



PROGRAMMING COURSE

Course title : Siemens 840D/Horizontal & Vertical 4 Axis Machining Centre
Course length : 4 days **Course cost : £1,000.00 (+ VAT)**

Day 1	Start Time	Finish Time
<p>Design and structure of the control Numbering of the part programs Introduction to coordinate system for the machine General structure of the part programs Explanation of G code groups, modal, non modal and active at start G Codes - Group 3 – Programmable zero offset (G59) Group 8 – Adjustable zero offsets (G54 – G57) Group 1 – Linear movements (G0, G01, G02, G73 etc) Group 7 – Tool nose compensation (G40, G41, G42) Group 13 – Inch/metric system selection (G70, G71, G700, G710) Group 14 – Absolute or incremental programming (G90, G91) Group 15 – Feedrate type (G94, G95, G96, G97) Write example program</p>	9:30 AM	5:00 PM
Day 2	Start Time	Finish Time
<p>Explanation of M code groups M Codes – Explanation of M codes and extensions (Unit number) M functions for the spindles (Group 1) M functions for the coolant valves (Group 4) M functions for the axes (Group 11) (mirror image etc) M functions for the handling system (Pallet system) Short contour description Write Example program</p>	9:00 AM	5:00 PM
Day 3	Start Time	Finish Time
<p>Explanation of Siemens cycles Cycle82 – Drilling Cycle84 – Rigid tapping Cycle93 – Grooving Cycle94 – Undercut Explanation and use of Wait marks Machining planes (G17, G18, G19) Write Example program</p>	9:00 AM	5:00 PM
Day 4	Start Time	Finish Time
<p>Complete program examples</p>	9:00 AM	4:00 PM