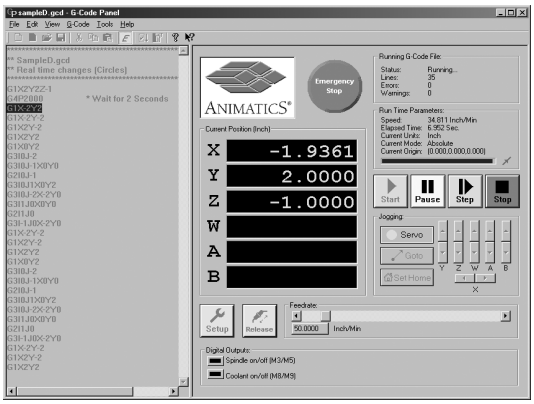
SMNC



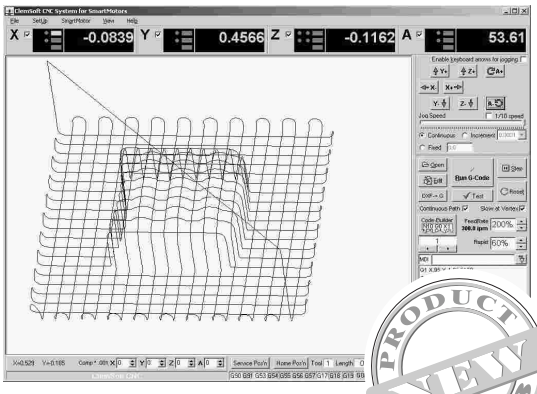
SMNC features a main

appearance to a traditional CNC system.

features comparable to a CNC system, SMNC includes:

- Linear and Circular motion control of multiple axes
- Configure Motors across multiple Serial ports
- Ability to set up on axis to mimic or copy another for gantry systems
- Smooth control of accel/decel for sensitive curvilinear motion
- Large numerical display
- DXF input available
- 6 axis of control (inludes axes mimic and spindle)
- RS-232 and/or RS-485 communication
- User definable M-Codes (for Digital output)
- Each axis is configurable to match your hardware
- Side-screen displays your source code as it is executed

JEN CNC



With a Graphical Interface similar to other CNC systems, JEN software also includes:

- Full Off-line simulation
- True 4 axis interpolation on a single line of G-Code
- Conversational/Interactive G-code builder
- Advanced Machine Settings screen for added versatility
- Graphic DXF input, user selects entities in desired order of machining
- 2D and 3D Real-Time path displays
- Jog and Record positions and then export as DXF file format
- Supports Windows compatible joystick input
- Supports on-screen M-Code definable button
- Supports on-screen point and click real-time moves (G0, or G1)
- Fully configurable I/O control
- Automatically updates downloaded programs on motor change-out
- DXF import option includes automatic Z axis up/down feature

G and M codes supported by either SMNC or JEN

Codes	*Description*	*SMNC*	*JEN*
G00	Rapid Motion	●	●
G01	Linear Motion	●	●
G02	Clockwise Circular Arc	●	●
G03	Counterclockwise Circular Arc	●	●
G04	Dwell	●	●
G10	Coordinate System Reset		●
G12	Rotary Axis Rewind		●
G17	X-Y Plane Selection	●	●
G18	Z-X Plane Selection	●	●
G19	Y-Z Plane Selection	●	●
G20	Change units to Inch	●	●
G21	Change units to Millimeter	●	●
G28	Return to 1st reference point	●	●
G30	Return to 2nd reference point	●	
G12	Z Axis Tool height probe		●
G40, 41, 42	Cutter Compensation	●	
G43	Tool Length Compensation		●
G53	Coordinate system 0 (machine limits)		●
G54 .. G57	Coordinate system 1 .. 4	●	●
G58, G59	Coordinate system 5, 6	●	
G76	Repeat a section of the program		●
G90	Set Absolute coordinates	●	●
G91	Sets Relative coordinates	●	●
G92	Set Coordinate System Offset	●	●
F	Change feedrate	●	●
S	Change spindle speed	●	●
M0	Pause	●	
M01	Program Pause (wait for input)		●
M02	End Program	●	●
M03	Spindle On (CW)	●	●
M04	Spindle On (CCW)	●	●
M05	Spindle Off	●	●
M06	Tool Change		●
M08	Coolant On	●	●
M09	Coolant Off	●	●
M21	Continuous Path On		●
M210	Set Maximum Angle		●
M22	Continuous Path Off		●
M23	Slow at Vertex On		●
M24	Slow at Vertex Off		●
M30	Program End and Reset I/O	●	●
M50… M55	Turn on Specified Output		●
M60… M79	Turn off Specified Output		●
M95	Haas© Brand Dwell (same as G4)		●
M98	Go To a Subroutine (ex. M98 E1234)		●
M99	Return from Subroutine Call		●