



PROGRAMMING COURSE

Course title : T1, T2 & T3 Turning Course Training Module for Fanuc Controls OT/16T/18T/21T /30T.

Including C Axis Milling & Y Axis Machines.

Course length : 4 days

Course cost : £600.00 (+ VAT)

Day 1: Monday

Start Time

Finish Time

General Layout of Machine, Control & P.M.C. Keyboard Explanation.
 Axes Configuration.
 G28 Reference Point return.
 G30 & G130 Setting 2nd Reference Point return.
 Absolute & Incremental Programming, "U" & "W".
 Work Shift Setting & G10 use.
 Work Co-ordinates G54~G59 & G10 use.
 How To Start making a Program. Safety Start.
 G20~G21 Inch~Metric, G40, etc.
 G Code Description Type A, B or C.
 Memory Arrangement and Size Calculation.
 G00~G01 Rapid Traverse & Feed Rate Commands.
 G98~G99 Feed/rev & Feed/mm.
 G50~G92 Clamping Maximum Spindle Speeds.
 G02~G03 Circular Interpolation using "R", "I" & "K".
 G70,G71,G72, & G73 Multi-Rep Cycles.
 C & R Chamfer Corner Radius Function.
 C, R & A Direct Drawing Input.
 G96~G97 Constant Surface Speed Control.
 How To End a Program. M02, M30.
 Programming Examples using the above data.

9:30 AM

5:00 PM

Day 2: Tuesday

Start Time

Finish Time

G32~G92 & G76 Tapping & Threading Cycles.
 G84 Canned cycle for Tapping.
 M98~M99 Sub-Program use & nesting.
 G74~G75 Canned Cycles for Grooving & Drilling.
 G83 Canned cycle for drilling.
 G90~G94 Canned Cycles Roughing, & Facing (if required).
 G41~G42 Cutter Compensation, Imaginary Tool Points.
 Applying & Cancelling Tool Offsets, Geometry & Wear.
 G10 Programmable Offset Input.
 G54~G59 Work Co-Ordinate selection, Setting Work Shift.
 G22~G23 Stored Stroke Protected Area.
 Manual Guide Use & Fixed Forms (if applicable).
 M Codes explanation (relative to the machine and available options).
 Programming Examples using the above data.

9:30 AM

5:00 PM

Day 3: Wednesday

Start Time

Finish Time

Brief Explanation of Macro Programming & uses. (See also Macro Course).
 Editing Programs. Copy, Move & Merge Functions (If Available).
 Programs Input & Output through RS232C, Baud Rate Setting, Stop Bits etc.
 Tail-stock Programming (if applicable).
 Programming of Manual Guide.
 Setting Up Fixed Forms.
 Tool life management (Optional).

9:00 AM

1:00 PM

Continued overleaf

T1, T2 & T3 Turning Course Training Module for Fanuc Controls OT/16T/18T/21T /30T.

Training Module for Fanuc OT/16T/18T/21T /30T (C Axis Milling)

Day 3: Wednesday

Start Time

Finish Time

General Layout of Machine & Axes Configuration X, Z, C.
M code explanations relative to the machine.
G83~G85 C Axis Drilling on Z Axis Front Face.
G87~G89 C Axis Drilling on X Axis Cross Plane.
Using Rigid Tap Function on Driven Tools.
Program examples using C Axis.
Manual Guide Use & Fixed Forms (if applicable).

1:00 PM

5:00 PM

Day 4: Thursday

Start Time

Finish Time

Milling on Front Face using G112~G113 Polar Co-Ordinate Rotation.
Milling on Side using G107 Cylindrical Interpolation.
G02 & G03 Circular Interpolation "R" "I" & "K".
G17, G18, G19 Plane Selection.
Using Manual Guide on C Axis Work (if applicable).

9:00 AM

1:00 PM

Training Module for Fanuc OT/16T/18T/21T/30T (Y Axis Machines)

Day 4: Thursday

Start Time

Finish Time

Y Axis Program Drilling on Main Spindle.
Y Axis Program Milling on Main Spindle.
Plane Selection G17 & G19.
Thread Milling.
Using Manual Guide & Fixed Forms (if applicable).
Program Examples using the above information.

1:00 PM

5:00 PM

NB 1: Machine availability for use on examples will vary and will be advised upon on the day.

NB 2:T3 course follows on from the T1 & T2 Basic Turning Courses and all courses should be completed together.