



## PROGRAMMING COURSE

**Course title : M2 Fanuc 16M/18M/21M/OM/30M  
(4 Axes Vertical & Horizontal Machining Centres)**  
**Course length : 4 days**                      **Course cost : £600.00 (+ VAT)**

### Day 1: Monday

**Start Time**

**Finish Time**

Introduction & General Layout of Machine.  
 Operator , Keyboard & Soft-key Explanation.  
 Axes Configuration. X, Y, Z .  
 Library & Memory Configuration.  
 How to Start Making a Program, Safety Start.  
 How to End a Program.  
 G28~G30 Reference Point Return.  
 G90~G91 Absolute & Incremental Programming.  
 G00~G01 Rapid Traverse & Feed Rate Commands.  
 G94~G95 Feed/Rev or Feed/mm.  
 G02~G03 Codes Using "I", "J" & "K".  
 Helical Interpolation.  
 G17,G18 & G19 Plane selection.  
 Work Co-Ordinate Setting for G52~G59 & G92.  
 G43, G44 & G49 Applying Tool Length Offsets.  
 Manual Guide Examples (if used).  
 Using G10 Function for Programmable Data Input.  
 Examples Using The Above Information.

9:30 AM

5:00 PM

### Day 2: Tuesday

**Start Time**

**Finish Time**

G40, G41 & G42 Cutter Compensation.  
 Applying Radius Offset.  
 G80~G89 Canned Cycles for Drilling, Tapping & Boring etc.  
 G98~G99 Initial & Return Heights G32 Threading Cycle.  
 G04 Dwell & Exact Stop.  
 Rigid Tapping Function & Explanation.  
 Scaling ,Programmable Mirror Image.  
 Polar Co-ordinate Command.  
 Description of M codes relative to type of machine.  
 M98~M99 Sub Program Call & Use.  
 Using Block Delete.  
 Optional & Program Stop, M00 & M01.  
 Manual Guide Examples (if used).  
 Editing Programs.  
 Programming Examples.

9:00 AM

5:00 PM

### Day 3: Wednesday

**Start Time**

**Finish Time**

Brief Explanation of Custom Macro's ( see also Custom Macro Course).  
 Programming Examples.  
 Program Input & Output through RS232C.  
 Manual Guide Examples (if used).  
 Tool Life Management & Tool Load Monitor (if applicable).  
 Baud Rate Setting, Stop Bits etc.  
 How to change a Parameter.  
 Time finishing on 3rd Day To be Advised.

9:00 AM

4:00 PM

Continued overleaf

## M2 Fanuc 16M/18M/21M/OM/30M (4 Axes Vertical & Horizontal Machining Centres)

### Fanuc Horizontal Programming Course Content (Additional 1 Day).

Day 1: Thursday	Start Time	Finish Time
<p>Introduction &amp; General Layout of Machine. Axes Configuration. X, Y, Z, "A" &amp; "B" Additional Axes. G02~G03 Codes Using "I", "J" &amp; "K", Helical Interpolation. B Axis Programming including any other Indexing Unit. G17,G18 &amp; G19 Plane selection for Helical and Three Dimensional Use. G10 Data Input for Tool Offsets, Work Offsets, &amp; Parameters. Work Co-Ordinate Setting for G52~G59, Additional 48 Work Offsets. Examples Using The Above Information. Description of M codes relative to type machine. M98~M99 Sub Program Call &amp; Use. G65, G66 &amp; G67 Macro Programming and uses (Custom Macro B). Output of Data through RS232. Editing Programs (Copy, Move &amp; Merge Functions.) Programming Examples. Cylindrical Interpolation (if applicable). Program Input &amp; Output through RS232C.Baud Rate Setting, Stop Bits etc. Time finishing on 4th Day To be Advised</p>	9:30 AM	4:00 PM

NB 1: Machine availability for use on examples will vary and will be advised upon on the day.

NB 2: M2 course is for 4 Axes Horizontal & Vertical Machining Centres, M1 Course may also be required.