PWT ETHERNET ULTRALIGHT CONTROL DESCRIPTION:

1. General description

EB NC Control is DSP (Digital Signal Processor) based NC controller specifically designed for multi-axis control integrated with machine and process variables. Process variables are controlling electron beam power. Machine control variables are analog and discrete inputs and outputs that are controlling machine axis and electrical signals.

2. Computer control and interface

State-of-the–art NC computer provide following control functions completely integrated into a single system:

2.1.1 EB NC control integrates beam control module (HV, Filament, Beam current, Focus, and Beam Deflection) with motion control module (X, Y, Z, Rotary, Tilt,) through advanced NC programming.

2.1.2 Process control and part programming allows system repeatability to produce welding parameters, such as HV amplitude, Beam Current, Focus, deflection, axis position and travel speed within a close tolerance.

2.1.3 Data collection of machine variables allows quality assurance by collecting and storing preset and feedback process variables producing a permanent record for further evaluation and quality control.

2.1.4 State of art fiber optic communication between Computer control and process I/O control provides an exceptional degree of noise protection and fast transition of data.

2.1.5 Diagnostic messages provide fast identification of machine status, warnings, and fault identification.

2.1.6 Basic control components are shown in the following pictures. These are:

- Industrial computer with Flat screen monitor
- DSP control card (located in the industrial PC)
- Fiber optic lines (2 cables)
- Industrial I/O rack with control cards

2.1.7 Block diagram is shown in the Control System Picture.

2.1.8 Operator console

NC Control card is located in the industrial PC and provides interface to:

- 32 Discrete Inputs and outputs (used on the operator console, like push buttons, lights, etc.)
- 8 channels on-board 12-bit A/D converter

2.1.9 Card Rack is located in control enclosure and contains:

- servo modules
- analog and discrete modules

2.1.10 Hand held pendant

2.2 NC Controller

The NC Controller utilizes a DSP processor, which offers ultimate power and flexibility to control simultaneously up 32 axes.
The power of the controller is depicted by:

a. / processing speed
b. / trajectory power (axis updated and position correction)
c. / input bandwidth (speed of axis and process variables updated)

NC Controller features:
Maximum number of axis…32
Maximum number of coordinate system…16
CPU Frequency…80 MHz
Instruction per clock cycle…120MHz
Compiles PLC programs …16
Battery Back up
Multi block look ahead function for acceleration control
Multiple communication ports capability

2.3 Card Rack will be located in the control cabinet

2.3.1 Will be provided with 19” rack mount

2.3.2 Servo control module provides:

a. / Connection for six encoders (quadrate encoder)
b. / Connection for MT+, MT-, and Home limit switches
c. / Drive enable 24V open collector output

2.3.3. Digital inputs

a. / 24 Inputs
b. / 24V DC sink / source, optically isolated

2.3.4 Digital outputs

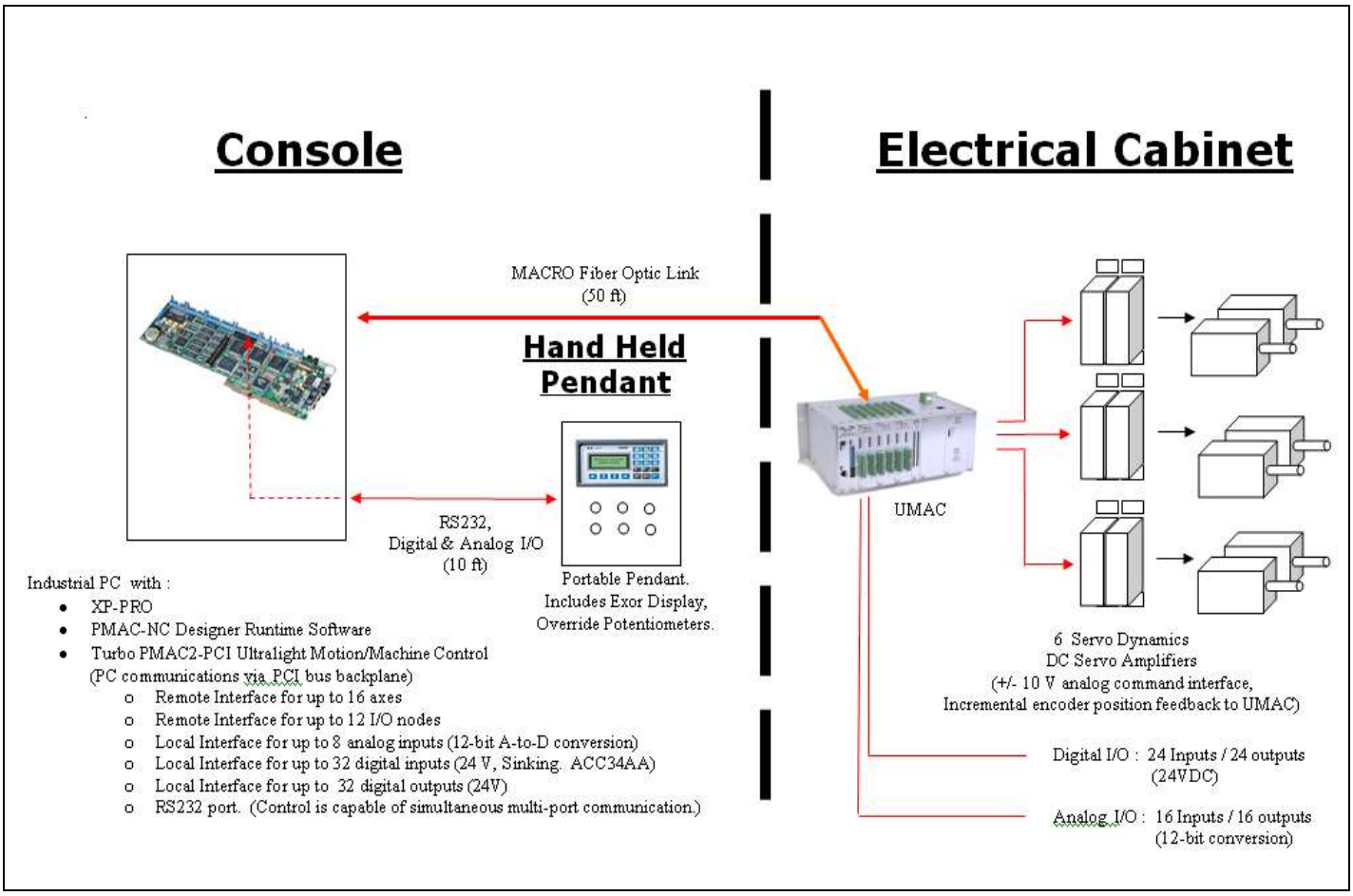
a. / 24 Outputs
b. / Open collector 24V DC

2.3.5. Analog inputs

a. / Analog Inputs Card (16 channels, 12-bit conversion)

2.3.6. Analog outputs

a. / Analog outputs (12 channels, 12-bit conversion)



Control System Diagram

Deflection Monitor

Operator Console

3.0 Programming and interface

The industrial computer CPU communicates through a back plane to the motion control card. The programming software resides on the Windows platform which allows the operator interface to communicate to all cards located in the computer card rack. The motion program is executed in the NC Control cards, by the DSP processor, allowing the user to utilize all PC type hardware and software, networking, etc.

3.1 NC Programming

The NC programming format utilizes a Windows operating platform for machine control programs. The package contains all necessary features for editing, storing, printing and executing program files.

The system executive program integrates:

a. / Axis motion and positioning (X, Y, Z, rot, tilt axis)
b. / Process variables (HV, BC, Fil, Focus, Deflection)
c. / PLC programs (start, stop, limit switches, push buttons, lights, etc.)

The main program, including the axis and process variables control, is running in the foreground, while the PLC program is running in the background. The Visual Basic screens provide the operator screens in the Windows environment. Other programs such as Microsoft Office, Excel, Access, etc., can be running simultaneously to collect or coordinate other activity executed on the machine.

3.1.1 Motion program

Up to 256 programs can be stored in the memory of the NC Controller. Virtually, an unlimited amount of programs can be stored on the hard drive, from which they can be loaded into the NC Controller. Any motion program can be run with a defined coordinate system. Different motion control languages can be used:

a. / simple NC motion program
b. / C+
c. / G-Codes
d. / program editor can be a dedicated in the MMI, Word, or others.

3.1.2 PLC programs

While the motion programs are running sequentially (DSP CPU executes line by line motion program codes) in the foreground, the NC Controller can run up to 32 asynchronous PLC programs in the background. These PLC program performs logical operations such as turning ON/OFF lights based on the status of certain inputs. Typical PLC cycle time is 5-10 milliseconds.

3.1.3. List of some NC control programming functions:

Jogging commands
- Jog to position, jog incremental, jog at speed, and jog to distance

Reporting command
- -report position, speed, position error, velocity error

Buffer control commands
- -establish lead screw compensation,
- -lead screw compensation table

Motion command statements
- -move commands
  - -simple move, blended, rapid
  - -circle move

Move mode commands
- -linear
- -rapid
- -circle (used for circular interpolation)
- -PVT (position-velocity-time)
- -Spline (torque limited)
- -CC type (cutter compensation)

Axis attribute commands
- -ABS absolute
- -INC increments

Move attribute commands
- -TM time of move
- -F federate
- -TA acceleration time
- -TS S-curve time

Variable assignment
- -I, P, M Q type of variables

Program Logic Control
- -OR, AND, IF, ELSE, WHILE, WAIT

Action commands
- -send message
- -enable PLC

Variables

-global
-motor variables
-motor movement variables
-motor servo variables
- PID tuning loop
Coordinates system variables
Encoder variables

3.1.4 Additional features will be provided

- Linear Axis Interpolation
- Circular axis interpolation
- Axis acceleration / deceleration
- Axis positions collection
- Process variable override
- Program control
  -start, shutdown, home
- Data logging
- Collection /Digitizing System

3.1.5 System displays

- File directory display
- Part program display
- Data logger display
- Axis position display
- Process monitor display
- Error Queue Display
- Analog meter display
- Timer display
- I/O display
- Servo monitor display
- Servo tuning
- Analog set point display

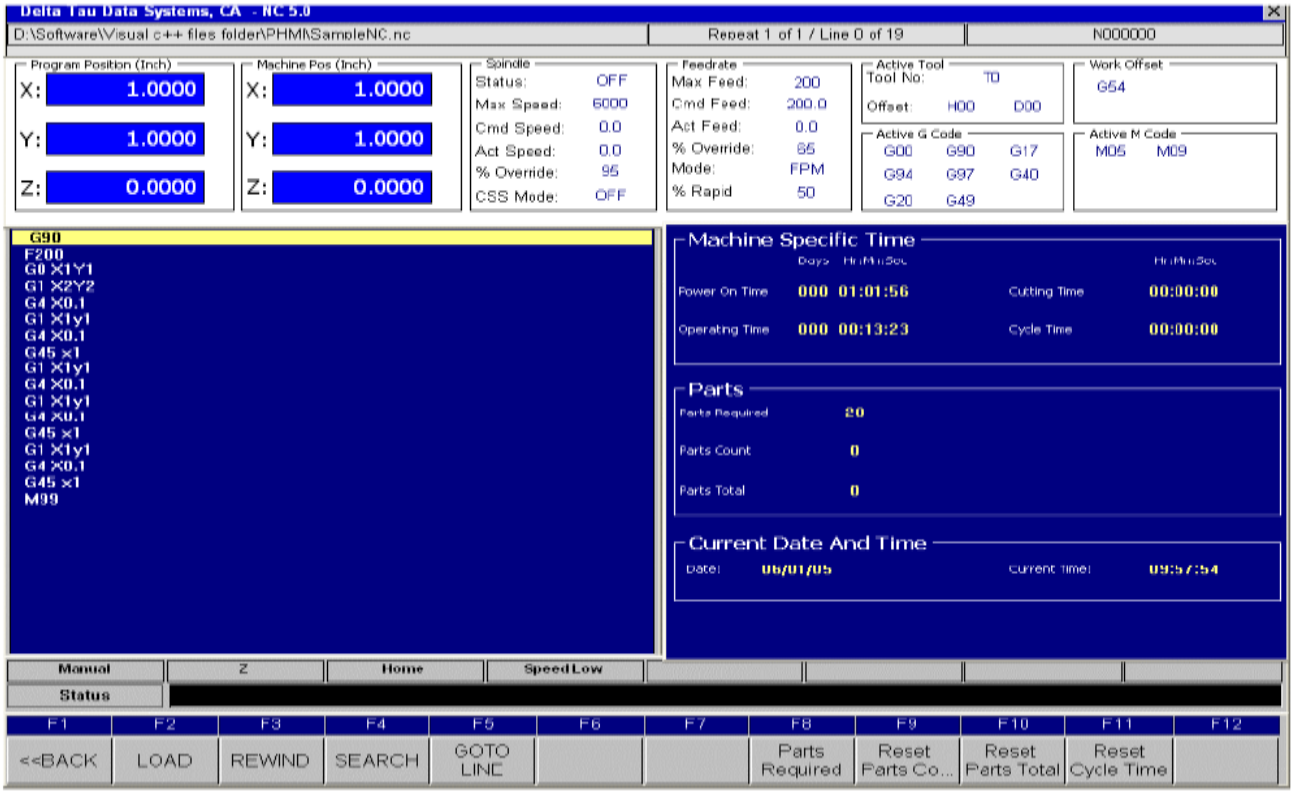
CNC Pro2 Machine Software

CNC Pro2 software is an HMI Designer rapid development utility. All screens and functionality were created within the HMI environment. The software is distributed as a CNC human machine interface with built in customizable standard features for **Electron Beam Welding**. It is customized with respect to number of axes, type of machine, tool offset display, custom messaging, etc. The software is customized by the HMI Designer software. The HMI Designer enables the user to re-configure existing screens as well as design custom new screens and functionality.

CNC Pro2 is a Windows-based customizable GUI for PC based CNC control. NC HMI has all of the features of previous versions with the added advantage of easy user screen customization. This new version brings unprecedented levels of block throughput speed, motion accuracy, and the flexibility of a PC-based environment to the shop floor.

The Windows-based environment allows users to combine NC HMI with their favorite PC compatible CAD/CAM or conversational package. This gives the operator or programmer the flexibility to program directly at the machine or remotely at a desk. The multi-tasking utilities of the Windows operating system give the user the capability to run CNC parts while programming simultaneously. Connectivity tools such as Ethernet and USB2.0 are easy to set up and provide unparalleled reductions in time spent transferring part program files. The part program size is limited only by the hard drive space.

The Main Operator Screen



Software Features:

- **Windows XP, 7, 8, & 10 Compatible**
- **Fully Configurable Open Architecture GUI**
- **Built-in AutoPilot Software for creating standard PLC's**
- **Up to 1000 blocks/sec. Throughput with Segmented Block Look ahead (Hardware Dependent)**
- **Open PLC**
- **Linear, Circular and Helical Interpolation**
- **S-Curve High Speed Jerk Control**
- **Enhanced graphical diagnostic features**
- **User-friendly NC operator interface**
- **No limit on part program size, the hard disk space is the only limitation**
- **User definable G, M, and T codes**

- **User programmable error messaging using PLC, includes logical pop-up windows**
- **PC-based network and USB connectivity**
- **Feedrate and rapid traverse limited only by machine dynamics**
- **One or two dimensions lead screw compensation/backlash tables**
- **Tool radius/length/wear compensation**
- **3D-cutter compensation**
- **Linear, circular, and helical interpolation. Cylindrical interpolation capability**

Turbo PMAC Look ahead Function

Turbo PMAC can perform highly sophisticated look ahead calculations on programmed trajectories to insure that the trajectories do not violate specified maximum quantities for the axes involved in the moves. This permits the writing of the motion program simply by describing the commanded path. Vector feedrate becomes a constraint instead of a command; programmed acceleration times are used only to define corner sizes and minimum move block times. Turbo PMAC will control the speed along the path automatically (but without changing the path) to ensure that axis limits are not violated.

Look ahead calculations are appropriate for any execution of a programmed path where throughput has been limited by the need to keep execution slow throughout the path because of the inability to anticipate the few sections where slow execution is required. The look ahead function's ability to anticipate these problem areas permits much faster execution through most of the path, dramatically increasing throughput. Because of the nature of the look ahead calculations – trajectory calculations are done well in advance of the actual move execution, and moves are kept within machine limits by the automatic adjustment of move speeds and times – they are not appropriate for some applications. Any application requiring quick reaction to external conditions should not use look ahead. In addition, any application requiring precise synchronization to external motion, such as those using PMAC's external time base feature should not use look ahead.

When the look ahead function is enabled, Turbo PMAC will scan ahead in the programmed trajectories, looking for potential violations of its position, velocity, and acceleration limits. If it sees a violation, it will then work backward through the pre-computed buffered trajectories, slowing down the parts of these trajectories necessary to keep the moves within limits. The calculations are completed before these sections of the trajectory are actually executed.

Turbo PMAC can perform these look ahead calculations on **LINEAR** and **CIRCLE** mode moves.

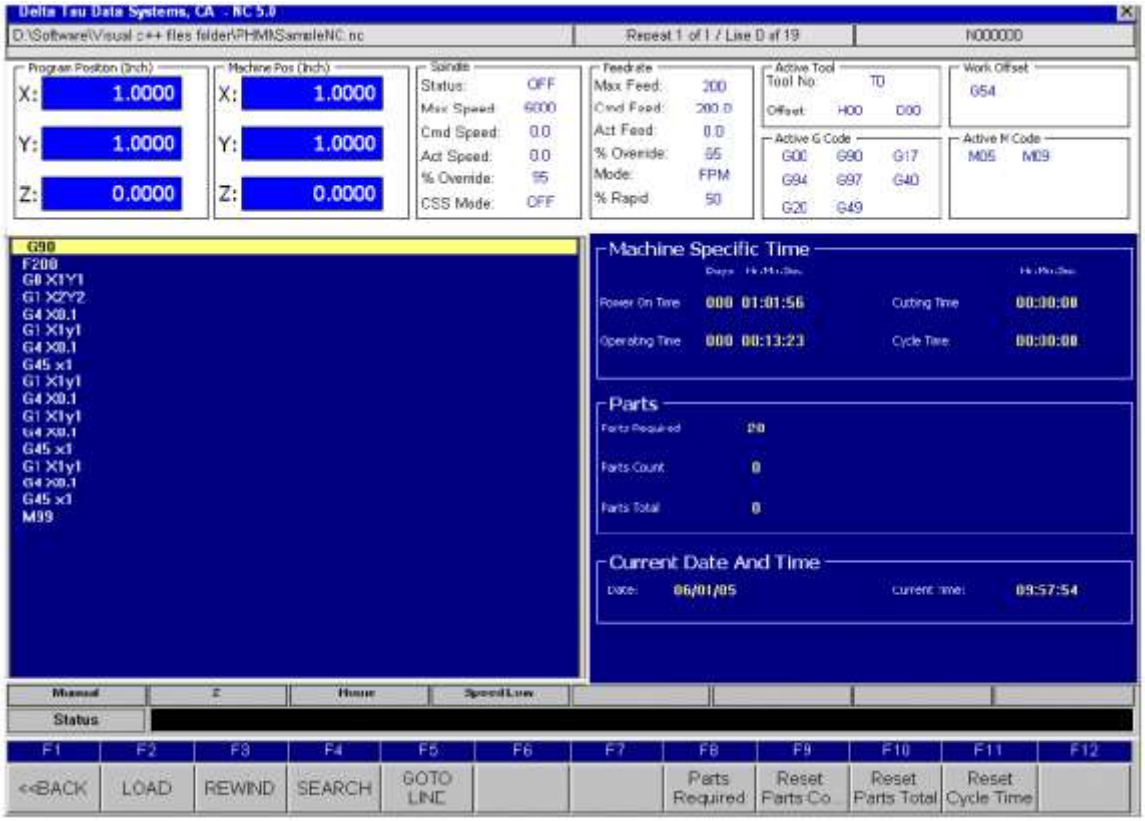
FUNCTION KEYS

F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12
POS	PROG	OFFSET	TOOLS	EDITOR	DIAG	MSGS	OPER				

PROGRAM OPERATIONS

F1 – PROG

This function key will display the sub menus available for part program functions.



F2 - Load

This button allows loading a program for part program execution. The loaded program will appear at the top of the screen in the loaded program title bar.

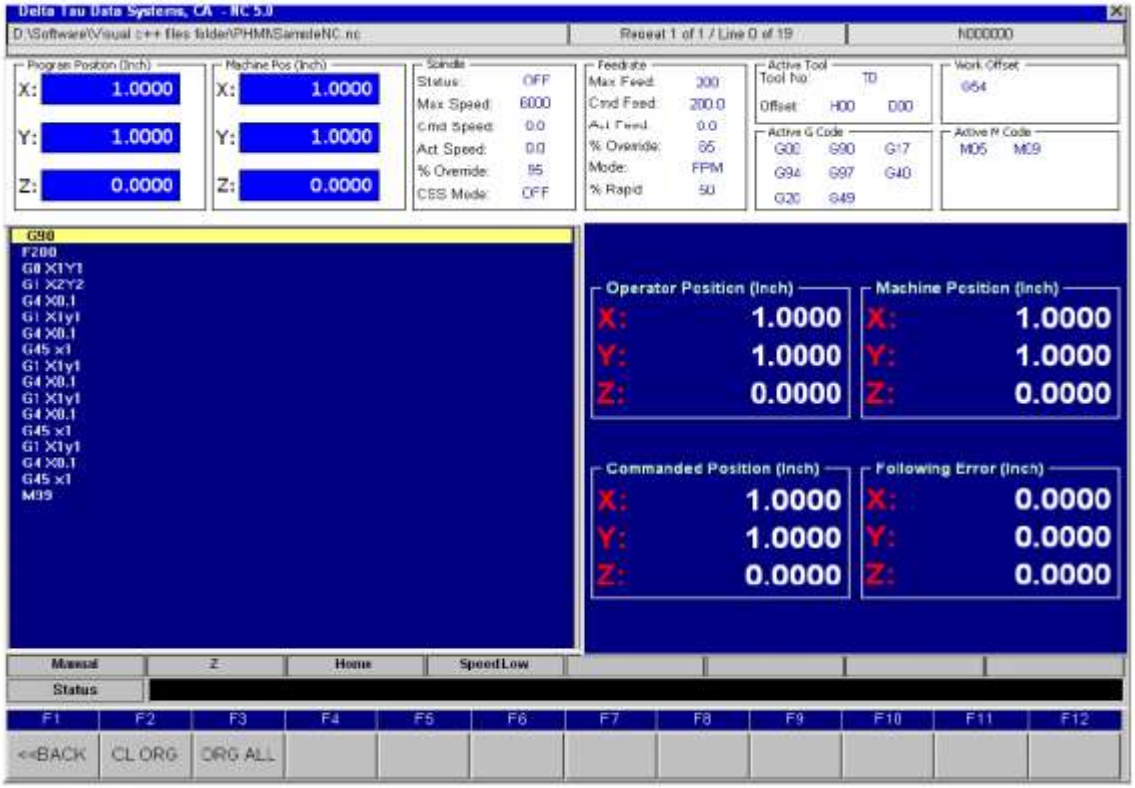
F4 - Search

If it is desired to start program execution at a location in the part program other than the beginning, a textual search can be performed to locate the start point of program execution. After the program start point has been set with a search command it is the operator's responsibility to insure that all miscellaneous M codes are manually set to the desired state.

POSITION DISPLAY OPERATIONS

F2 – POS (Position Sub Menu)

The Position sub menu displays additional position data useful for operator diagnostics. The Machine Position field displays the position of the machine with respect to where the machine has been zero referenced also referred to as homed. The machine position has no meaning until the machine has been homed. The Following Error Position field refers to the deviation of current actual position from the machine commanded position. The commanded position in manual mode refers to the instantaneous desired position, however in auto mode this register refers to the move destination position.



EDIT OPERATIONS

F1 – Editor

This function is used in loading or modifying a current or new part program. Advance features such as Search and Replace are available. Program editing is possible in Manual mode only. On this key, submenu keys are displayed. The Sub-key functions are as follows:

F2 – Edit Program

This function key will load current file from NC execution window to the Editor window. User can edit the current program. This is active only in **MANUAL** mode.

F3 – Load Program

This function key will load current file from the Editor window to NC execution window. This is active only in **MANUAL** mode.

F4 – SAVE

This function key will SAVE current open file from Editor Window. If the file name does not exist then it will open File Save dialog Box to Name the file and to save.

F5 – FIND

This function key will open dialog box in the Editor window to **find** string from current opened file.

F6 – REPLACE

This function key will open dialog box in the Editor window to **find and replace** a string from the current opened file.

F7 – CUT

This function key will cut the selected string from current opened file. The string can be selected by holding **Shift + Arrow** key. This is same as pressing the CNTL + X keys.

F8 – COPY
This function key will copy the selected string from current opened file. The string can be selected by holding **Shift + Arrow** key. This is same as pressing the CNTL + C keys.

F9 – PASTE
This function key will paste the cut or copied string to current opened file. This is same as pressing the CNTL + V keys.

F10 – UNDO
This function key will UNDO last activity. This is same as pressing the CNTL + U keys. (Equivalent of standard WINDOWS CNTL + Z function.)

F11 – REDO
This function key will REDO last activity.

CUTTER RADIUS COMPENSATION
Turbo PMAC provides the capability for performing cutter (tool) radius compensation on the moves it performs. This compensation can be performed among the X, Y, and Z axes, which should be physically perpendicular to each other. The compensation offsets the described path of motion perpendicular to the path by a programmed amount automatically, compensating for the size of the tool. This permits the user to program the path along the edge of the tool, letting Turbo PMAC calculate the tool-center path, based on a radius magnitude that can be specified independently of the program.

DISPLAY USER MESSAGES

There are four different types of user messages possible – Fatal Error, Error, Warning, and Message box. These messages are displayed in NC with different colors indicating different actions. The user messages are written in the ERRORS.DAT file.
Fatal Errors are displayed as, and can be edited under the [FATAL] section in ERRORS.DAT file.
Errors are displayed as, and can be edited under the [STOP] section in ERRORS.DAT file.
Warnings are displayed as WARNING, and can be edited under the [WARNING] section in ERRORS.DAT file.
Messages are displayed as, and can be edited under the [MESSAGE] section in ERRORS.DAT file.

CNC OPERATION AND PROGRAMMING

Program Context Display
The program context displays three fields relative to the file loaded for execution. These three fields are always present on the screen. The first field indicates the file to be executed. In MDI mode, the file to be executed is changed to the MDI buffer; otherwise the file loaded for execution is the file chosen by the operator. Upon initial installation in AUTO mode, the first field may contain the string NO BUFFER; this occurs because a file has never been loaded for AUTO mode execution. The second field displays the number of times a program has been repeated due to either a M99 code at the end of a main part program or the number of times a sub-program has been repeated due to a M98 L_ call. In addition, the current line of execution in the part program and the total number of lines in the part program is displayed. The third field displays the last executed N label of a part program.

Program Position

This area lists the current program position display. This display corresponds exactly to the positional data in the part program.

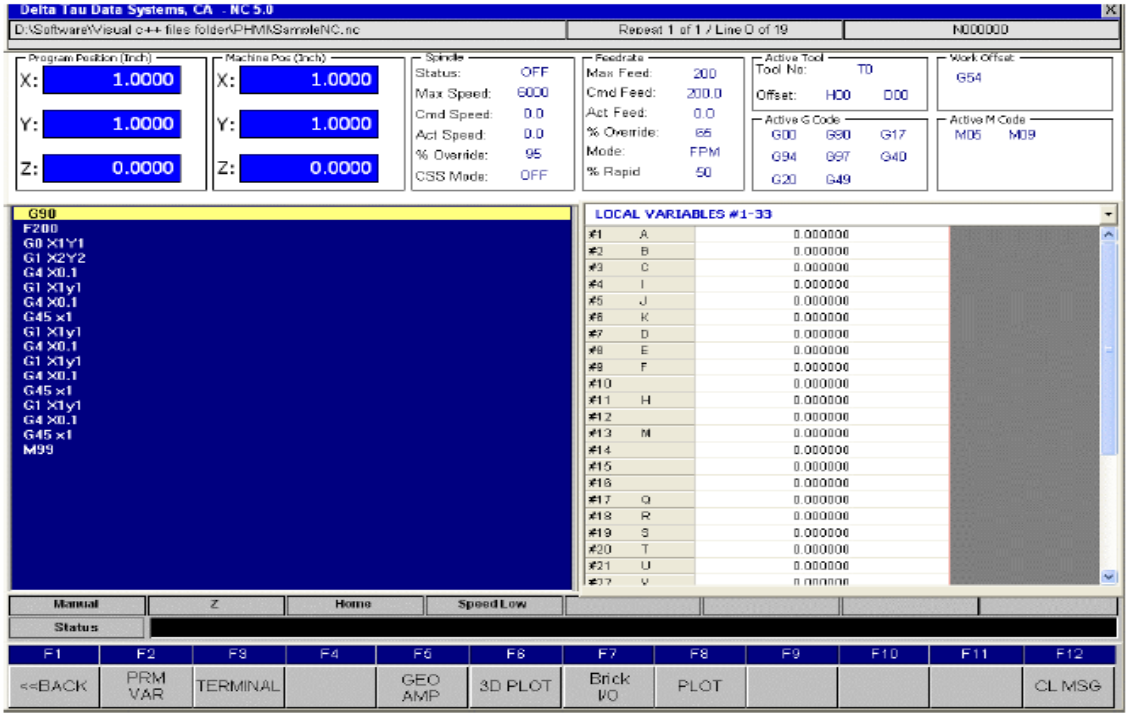
Machine Position/Distance to Go

This area displays either the current machine position or the position with respect to the zero reference return position (often referred to as home position). Machine position is displayed when the machine mode is manual. When the machine mode is auto or manual distance, to go is displayed. Distance to go indicates the amount of movement left in the current move.

DIAGNOSTIC OPERATIONS

F6 – DIAG

This function key displays the Diagnostic page. As a default on this menu parametric variable display, Terminal window, 3D Plot, and Plot functions are available. This addition is strictly the Machine Integrators responsibility using the **HMI NC Development system.** The user can specify diagnostics requirements specific to the machine.

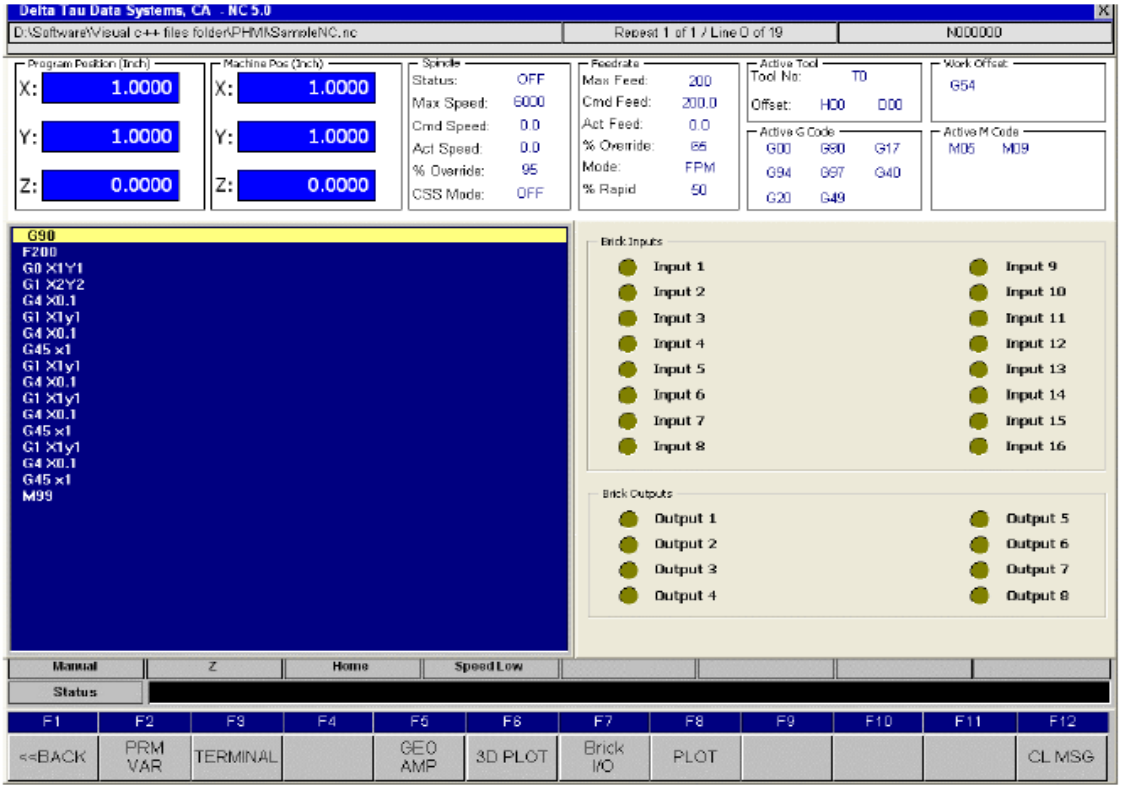


F6 – 3D PLOT

This function key will open real time 3D plotting. It will plot commanded position and following error for three axes, X, Y, and Z.

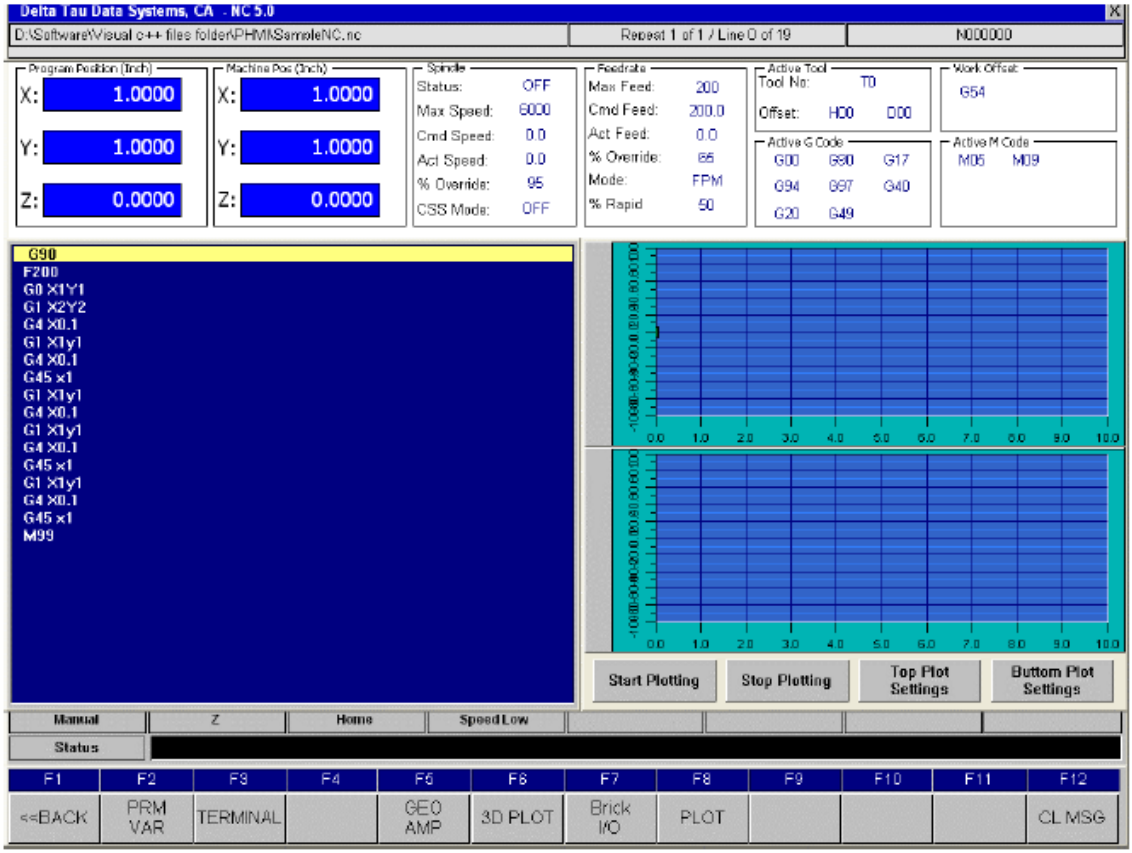
F7 –I/O Display

This function key will display status of input and outputs available on control.



F8 – PLOT

This function key displays the strip chart PLOT for the configured axis.



PROGRAMMERS GUIDE: G-CODES

The default G-codes delivered with CNC Pro2 is designed to emulate a Fanuc 10 style of G-codes. Hence a CNC program posted for a Fanuc 10 should work without any changes.

PMAC-NC Pro2 for Windows Machining Center G Code Library/Summary

G-Code	Function
G00	Rapid Traverse
G01	Linear Interpolation
G01.	1 Spline Interpolation
G02	Circular Interpolation, CW
G03	Circular Interpolation, CCW
G02 & G03	Helical Interpolation (X, Y, & Z in the G code command line)
G04	Dwell
G09	Exact Stop Check
G10	Program Data Input
G17	XY Plane Selection
G18	ZX Plane Selection
G19	YZ Plane Selection
G20	Inch Mode
G21	Metric Mode
G27	Reference Point Return Check
G28	Return To Reference Point
G29	Return from Reference Point
G30	2nd Reference Point Return
G31	Move until Trigger
G40	Cutter Compensation Cancel
G41	Cutter Compensation Left
G42	Cutter Compensation Right
G43	Tool Length Compensation, + Direction
G44	Tool Length Compensation, - Direction
G45	Tool Offset Increase
G46	Tool Offset Decrease
G47	Tool Offset Double Increase
G48	Tool Offset Double Decrease
G49	Tool Length Compensation Cancel
G50	Scaling Cancel
G51	Scaling
G52	Local Coordinate System Setting
G53	Machine Coordinate System Setting
G54	Work Coordinate System 1
G55	Work Coordinate System 2
G56	Work Coordinate System 3
G57	Work Coordinate System 4
G58	Work Coordinate System 5
G59	Work Coordinate System 6
G90	Absolute Command Mode
G91	Incremental Command Mode
G90.1	Arc Radius Abs/Inc Mode
G91.1	Arc Radius Abs/Inc Mode
G92	Absolute Zero Point Programming
G92.1	Absolute Zero Point Programming Cancel
G93	Inverse Time Feed
G94	Feed per Minute
G95	Feed per Revolution
G98	Return To Initial Point in Canned Cycle
G99	Return to R Point in Canned Cycle

Expressions:

The evaluation of an expression is how data is created and how decisions are made in a parametric program.

Symbol	Meaning	*Precedence*
EQ	Equal (cond.)	1
NE	Not equal to (cond.)	1
GT	Greater than (cond.)	1
GE	Greater than or equal to (cond.)	1
LT	Less than (cond.)	1
LE	Less than or equal to (cond.)	1
+	Binary Addition	2
-	Binary Subtraction	2
OR	Bitwise Logical or	2
XOR	Bitwise Exclusive or	2
*	Multiplication	3
/	Division	3
AND	Bitwise Logical product	3
MOD	Remainder	3
+	Unary +	6
-	Unary -	6
POPEN	Peripheral I/O device open	7
PCLOS	Peripheral I/O device close	7
DPRNT	Print to Device	7

Trigonometric functions

#[	Indirect operation	7
ABS	Absolute value	7
ACOS	Arccosine	7
ASIN	Arcsine	7
ATAN	Arctangent	7
COS	Cosine	7
EXP	Exponential	7
FIX	Truncation (floor)	7
FUP	Round up (ceiling)	7
LN	Log (natural, base e)	7
ROUND	Round off	7
SIN	Sine	7
SQRT	Square root	7
TAN	Tangent	7

Program Control:

Parametric programming allows additional control of program processing. The following constructs, when combined, provide the NC programmer with complete flexibility and control of the program:
- Branching GOTO
- Conditional block execution IF
- Iteration WHILE.