



ISO Programming Course

Title: Siemens 3 Axis Milling

Controls: 840D, 810D, 828

Duration: 3 Days

Day 1	Start	Finish
G & M Code Introduction	9:00am	4:00pm
Machine Motion (G90 & G91)		
Workpiece Co-ordinate System (G54-G59 & G505–G510)		
Program start		
Toolchange		
Programmable defaults		

Day 2	Start	Finish
Work Co-ordinate Setting	9:00am	
Tool Length Setting		
Cutter Radius Compensation (G40, G41, G42)		
Helical Interpolation		
Hole canned cycles (CYCLE81 – 89)		
Test Piece 1		

Day 3	Start	Finish
Rigid Tapping Function & Explanation.	9:00am	4:00pm
Test Piece 2		
Test Piece 3		
C, R & A Direct Drawing Input.		
C & R Chamfer Corner Radius Function.		
Test Piece 4		
Test Piece 5		
Brief Explanation of R Variable Programming & uses.		