



ISO Programming Course

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

Duration: 3 Days

Day 1	Start	Finish
	9:00am	4:00pm
<p>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 G98-G99 Feed/rev & Feed/mm. G50-G92 Clamping Maximum Spindle Speeds.</p>		

Day 2	Start	Finish
	9:00am	4:00pm
<p>G00-G01 Rapid Traverse & Feed Rate Commands. G02-G03 Circular Interpolation using "R", "I" & "K". Absolute & Incremental Programming, "U" & "W". G17-G18-G19 Plane Selection G28 Reference Point return. G30 Setting 2nd, 3rd, 4th Reference Point return. Test piece for G01 - absolute and inc Test piece for G02/G03 - absolute and incremental How To End a Program. M02, M30. M98-M99 Sub-Program use & nesting. G22-G23 Stored Stroke Protected Area. G41-G42 Cutter Compensation, Imaginary Tool Points. G70-G73 Multi-Repetition Cycles. G90-G94 Canned Cycles Roughing, & Facing (if required). G80 - G89 Canned cycles. G96-G97 Constant Surface Speed Control. G32 & G92 Threading Cycles.</p>		

Day 3	Start	Finish
	9:00am	4:00pm
Test Piece 2 (Drill - Rough Bore - Finish)		
G74-G75 Canned Cycles for Grooving & Drilling.		
Test Piece 3 (Side Grooving - Face Grooving - Drilling)		
G32-G92 & G76 Threading Cycles.		
Test Piece 4 (Rough Turn - Drill - Bore - Thread – Part)		
C & R Chamfer Corner Radius Function.		
Test Piece 5 (Turn using direct Drawing)		
Inputting and Outputting Programs (RS232 / Mem Card)		
Backup control		
P/S Alarms		
Brief Explanation of Macro Programming & uses. (See also Macro Course).		
Program your own component (if time left)		