**FANUC 0i-TF CONTROL SPECIFICATIONS**

***TYPE OF CONTROL***

* 32-Bit Multi-Processor Continuous Path Control

***HARDWARE COMPONENTS***

* 8.4” Color TFT LCD Screen
* PMCIA Flash Memory Card Interface
* USB Port
* Ethernet Port
* RS232 Interface

***SCREEN DISPLAY***

* Window Oriented Operator Interface
* Display of Current Block and Position Check Screen During Program Execution
* Screen Texts: English (24 additional languages available)
* Machine, Absolute, Relative, and Distance to Go Position Display
* Spindle Speed / Feed Rate Display
* Spindle / Servo Load Monitor
* Current Alarm / Alarm History Display

***OPERATION***

* Program Protection on Machine Control Panel
* 512Kbyte Part Program Storage
* 400 Part Programs Storage
* Built In Run Hour / Parts Counter / Cycle Time Display
* Tool Life Management
* Tool Measurement Compensation (with Tool Presetter)
* Background Editing
* Help Function
* Graphic Display
* 1 Position Spindle Orientation
* Optional Block Skip
* Program Restart
* Offset Guard Function
* Rigid Tapping

***MODES***

* Automatic
* Control of Automatic Mode by:

-     Feed Hold / Cycle Stop

-     Block Skip

-     Single Block

-     Optional Stop

-     Dry Run

* Jog
* MDI (Manual Data Input)
* Edit

***MACHINE CONFIGURATION FOR AXES***

* Inch or Metric Programming
* Switching Between Metric and Inch for Input and Offsets, Display, and Programmed Path
* Feedrate and Rapid Traverse: Minimum Input Feedrate in Inches / min = .0001 inches / min
* Revolution Feedrate: Minimum Input Feedrate in Inches / Rev. = .0001 Inches / Rev.

***MACHINE SPINDLE CONFIGURATIONS***

* Constant Spindle Speed and Constant Surface Speed Control
* Programmable Maximum Spindle Speed Control
* Thread Cutting: Thread Pitch Constant Minimum Pitch in Inches / Rev. = .0001 Inches / Rev.

***OVERRIDES, OFFSETS AND COMPENSATIONS***

* Feedrate Override 0% to 150%
* Rapid Override 0%, 25%, 50%, and 100% (0% is user adjustable)
* Spindle Speed Override 50% to 120%
* 128 Pair Tool Offsets
* Work Shift / Work Offsets (G54-G59)
* Tool Nose Radius / Tool Geometry / Tool Wear Compensation
* Backlash Compensation

***CNC PROGRAMMING***

* Diameter / Radius Programming
* Insert, Alter, Delete
* Character, Program & Sequence Number Search
* Cut, Copy, and Paste Function
* Corner Chamfer C / Radius R Programming
* Direct Drawing Dimension Programming
* Absolute / Incremental Programming
* Thread Chamfering On/Off
* Mirror Image / Each Axis
* Stored Stroke Check / 1st, 2nd, 3rd, and 4th Reference Return Positions
* Skip Function (G31)
* Sub-Program Call (10 Level Nest)
* Variable Lead Threading (G34)
* Turning Canned Cycles (G90-G94)
* Multiple Repetitive Cycles Type I & II (G70-G76)
* Drilling Cycles (G73, G75, G80-G89)
* Circular Interpolation by R, I, J, K
* Feed Per Minute (G98) / Feed Per Revolution (G99)
* Programmable Data / Parameter Input
* G-Code System A/B/C select
* Custom Macro B / 600 User Variables

***SAFETY AND DIAGNOSTIC FUNCTIONS***

* Safety Routines Permanently Active for Measuring Circuits, Over Temperature, Battery,
* Voltage, Memory, Limiting Switches, Hydraulic System, and Cooling Systems.
* Self-Diagnostics
* Servo Monitoring
* Spindle Monitoring