# Milling 4 & 5 Axis VMC with Fanuc Control

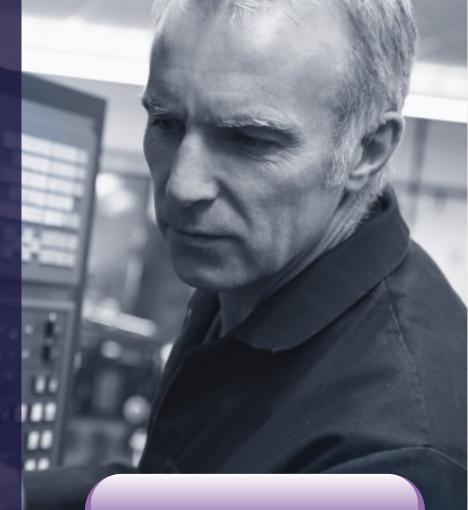
Dates: 21st March - 24th March

Location: IMR training facilities, Mullingar

**Duration: 3 Day Course** 

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

Contact: marketing@imr.ie



#### **Register Interest**



# 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

Day 1 & 2 Overview
Polar Co-Ordinates
M Code Description
Sub Program Call M98/M99
Optional/Program Stop M00/M01
Editing Programs
Custom Macros
Changing Parameters
Programming Examples
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 foor 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** 

\*Limited to 8 participants

**<u>Register Interest</u>**