



SMNC™, Animatics' G-Code based servo motion control software, uses numeric control to deliver multi-axis contouring for your Animatics SmartMotor™

applications. SMNC provides a set of features that are comparable to any CNC system, including a user interface that is similar in appearance to a traditional CNC system. Review the table in this section to see the G and M Codes that are supported by SMNC software.

SMNC standard features include:

- Linear and circular motion control of multiple axes
- Configures SmartMotors across multiple serial ports
- Converts CAD-DXF files into motion control G and M Codes
- Duplication of axis motion for gantry systems
- Smooth control of acceleration and deceleration for sensitive curvilinear motion
- Large numerical display
- 6 axis control, includes axes mimic and spindle
- RS232 and RS485 communication
- User-definable M-Codes for digital output
- Displays source code during execution
- Writes and edits any G-Code program in the source view, with added support for:
 - 3D linear movements
 - CW and CCW Circular movements
 - CW and CCW Helical Movements
 - Wait, Pause, definable M-Codes, Spindle Commands and more
- Defines up to 30 M-Code commands and views their status during execution
- Imports graphic files with DXF format and converts them to G-Code programs
- Exports G-Code programs to Coordinated Motion Files
- Checks and runs a G-Code program or Coordinated Motion file
- Checks features during G-Code program execution:
 - Feed Hold
 - Single step
 - Reset (End)
 - Emergency stop
- Jogs the device using the Jog Buttons, and moves it to any location using the Go to utility on the Panel view
- Controls the Spindle using related buttons on the Panel View. The Spindle can be a SmartMotor or any other type of motor controlled by M-Codes and digital outputs
- Sets any connected SmartMotor in Coordinated Motion Mode, Spindle Mode, Mimic Mode, Rotary/Vector Mode, or Uncoordinated Mode

New features:

- Define up to 30 Inputs to perform SMNC functions, such as Start, Reset, and Feed Hold, or as interlocks
- Define up to 10 different tools. The current tool can be changed in a G-Code program.
- Password protection to control user access for many features
- Define up to 6 different coordinate systems (G56-G59)
- Customizable homing methods

G and M codes supported by SMNC

Codes	Description
G0	Rapid Linear movement
G1	Normal Linear movement
G2	Clockwise circular movement
G3	Counter-Clockwise circular movement
G4	Wait
G17	Select the X-Y plane for circular movements
G18	Select the X-Z plane for circular movements
G19	Select the Y-Z plane for circular movements
G20	Change units to inch
G21	Change units to Millimeter
G28	Return to the 1st Reference point
G30	Return to the 2nd Reference point
G40	Cancel cutter compensation
G41	Start cutter compensation left
G42	Start cutter compensation right
G43	Start tool length compensation
G49	Cancel tool length compensation
G54	Use preset coordinate system 1
G55	Use preset coordinate system 2
G56	Use preset coordinate system 3
G57	Use preset coordinate system 4
G58	Use preset coordinate system 5
G59	Use preset coordinate system 6
G80	Cancel Modal Motion (Used with canned cycles)
G81	Canned cycle: drilling
G82	Canned cycle: drilling with dwell
G83	Canned cycle: peck drilling
G85	Canned cycle: boring, no dwell, feed out
G89	Canned cycle: boring dwell feed out
G90	Change coordinate system to absolute
G91	Change coordinate system to incremental
G92	Change the logical origin
G98	Initial level return mode in Canned cycle
G99	Retract-point level return mode in Canned cycle
G101	Move the rotary axis
D	Change the tool index for cutter compensation (G40, G41, G42)
F	Change the Feedrate (Normal Speed)
H	Change the tool index for tool length compensation (G43, G49)
S	Change the Spindle Speed
T	Current tool index (M6)
M0	Pause
M1	Optional Stop
M2	End of program
M3	Turn on Spindle Clockwise
M4	Turn on Spindle Counterclockwise
M5	Turn off Spindle
M6	Change current Tool
M8	Turn on the Coolant
M9	Turn off the Coolant
M30	End of program and force turning off all of digital outputs
M99	End the program and restart it



Animatics' JenCNC is a full-featured 3D CNC software package designed exclusively to run Animatics SmartMotors.

JenCNC combines features of both a CAD/CAM and a motion-control software package into a unique graphical user interface for controlling two to four SmartMotors™ in true 3D coordinated motion. With real time 2D and 3D plotting to the screen, DXF-to-G-Code conversion, and conversational G-Code building, your CNC machine will be up and running in no time. JenCNC utilizes a simple serial interface to communicate and control SmartMotors. Using custom algorithms, motion is optimized for smooth control and continuous operation for all your machining needs. Due to its ability to run in a constant vector velocity, regardless of changes in direction, the software is ideal where dispensing or flow rate of glue or adhesives is critical to the process.

JenCNC is built on many years of in-field testing and customer feedback in real-world applications, including:

- Routers (gantry machining of aluminum, foam, vinyl, and wood)
- Hot-wire EPS foam cutting
- Plasma (oxy-fuel) cutting with torch height control
- Machining forms for vacuum-form molding of plastics
- Milling machine retrofits
- Sign making
- Engraving
- CNC drilling
- Gasket cutting
- Adhesive applicators

We strive to continuously improve its capabilities according to the needs and demands of our customers in order to provide the best possible solution for any need that may arise. Custom features can be added at a competitive rate. Please contact Animatics at 408.748.8721 for more information.

DXF to G-code converter:

JenCNC's built-in DXF-to-G-Code converter allows you to open and view DXF files. Once in view, you can select entities in the order you wish to have the motors move. The converter includes a set-up for the Z-Axis. If you left click from one entity to the next, the Z-Axis motion G-Code is automatically generated.

All entities connected end-point to end-point will produce a continuous path until the need of the selection is reached. If the interpreter comes to a "Y" in the path, it will choose the path of least resistance (angular displacement).

There is an additional "Join" tolerance set-up. You can set the distance tolerance from one entity to the next in case the entities are not actually connected at their end-points.

If within that tolerance, the interpreter assumes connection and continues the path through the entities as one continuous motion. This can be very useful if the original DXF file was created free-hand, as part of an artist rendering where O-Snaps may not have been used.

JenCNC features

- Upon start-up, JenCNC automatically detects motors and does a system update if any Animatics SmartMotor™ was changed out. This allows you to place the shortcut in the start-up directory to allow automatic restart on loss of power
- A full machine settings window allows for customization to physical dimensions of the machining space
- Machine tolerance levels can be set to ensure that no product damage occurs in the event of motor drop-out or path divergence
- Slow-down proportional-to-angle can be tailored to minimize machining time while providing the best surface finish through sharp turns
- Customizable G-Codes for user-defined tooling positions
- Customizable M-Codes for I/O control and SmartMotor commands or subroutine calls
- Ability to call G-Code subroutines
- Ability to repeat a section of G-code any number of times
- Z-Axis (tool length) offsets
- SAE or metric scaling
- On-screen and keyboard real-time jogging
- Auto-detection of Windows-compatible joystick for jog control
- Menu-selectable Inputs set-up window
- Wait-on-input definable M-Codes
- Customizable outputs assignable to user-definable M-Codes
- CMM probing for setting tool offset
- Terminal screen diagnostics page for testing and troubleshooting
- User-selectable homing routines with configurable offsets
- Advanced settings screen includes events set-up for E-Stop conditions

JenCNC G-Codes Implementation

G-Codes not listed below are ignored by the interpreter.

JenCNC G-Codes:

- G00** Rapid Motion
- G01** Linear Motion
- G02** Clockwise Circular Arc
- G03** Counterclockwise Circular Arc
- G04** Dwell
- G10** Coordinate System Reset
- G12** Rewind the Rotary Axis
- G17** X-Y Plane Selection (This is the normal plane of operation)
- G18** Z-X Plane Selection
- G19** Y-Z Plane Selection
- G20** and **G70** Inch Mode
- G21** and **G71** Metric Mode
- G28** Go to Tooling Position
- G37** Probing
- G41** Left side Tool Compensation
- G42** Right side Tool Compensation
- G43** Tool Length Compensation
- G53 - G59, G110.- G120** Coordinate offsets
- G76** Repeat a section of the program
- G80** Clear any **G8x** modal mode
- G81, G82, G83, G85, G86** Canned Drill cycle Routines
- G90** Absolute Mode
- G91** Relative Mode
- G92** Set Coordinate System Offset

JenCNC M-Codes

- M01** Program Pause
- M02** End Program (See **M30** below)
- M03, M04** Turns on User Defined Input
- M05** turns off **M03** selected Output
- M06** Tool Change
- M08** User Defined
- M09** Turns **M08** OFF
- M14** Turn ON any single or multiple outputs
- M15** Turn OFF any single or multiple outputs
- M20** Send Torch to the Home Value position
- M21** Continuous Path
- M210** Set Maximum Angle Amount
- M22** Continuous Path Off
- M23** Slow at Vertex On
- M24** Slow at Vertex Off
- M25** Px.xxx Sets the pierce delay time with the P Word.
- M30 & M02** Program End and Reset
- M41, M42, M43** Turns on selected output as soon as motion starts
- M45, M46, M47** Turns off selected output(s) immediately
- M50 - M57** Waiting for Inputs from the real world
- M60 - M79** Outputs to the real world
- M80 - M85** User Programmable SmartMotor Command String Calls
- M86 - M91** User Programmable Sound File Player Functions
- M95** Dwell (exactly the same as a **G04**)
- M98** Go To a Subroutine
- M99** Return from a Subroutine

