

## NC-400

### Fanuc G Code (Tool Path) Programming

---

#### *Course Description*

This class teaches the students the fundamentals of basic programming. Students learn Cartesian Coordinate Systems, G codes, basic M codes, word address, canned cycles, cutter radius compensation, and sub programs. Participants will be given many different examples, as well as write many programs. Safety lines and technique will be covered.

#### *Customer Supplied Training Equipment*

The customer will provide a classroom and access to a machine tool with a Fanuc controller suitable for training purposes. The instructor will need unlimited access to the machining center for the duration of the class.

#### *Topical Outline*

- Cartesian Systems
- Positioning
- Absolute Location
- Incremental Location
- Word Address
- G Codes
- M Codes
- Alpha Codes
- Datum's
- Program Structure
- Part Zero
- Machine Zero
- Reference Planes
- Work Coordinates
- Canned Cycles
- Cutter Radius Compensation
- Sub Programs
- G10 Functions
- Programming Examples
- Machine Tool Options

If you wish to add, remove or concentrate on any of the topics please let us know. We will accommodate any special requests.

#### *Prerequisites*

Participants should have a good basic knowledge of machining practices.

#### *Course Length*

36 hours

#### *CEU's*

3.3