ISO Programming Course

Title: Basic Milling Course for Fanuc Controls Oi-M/16i-M/18i-M/21i-M /3xi.

Duration: 3 Days

Day 1	Start	Finish
	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	Start	Finish
	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

Day 3	Start	Finish
	9:00am	4:00pm

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)