Fanuc – Milling

Day 1

9:00am 4:00pm

- General Layout of Machine & Keyboard Explanation
- Axes Configuration.
- Program Memory Arrangement
- How to edit a program and create new
- Tool Offsets
- Work Offsets
- G10 Programmable data input
- How To Start making a Program. Safe Start.
- G20-G21 Inch-Metric, G40, etc.
- G Code Description Type A, B or C.
- M code descriptions
- Other addresses explained
- G94-G95 Feed/mm - Feed/rev.

Day 2

9:00am 4:00pm

- G00-G01 Rapid Traverse & Feed Rate Commands.
- Absolute & Incremental Programming, G90 & G91
- G02-G03 Circular Interpolation using “R”, “I” & “J”.
- G17-G18-G19 Plane Selection
- Helical Interpolation.
- G28 Reference Point return.
- G30 Setting 2nd, 3rd, 4th Reference Point return.
- Test piece for G01 - absolute and incremental
- Test piece for G02/G03 - absolute and incremental
- How To End a Program. M02, M30.
- M98-M99 Sub-Program use & nesting.
- G43 & H Offset
- G41-G42 Cutter Compensation
- G80-G89 Canned Cycles for Drilling, Tapping & Boring etc.
- G98-G99 Initial & Return Heights
- G04 Dwell
- Test Piece 1
<table>
<thead>
<tr>
<th>Day 3</th>
<th>Start</th>
<th>Finish</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>9:00am</td>
<td>4:00pm</td>
</tr>
</tbody>
</table>

Rigid Tapping Function & Explanation.
Test Piece 2
Test Piece 3
C, R & A Direct Drawing Input.
C & R Chamfer Corner Radius Function.
Test Piece 4
Test Piece 5
Inputting and Outputting Programs (RS232 / Mem Card)
Backing up the control
P/S Alarms
Brief Explanation of Macro Programming & uses. (See also Macro Course).
Program your own component (if time left)