



Siemens – 3 Axis Mill

Controls: Siemens 840D 810D
Duration: 3 Days

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