Siemens – 5 Axis Mill

Controls: Siemens 840D 810D
Duration: 5 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		
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.		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.		

Day 4	Start	Finish
5 Axis configuration Iso standard Axis naming and Why Cycle 800 Programming and how it works Examples of Cycle 800 in use	9:00am	4:00pm

Day 5	Start	Finish
Traori programming and it uses Differences between Cycle 800 and Traori Which method to use and when Examples of Traori in use	9:00am	4:00pm