
3. Applying Tool Length Offsets
4. Programmable Data Input Using G10
5. Cutter Compensation G40/G41/G42
6. Applying Radius Offsets
7. Drilling, Tapping & Boring Canned Cycles G80-G89
8. Initial Heights & Return Heights G98/G99
9. Exact Stop and Dwell G04
10. Rigid Tapping Function
11. Programmable Mirror Image
12. Programming Examples

Day 3 Module 3

1. Day 1 & 2 Overview
2. Polar Co-Ordinates
3. M Code Description
4. Sub Program Call M98/M99
5. Optional/Program Stop M00/M01
6. Editing Programs
7. Custom Macros
8. Changing Parameters
9. Programming Examples
10. Summary Of Course Content

Day 4 Module 4

1. Day 1, 2 & 3 Overview
2. Axis Configuration X, Y, Z, A & B Additional Axis
3. B Axis Programming Including Indexing Unit
4. G17, G18 & G19 Plane selection for helical and three-dimensional Use
5. Data Input f0or Tool Offsets, Work Offsets & Parameters Using G10
6. Additional Work Offsets & Work Co-Ordinate Setting for G52/G59
7. Description of M Codes
8. Macro Programming G65/G66/G67
9. Cylindrical Interpolation
10. Programming Examples
11. Summary Of Course Content

Learning Outcomes

At the end of the course the participants will have acquired advanced knowledge for CNC programming and operation of CNC Milling machines which have 4 & 5 axis capabilities.

Participants Profile

This course is designed for operators/programmers already competent in understanding the standard ISO codes for CNC Milling and require upskilling and to use the facility of the 4th and 5th Axis on a mill. The ideal candidates are already proficient in CNC operation and should have a good foundation in ISO G – codes.

Certification / Awarding Body

CPD Engineers Ireland

Register Interest

**Limited to 8 participants*