DEPARTMENT OF MECHANICAL ENGINEERING

FANUC – Training Centre

Syllabus:

CNC OPERATION AND PROGRAMMING-TURNING AND MILLING

A thorough knowledge of CNC Part Programming is essential to engineers and operators from machine tool builders and end-users. This course is suitable for Engineers who are responsible for component prove out and for operators to effectively operate the machines. It is also useful to the maintenance engineers to diagnose the faults.

Introduction:

- Designation of machine axis-Turning & Coordinate Systems
- Structure of part program, Basic G & M codes
- Absolute & Incremental systems
- Tool offsets &Tool nose radius compensation

Turning:

- Writing part programs using basic G codes and M codes
- Exercises and practice & Explanation on Tool offsets & TNRC
- Explanation of Canned cycles (G71 to G76)
- Examples & exercises using the Turning cycles
- Direct drawing dimension programming, Outputting program data

Milling:

- Designation of machine axis-Milling, Machine and work coordinate systems
- Set up and use of work coordinate systems
- Review of Basic G codes and M Codes-Milling
- Description of Cutter radius Compensation and Tool length compensation
- Examples & Exercises of milling part program
- Explanation of drilling canned cycles G81 to G89, G73 to G76
- Explanation on polar coordinates, Explanation on Tool life management
- Explanation of Chamfering and corner radius

- Examples and practice with canned cycles

Macro Programming:

- Concept of macro programming & Explanation on Macro variable
- Macro variables for offset data, work zero-point, current position data, group G codes, part Count and total parts required
- Creating alarms and messages using Macro variables
- Conditional and unconditional jumps
- Types of Macro call
- Explanation and examples for simple macro call (G65) and modal macro call exercises (G66) and