SMARTMOTOR NUMERIC CONTROL™

QUICK TUTORIAL

SmartMotor Numeric Control™, (**SMNC™**), is a G-Code based Servo Motion Control system developed by Animatics Corporation that delivers multi-axis contouring without an expensive control panel and without a cabinet full of controls. **SMNC™** is built on Animatics' SmartMotor™ technology that compresses a Controller, Amplifier and Encoder into a long life Brushless Servo Motor. A power supply, three motors, a cable and a PC complete the system. You can write G-Code programs or import them from AutoCad® DXF files, and check, analyze, and run them. You can also use **SMNC™** to write SmartMotor™ Coordinated Motion files and run them. In a Coordinated Motion file the position coordinates and time values are organized in a table. Executing the G-Code program or Coordinated Motion File results in smooth and continuous, multi-axis contouring.

SECTION CONTENTS:

INSTALLATION

When in a windows environment, insert the **SMNC™** installation CD in your CD-Driver and follow the instructions during the installation process. If you are installing from floppy disks, insert the first disk and run the setup.exe program contained on the disk.

At the end of the installation, an icon will be added to your desktop and a group named SmartMotor Numeric Control will be added to the start menu.
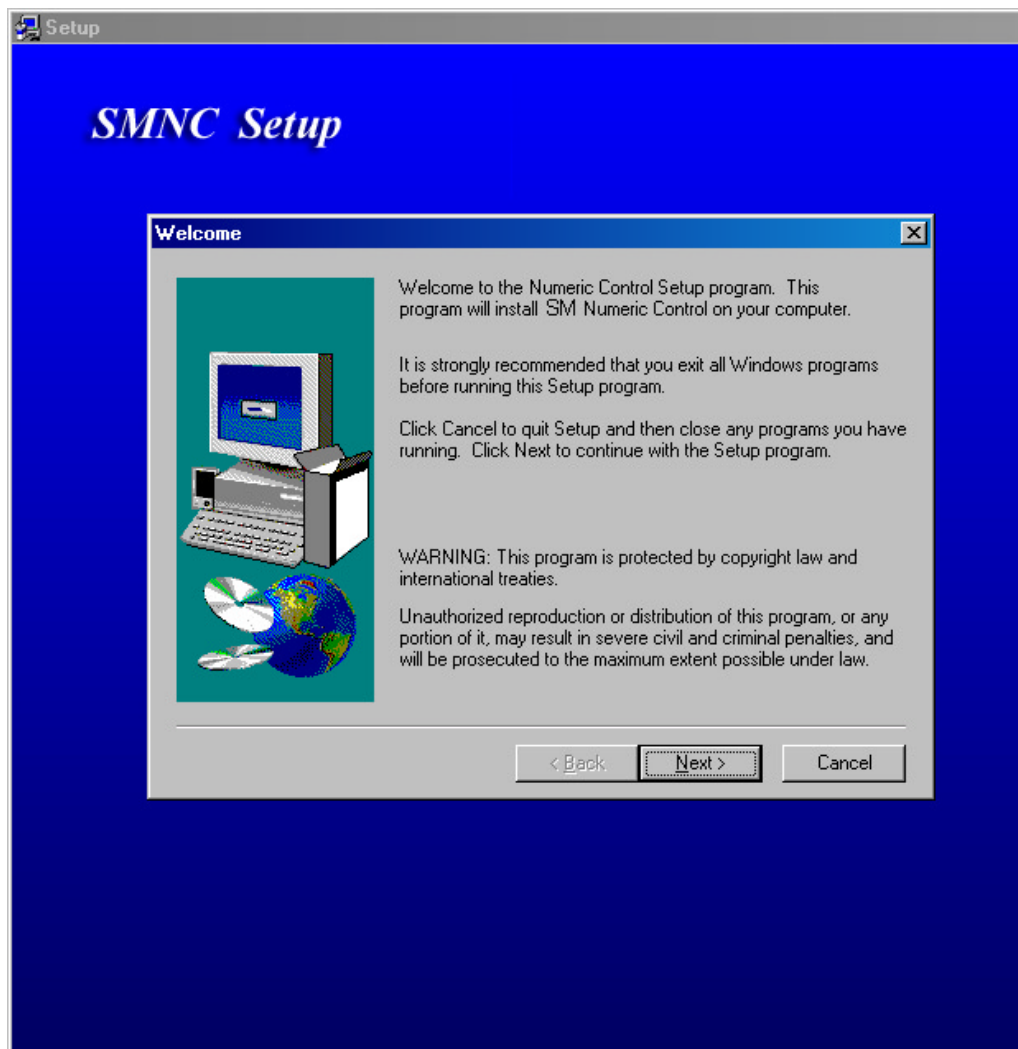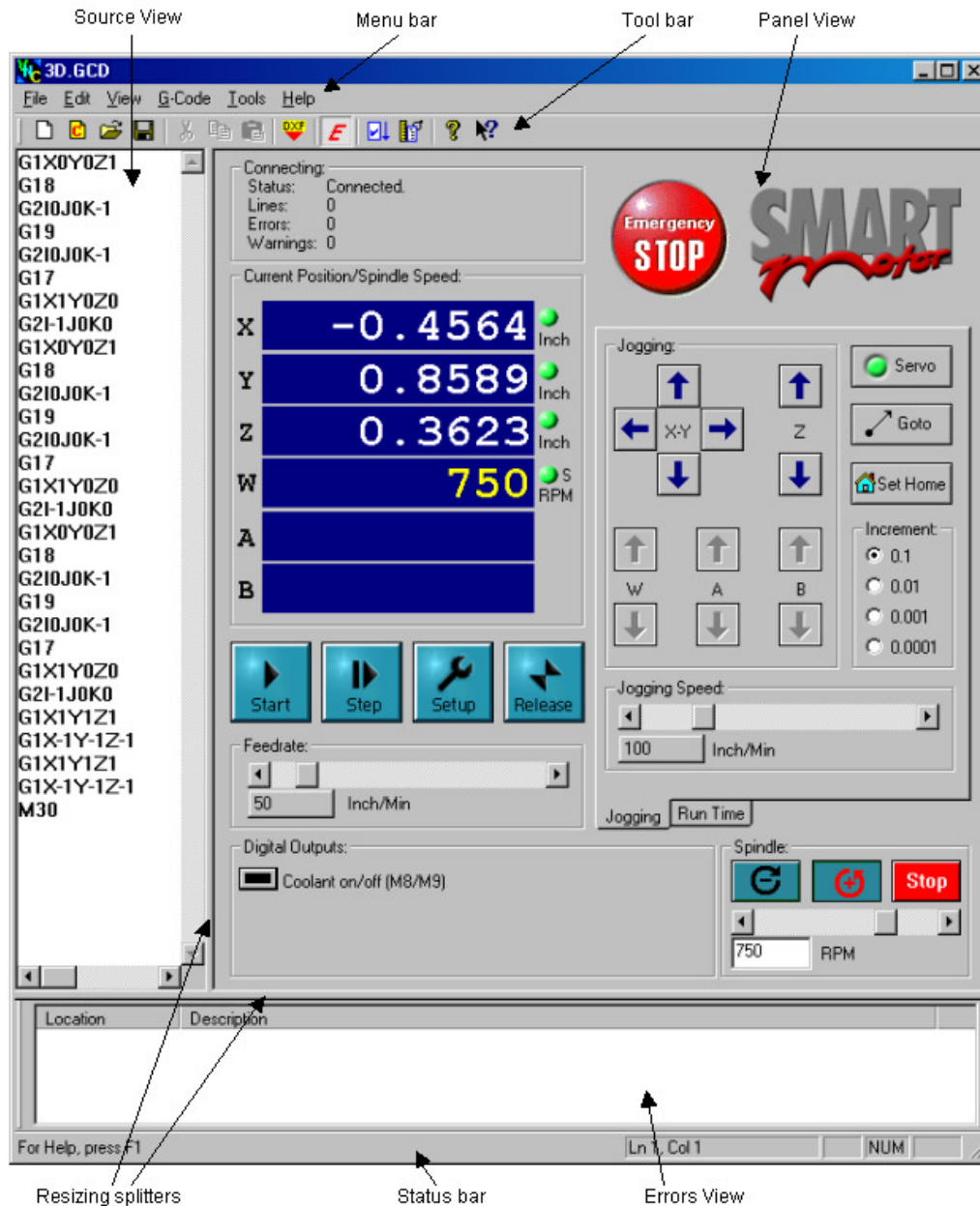

Figure 1: *SMNC™ installation program.*

RUNNING SMNC™

Double-click the **SMNC™** icon on your desktop to run it. Figure 2 shows the main window of **SMNC™**:

*The **SMNC™** screen is designed to emulate the appearance of a typical CNC terminal.*

Figure 2: *Main window components of **SMNC™**.*

CONNECTING THE MOTORS

CONNECTING THE MOTORS

Before you can use this program with motors, you must make the connections for power and serial communications. You may have from 1 to 6 motors, connected to your computer. The motors can be connected to the computer using an RS-232 Daisy Chain, an RS-232 Star, or an RS-485 Bus cable. Figure 3 shows some different communication configurations for motors.
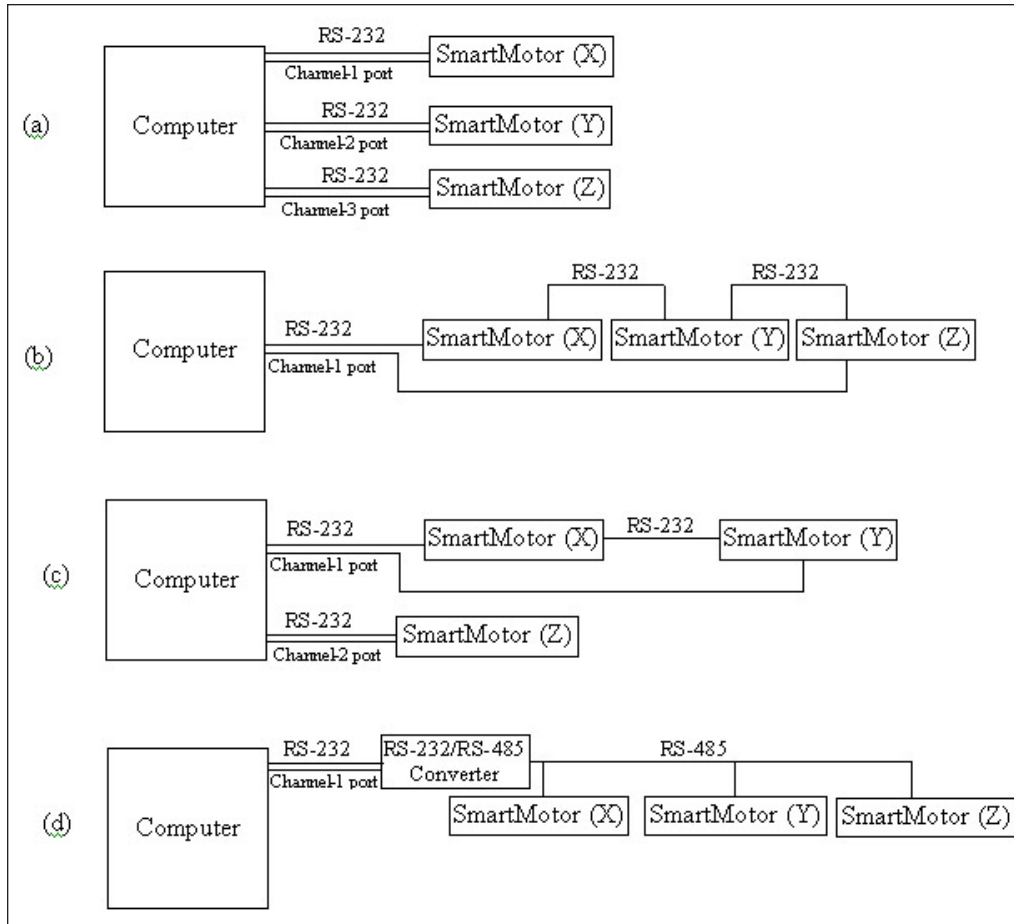

Figure 3: *Different Communication configurations:*

a) RS-232 Star configuration
b) RS-232 Daisy Chain configuration
c) A combination of RS-232 Star/Daisy Chain configuration
d) RS-485 Bus configuration

CONNECTING THE MOTORS
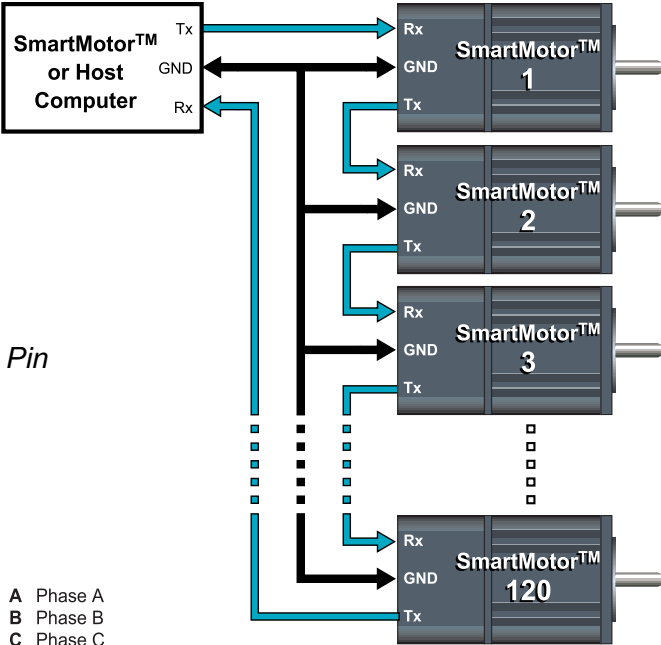
Figure 4: *Typical RS-232 Daisy Chain.*

Figure 5: *Motor Connector Pin Identifications.*

Different SmartMotor versions are available with a variety of different connector arrangements. The "D" type connectors are recommended for new designs.

4 Pin, 3 Phase Power

A   Phase A
B   Phase B
C   Phase C
D   Chassia

24 Pin Data I/O

A	RS-232 Ground	N	Ext Encoder A/Step/User I/O A
B	RS-485 Ground	P	Ext Encoder B/Direct/User I/O B
C	Ground	Q	Reserved
D	Ground	R	User I/O G
E	RS-232 Transmit	S	Left Limit/User I/O D
F	RS-232 Receive	T	Right Limit/User I/O C
G	RS-485 A	U	AniLink Data/User I/O E
H	RS-485 B	V	AniLink Clock/User I/O F
J	+5V Out, Fused	W	Reserved
K	Encoder A Out	X	Reserved
L	Encoder B Out	Y	Reserved
M	Encoder I Out	Z	Reserved

7 Pin Combo D-Sub Power and I/O

		1	Sync or I/O G
		2	+5V Out
		3	RS-232 Transmit
A1	+20V to +48V DC	4	RS-232 Receive
A2	Power Ground	5	RS-232 Ground

15 Pin D-Sub I/O
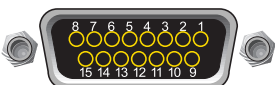
1	I/O A	9	Encoder B Out
2	I/O B	10	SM RS-232 Transmit
3	I/O C	11	SM RS-232 Receive
4	I/O D	12	+5V Out
5	I/O E	13	Ground
6	I/O F	14	Power Ground
7	I/O G	15	Power
8	Encoder A Out		

Encoder I/O

7   Ground
6   +5V Out
5   Encoder Index Out
4   Encoder B Out
3   Encoder A Out
2   Ext Encoder B/Step/User I/O B
1   Ext Encoder A/Direction/User I/O A

AniLink I/O

4   Clock/User I/O F
3   Data/User I/O E
2   Ground
1   +5V

Left/Right Limit I/O

3   Limit Input
2   Ground
1   +5V Out

Figure 6: *Motor Connector Arrangements.*
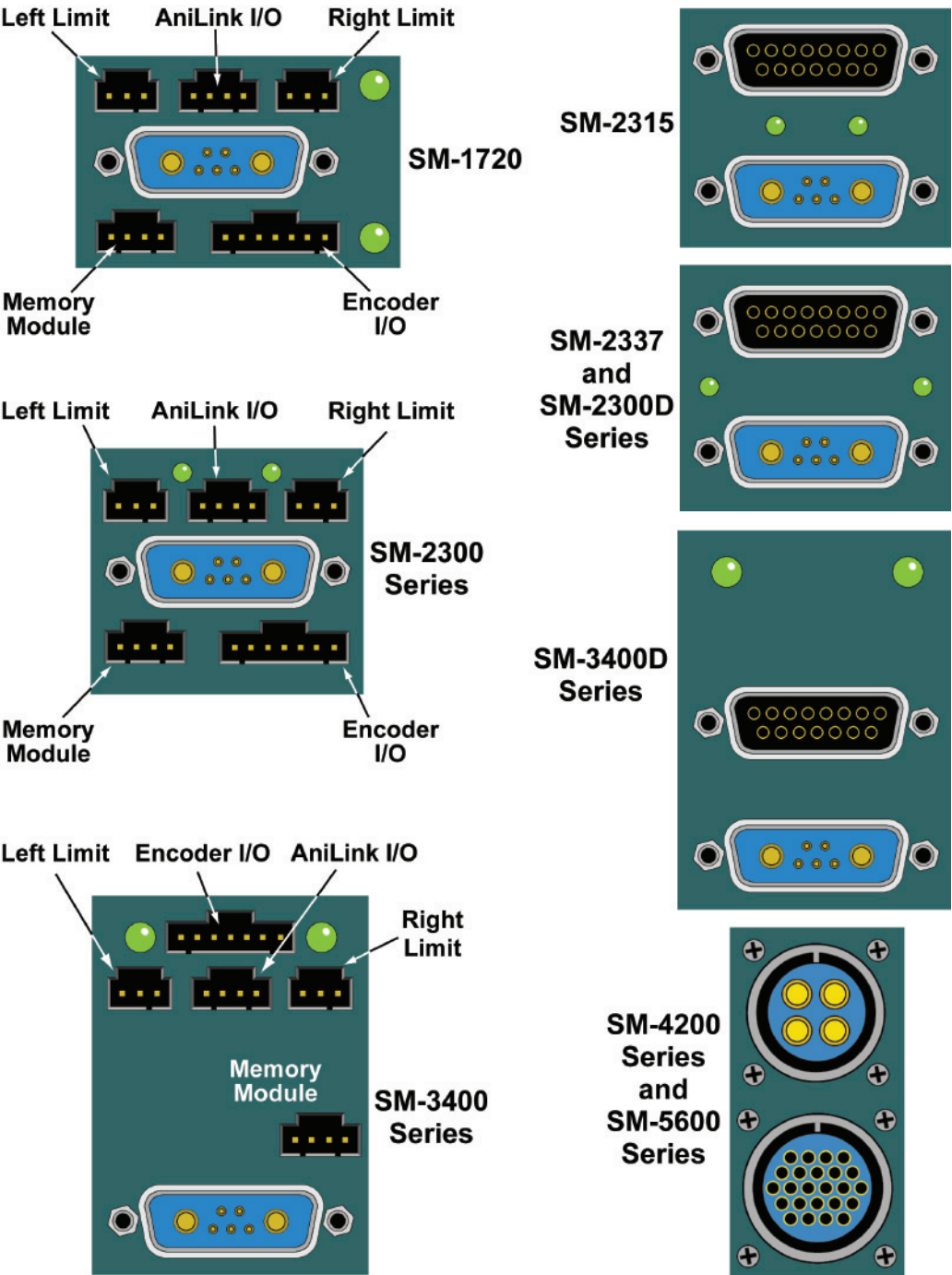
Standard Opto-Modules readily convert the TTL I/O of the SmartMotor into Industrial 24V power for interfacing to external devices

Typical DC Input Module (-IDC5 part numbers)

Typical AC Input Module (-IAC5 part numbers)

Typical DC Output Module (-ODC5 part numbers)

Typical AC Output Module (-OAC5 part numbers)

10
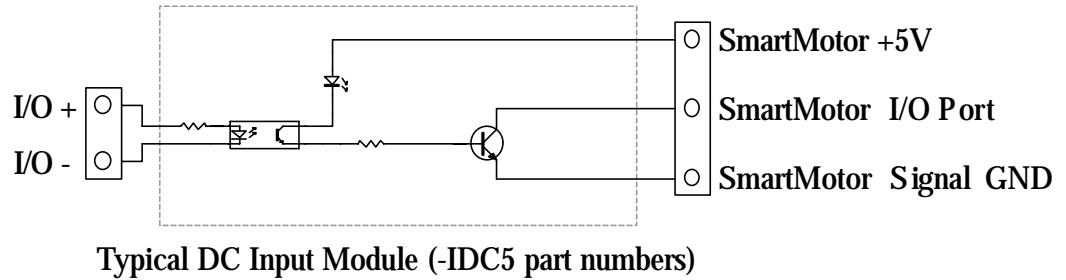
COMMUNICATION SETUP

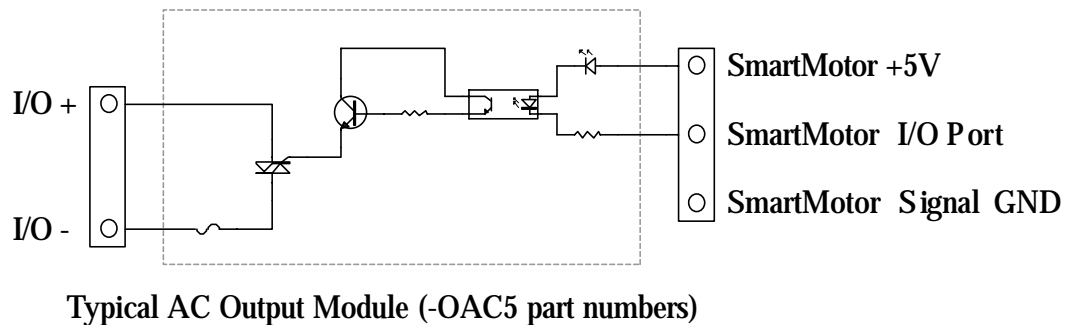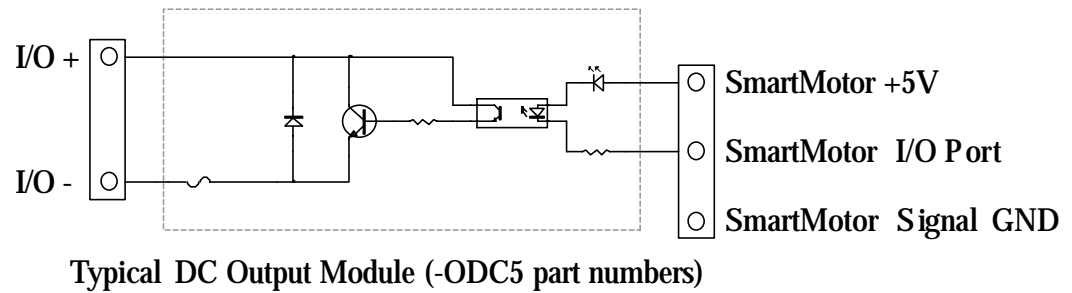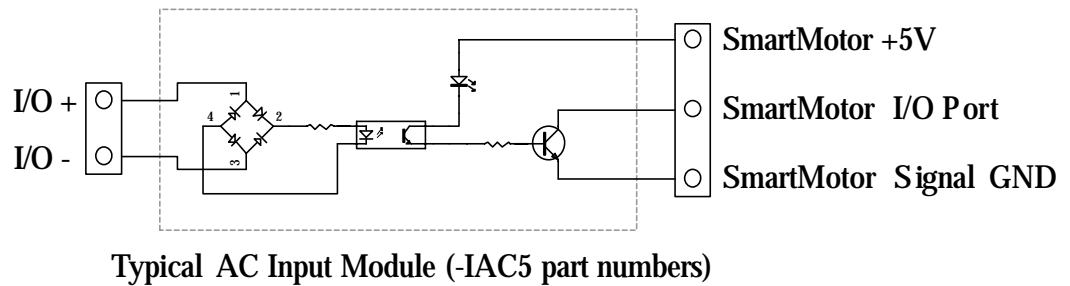On the menu bar, select the **Setup** sub-item from the *Tools* menu. The *Setup* property sheet will appear. Select the *Channels* page from the list at the left side of property sheet. (Figure 8)
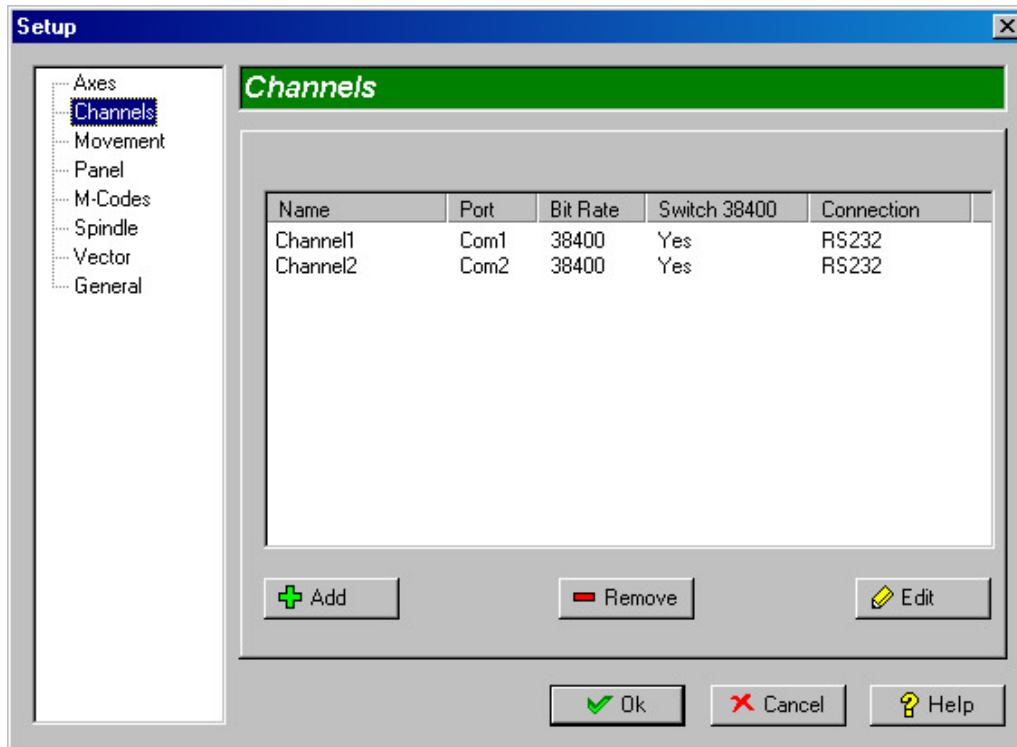
Figure 8: *Channels page of Setup property sheet.*

By default there are always two channels defined on this page. You can define more channels depending on your communication configuration. Click on the *Add* button to define another channel. The *Add new Channel* dialog box will appear. (Figure 9)

Enter the channel name, serial port and bit rate and leave other fields with their default settings. Click on the *OK* button and you will see the new channel is added to the list in the *Channels* page of *Setup* property sheet.

Figure 9: *Add new Channel dialog box*

COMMUNICATION SETUP

Figure 10: *Axes page of Setup property sheet.*

It may be necessary to further "tune" your SmartMotors if the motion is sloppy or unstable. To tune the motors press the Tuning Button. Generally speaking, if the motors buzz you need to lower KP. If the motors appear too soft or compliant, you need to raise KP. Raise KD to dampen the motion and lower KD if the motors make a grinding sound.

Now select the *Axes* page in the page list at the left to show the *Axes* page of *Setup* property sheet. (Figure 10)

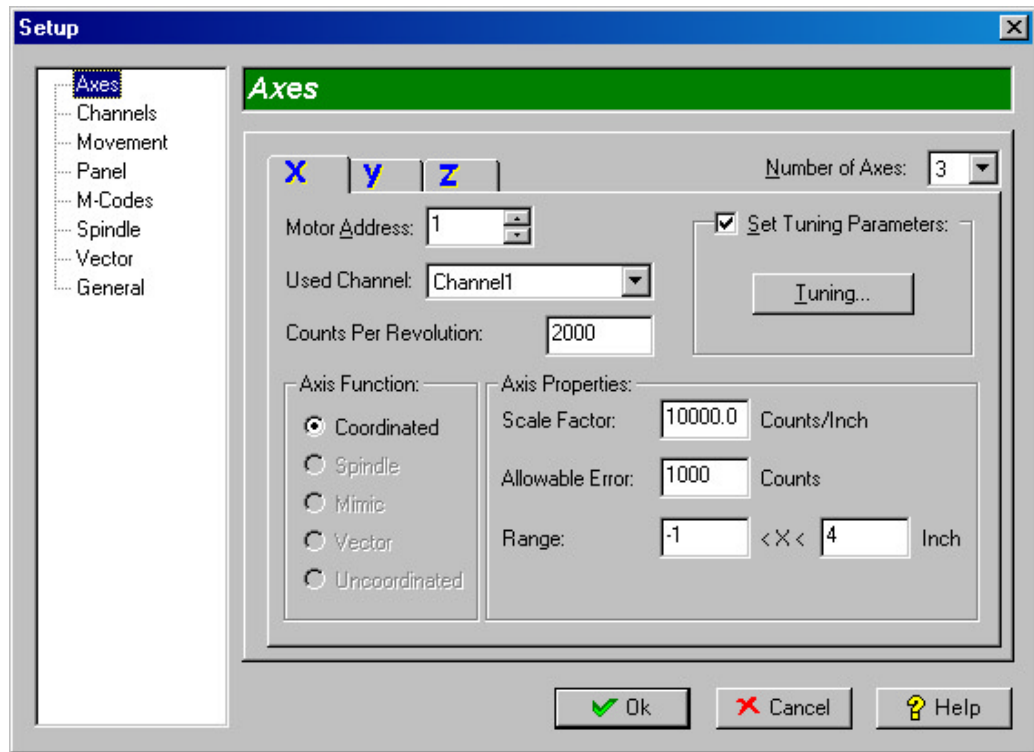Select the number of motors in the *Number of Axes* combo box. In this example we select 3 motors to setup the configuration shown in part (c) of figure 3. Therefore we will have 3 Axes, namely X-axis, Y-axis, and Z-Axis. Assuming all of the motors are connected to one port on the computer, do the following:

In the X-tab, enter 1 for *Motor Address*, and select *Channel1* for the *Used Channel*.

In the Y-tab, enter 2 for *Motor Address*, and select *Channel1* for *Used Channel*.

And in the Z-tab, enter 3 for *Motor Address* and select *Channel1* for *Used Channel*.

Leave other fields unchanged with their default settings. Click on the *OK* button of the property sheet.

Connect the motors as you configured and setup in previous sections and turn on the power. Click on the *Connect* button on the *Panel View* or select the *Connect* sub-item from the *Tools* menu. The program opens the serial port/s and both addresses and initializes the motors. After a few seconds, the current position of the device will be shown in the *Panel View*. The *Connect* button's face will change to *Release* and the jog buttons will be enabled.

JOGGING

Jogging Buttons:

Use these buttons to manually navigate along the axes. When you press a button, it moves along its Axis using the speed, acceleration, and step specified in the *Panel Page* of the *Setup* property sheet and this *Jogging Page*. You can use these Jogging buttons in different ways. (Figure 11)

1. Click on any of these buttons to move along the axis in a positive or negative direction. Each time you click on the button, the motor will move one step using the current increment value selected in the *Increment* group box. You can define the increment values in the *Panel* page of the *Setup* property sheet.

Figure 11: *The Jogging page on the Panel View.*

2. Click and hold the left mouse button on any of these buttons. The axis moves continually until you release the button. This movement uses the speed and acceleration defined in the *Panel* page of the *Setup* property sheet and *Jogging Speed* Scroll bar.

3. If your mouse is equipped with a wheel, you can middle-click on any of jog buttons and enter the wheel mode for that axis. In this mode you can use the wheel of mouse to move along the selected axis. To exit the wheel mode you should middle-click again on the same jog button. You can use the previous two methods for other axes while in *wheel mode* for the selected axis. If you middle-click on another jog button, an axis currently in wheel-mode will exit wheel mode, and the new selected axis will enter wheel mode.

Servo button:

When the indicator on the Servo button is Green, the motors are servoing (holding in position). Try to move the motors manually (but only if it is safe to do so: ***NEVER PUT YOUR HANDS ANY PLACE WHERE A SERVO RUNAWAY COULD CAUSE YOU HARM***). You will see that you cannot move the device easily. Now press the *Servo* button to turn it off. The indicator will change color to Black indicating the off state of motors. If it is safe to do so, now try to move the motors manually. You will see that the device moves easily. When the motors are in the *Servo* state, they apply torque when needed to maintain their current position; so you cannot move them by hand. The *Servo* state automatically becomes on, when you change the current position using the *Jogging*

buttons. Servo state is also on (and the servo button disabled), while running G-Code and Coordinated Motion Files. You can change the servo state for each individual SmartMotor™ using the Button/Indicator near the position value of each axis.

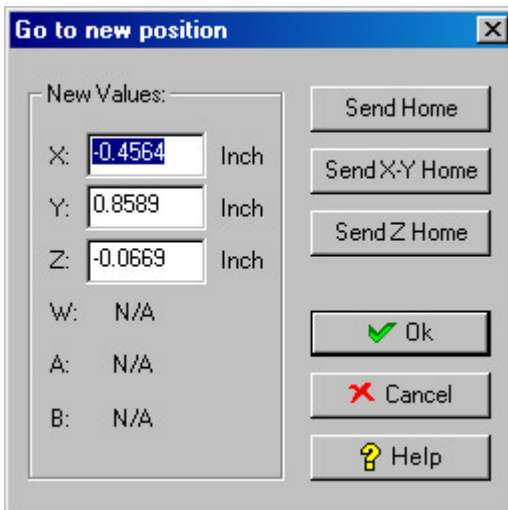**Figure 12:** *The Go to new position dialog box*

Goto Button:

Click on the *Goto* button on the *Panel View*. The Go to *new position* dialog box will appear. (Figure 12)

Enter a value of 1 for all axes and click on the Ok button. You will see that the device moves to point (1,1,1) and the current position boxes show the movement and freeze when the device reaches the destination point.

Click again on the *Goto* button and click the *Send X-Y Home* button. You will see that the device moves to the point (0,0,1). Try the *Send Home* and *Send Z Home* buttons in the *Go to new position* dialog box and see the results.

Set Home button:

Use the *Goto* or jog buttons to move to the coordinate (0.1,0.1,0.1). Now click on the *Set Home* button. The *Coordinate Transformation* dialog box will appear. (Figure 13)
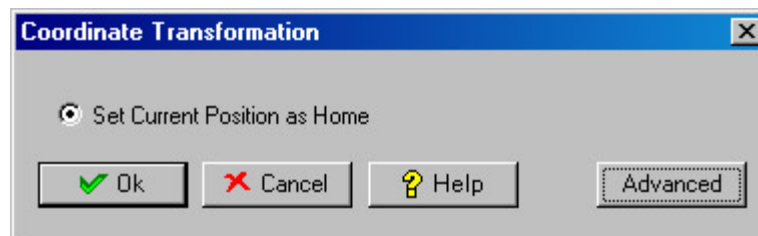
Figure 13: *Coordinate Transformation dialog box*

In this dialog box, you can set current position to home without movement. You can also use the advanced form of this dialog box and set current position for each axis to a value. Select the *OK* button. You will see the current position shown on the *Panel View* will change to (0,0,0), indicating that from now on, the current position is the home.

Increment Values:

You can change the increment for single-click jogging and wheel-mode jogging using these radio buttons. These four options are defined in the *Panel* page of the *Setup* property sheet.

Jogging Speed Scroll bar:

The Initial Jogging speed and acceleration are defined in the *Panel* page of the *Setup* property sheet, however you can change the jogging speed directly from *Panel View* using this scroll bar. The jogging speed value, as set by the scroll bar, is shown in a box below the scroll bar.

WRITING AND RUNNING A G-CODE PROGRAM

Click on the *Source View* to get the keyboard focus. Now type the following G-Code program in the *Source View* (exactly how G-Codes work will be described in the next section):

```
G1X0Y.5Z0
G2I.5J0
G1Z.5
G2I.5J0
G0Z0
X0Y0
```

From the *File* menu, select the *Save As* sub-item and in the *Save As* dialog box, type a name for the G-Code program and save it. Now press the *Start* button on the *Panel View*, or select the *Run* sub-item from the *G-Code* menu. **SMNC™** checks the program and then starts running. The device will go to the point (0,0.5,0) and draw a circle. Then it will go up to the point (0,0.5,0.5), draw another circle, return to the zero position and stop. Try running this program a few times. Note that when you start running a program, the *Setup* and *Connect/Release* buttons change into *Pause* and *Stop* buttons respectively. (Figure 14)

Now click the *Step* button. You will see that the device moves and stops. Click the *Step* button again. It will run a line and stop at the beginning of next G-Code statement on the next line. You can continue in Step mode using the *Step* button repeatedly or you may cancel Step mode by clicking the *Start* button.

Click on the *Start* button to run the program again and click the *Pause* button while it is running. You will see that the motors decelerate to a stop. You can cancel the pause mode and continue running the rest of program from this point by clicking again on the *Pause* button or simply clicking the *Start* button.

Figure 14: Running Buttons before and during running.

Click the *Start* button again to run the program. Try the *Stop* and *Emergency Stop* buttons when the program is being run. When you use the *Emergency Stop* the motors stop at once and the Servo turns off. The *Stop* button does the same, but the servo remains on. Note that this is the default setting for the Emergency stop and *Stop* buttons. You may change the default setting using the *General* page of the *Setup* property sheet.

You can use the Feedrate scroll bar on the Panel View to change the speed during the movement, Pause mode or Step mode. Please note that when you track the button of scroll bar, the new speed, shown in the box below the scroll

bar, will become effective only when you release the button.

Run the program and this time take a look at the Run time Parameters group box in the Run Time Page on the Panel View. (Figure 15). When you start a program, SMNC™ changes the page automatically so that you can see the Run Time Parameters. The speed is increased at the start of each segment and approaches the value selected by the Feedrate scroll bar.   It then decreases to zero at the end of segments in an amount proportional to the sharpness of the angle. If the motion segments are tangent, there is no slowing.

The Elapsed Time shows the time in seconds and increases as time passes.   The Current Units show that the values in the G-Code program on the current line are in Inch. The Current Mode shows that the current coordination mode at the running G-Code line is Absolute. The Current Origin shows that the current logical origin at the running G-Code statement is home.

Figure 15: *Run time parameters group box.*

The progress bar at the bottom of this group box shows the portion of the current G-Code line done relative to its total length.

UNDERSTANDING G-CODES
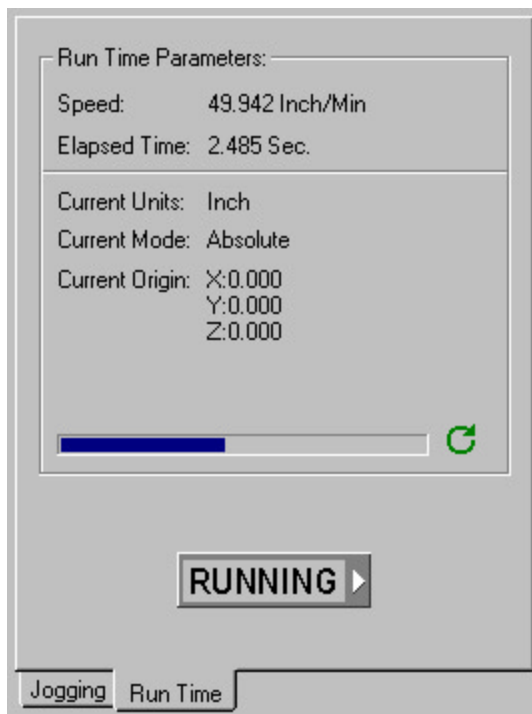
G-Codes are the most common language elements for rectangular axis motion, especially for the fabrication industry. They were invented when motion control electronics were very crude, and were therefore very simple. Fortunately, while motion control electronics are now very complex, G-Codes are still blissfully simple and very easy to learn. Basically, a single G-Code move is a single line of text. The text is comprised of alternating letters and numbers. The letter tells the control what to do and the number is sometimes part of the letter command and other times, the associated data. The text "G0" (that's a zero, not an o) means move straight and fast while "G1" means move straight and slowly. "G2" and "G3" refer to clockwise and counterclockwise circular motion. The codes X, Y, Z, W, A, and B are the ones that are followed by data. Each one of these letters corresponds to an axis of motion. Usually there are two or three axes in a system, X, Y or X, Y & Z. The number following each letter represents the coordinate each axis needs to reach. If two or more axes are mentioned on one line, then they will all move at the same time in such a way as to make a diagonal straight line.

For circular motion found in circles, arcs, and helices, two new parameters are needed to convey the radius value. The end position is described with the X, Y, Z, W, A, and B coordinates just the same as with a linear move, but with circles and arcs, the relative distances from the beginning position to the arc center are described with the letters I, J, and K. The actual radius of the arc is internally calculated based on this simple geometry. For an arc in the X-Y plane, only I and J are used. This is the default plane.

G-Codes are very efficient. Once you say G1, X1.5 or J2, you need not do it again until you want a change. If you have 10 moves, you will likely have 10 lines of G-Codes.

G2 is for clockwise arcs and G3 is for counterclockwise arcs. I, J, and K can be negative, and if no new X, Y, and Z coordinates are loaded, the end position will default to the start position and the motion will be a full circle.

M-Codes control non-motion features such as program flow, I/O and program termination. For example, M30 turns off all I/O and ends a program. M-Codes can be defined to turn On and Off the Digital Outputs in the motors.

Syntax Rules
- Single-line comments are allowed as long as you prefix the command with one of the following characters: / ' ; * (
- Tabs and spaces are ignored.
- Numbers are specified as integers or floating-point values in simple format.
- All commands must be in capital letters.
- The number of G commands per line is at most 2. If there are 2 G-Codes in a line the first one must be G17, G18, G19, G20, G21, G90, or G91.
- There can be zero or one M-Code, zero or one F-Command, and zero or one S-Command per line.
- M-Codes, S-Commands, and F-Command are processed (in the same order) before the G-Codes regardless of where they appear in the line.
- Line numbers are allowed only at the beginning of the line with letter N as their prefix.

UNDERSTANDING G-CODES

Implemented G-Code Commands:
G00   Rapid Linear movement
G01   Feed Linear movement (at Normal Feedrate)
G02   Clockwise circular movement
G03   Counter-Clockwise circular movement
G04   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
G90   Change coordinate system to absolute
G91   Change coordinate system to incremental
G92   Change the logical origin
F   Change the Feedrate (Normal Feedrate)
S   Change the Spindle Speed
P   Change G04 wait period in milliseconds

The following table shows the reserved M-Codes. You cannot delete or change a reserved M-Code, nor can you add an M-Code with these values (e.g. add another M0).

M00   Pause
M02   End of program
M30   End of program and force turning off all of digital outputs.
M3   Turn on the Spindle clockwise
M4   Turn on the Spindle counterclockwise
M5   Turn off the Spindle

The following M-Codes are defined by default. You may change them, or add/ remove M-Codes depending on your needs.

M8   Turn on the Coolant
M9   Turn off the Coolant

G0 Command

The "*G0*" command performs a rapid linear move using the *Rapid Acceleration* and *Rapid Feedrate* specified in the *Movement* page of the *Setup* property sheet. The move always ends with a complete stop before the next move, regardless of the setting for transition speed. This G-Code is typically used to position the axes to the starting point.

Example:
```
G0X1Y1          *Rapid move to 1,1
Z-0.25          *Rapid Z axis down
```

G1 Command

The "G1" command performs a linear move using the current normal acceleration and feedrate. *Normal Acceleration* and *Normal Feedrate* are defined in the *Movement* page of the *Setup* property sheet. *Normal Feedrate* can be changed using the *Feedrate* scroll bar in the *Panel View*. The settings for transition speed determine whether the motion stops between moves, slows down or just plows through to the next segment.

Example:
```
G1X1            *Move to 1,0,0 at default feed rate
Y1              *1,1,0
Z1              *1,1,1
X0Y0Z0          *0,0,0 Simultaneous 3 axis linear interpo-
lated           move
```

G2/G3 Command

These commands perform circular movements using the current normal acceleration and feedrate. *Normal Acceleration* and *Normal Feedrate* are defined in the *Movement* page of the *Setup* property sheet. *Normal Feedrate* can be changed using the *Feedrate* scroll bar in the *Panel View*. The settings for transition speed determine whether the motion stops between moves, slows down or just plows through to the next segment.

A circular or helical arc is specified using either G2 (clockwise arc) or G3 (counterclockwise arc). The axis of the circle or helix must be parallel to the X, Y, or Z-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane), or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

If a line of code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes.

Ellipses are not currently supported. If **SMNC™** encounters a G2/G3 command that represents an ellipse, it will generate an error.

21

Example:

```
G0X0Y0        * Go to 0,0
G2I1X1Y1      * clockwise quarter arc starting from 0,0
G18           * Set  XZ plane for circular movements
G1X0Y-2Z-1    * Go to position (0,-2,-1)
G2I0K1Y-1     * Draw a Helix with center of projected
              * circle at (0,0,0), Radius=1 and Height=1
```

G4 Command

The G4 command waits for a specified number of milliseconds. G4 without parameters will simply force a stop at a vertex. The number of milliseconds to wait is defined by the "P" Parameter.

Example:

```
G4P1000       *Wait for 1 second
```

G17/G18/G19 Command

These commands select the current plane for circular and helical movements. G17, G18 and G19 select the X-Y, X-Z and Y-Z planes respectively. Any change in the value of a coordinate that is not in the current plane, will not be accepted and will generate an error.

Example:

```
G1X1Y0Z0      * Go to (1,0,0)
G17           * Select X-Y Plane
G2I-1J0K0     * Draw Circle Center (0,0,0) R=1 in X-Y Plane
G1X0Y0Z1      * Go to (0,0,1)
G18           * Select X-Z Plane
G2I0J0K-1     * Draw Circle Center (0,0,0) R=1 in X-Z Plane
G19           * Select Y-Z Plane
G2I0J0K-1     * Draw Circle Center (0,0,0) R=1 in Y-Z Plane
```

G20/G21 Commands

These commands set the current length units to Inch and Millimeter respectively. The default units are specified in the Movement Page of the *Setup* Property Sheet. The default of the default is Inches. G20 and G21 codes can be mixed with the G0, G1, G2, and G3 codes within the same line.

Example:

```
G0X0
G21G1X10      *move 1cm along the X axis from 0,0
```

G90/G91 Commands

These commands set current coordinate system to Absolute and Incremental respectively. The coordinate system is always Absolute When a program starts. G90 and G91 can be issued before the G0, G1, G2, and G3 codes on the same line.

Example:

```
G1X1             * move to absolute (1,0)
G91G1X-1         * move back to absolute (0,0)
```

G92 Command

This command sets the logical origin for the rest of the G-Code program. Please note that to define the current position as the new logical origin, you must include all installed axes on the line.

Example 1:

```
G1X1             * move to (1,0)
G92X0            * set (1,0) to (0,0)
G1X-1            * move back to (-1,0) logical or (0,0)
absolute
```

Example 2:

```
G1X1Y-2Z3        * Move to (1,-2,3)
G92X0Y0          * set (1,-2,0) as new logical origin
G1X1Y1           * Move to (1,1,3) logical or (2,-1,3)
absolute
```

F Command

This command changes current *Normal Feedrate* for subsequent non-G0 moves. The "G20" and "G21" codes affect the units of the feedrate. The time units are always minutes. The current *Normal Acceleration* specified in the *Movement* page of the *Setup* property sheet is used to accelerate or decelerate to the speed specified by the "F" command. The program always starts with the speed adjusted on the panel by the *Feedrate* scroll bar. This code can be mixed with movement G-Codes and is processed before them.

Example:

```
G1X1F10          *move to 1,0 at a speed of 10 inch/min
                 (F command *is processed first)
```

S Command

This command changes the current speed of spindle. The number following the "S" letter indicates the new speed of spindle. The value shows the percentage of maximum spindle speed as set in the *Spindle* page of the *Setup* property sheet and the sign of the number specifies the direction of rotation. Negative numbers are used for clockwise rotations, and positive numbers are used for counterclockwise rotations. For example, S-100 turns the spindle on with maximum speed in the clockwise direction. An S0 command will turn the spindle off. You can mix M-Codes and S-Commands to control the Spindle. M3, M4, and M5 are equivalent M-codes for S-100, S100, and S0.

Example:
```
G0X0Y0        * move to home
S50           * counterclockwise, %50 of max speed
G1X1          * go to (1,0)
M5            * turn off the spindle
Y2            * go to (1,2)
S-100         * clockwise, max speed
```

M0 Command

This command forces a pause for all motors. When this command is reached, all motors decelerate to a stop using the current acceleration as defined in the *Movement* page of the *Setup* property sheet. Its function is exactly the same as the *Pause* command in the menu or *Panel View*. To continue running the program, the user must select the Run or *Pause* command from menu or click on the appropriate buttons on the *Panel View*.

Example:
```
G1X0Y0
M00           *Pause and wait for user command
G1X1
```

M2 Command

This command stops interpreting G-Codes. It causes the motors to stop and sets the SmartMotor™ positions to the last known valid values. It then resets all digital outputs and turns spindle and servo on/off depending on the settings in the *General* page of *Setup* property sheet. If a command is after the "M2" command, or in the same line, it never gets executed and **SMNC™** generates a warning when checking the G-Code program. It is not necessary to include the "M2" command at the end of the program.

Example:
```
M2            *end program
G1X1          *this line never gets executed
```

M30 Command

This command stops the interpretation of G-Codes, and, unlike the "M2" command, turns off all digital outputs, regardless of the settings in the *General* page of *Setup* property sheet. It stops the motors and sets the SmartMotor™ positions to the last known valid values. The spindle and servos are turned on/off, depending on the settings in the *General* page of *Setup* property sheet. Any command after the "M30" command, or on the same line, never gets executed and **SMNC™** generates a warning when checking the G-Code program. It is not necessary to include the "M30" command at the end of the program.

Example:
```
M30          *end program and turn off all digital outputs
G1X1         *this line never gets executed
```

M3 Command

This command turns the Spindle on clockwise. If a SmartMotor™ is being used as a Spindle, it will accelerate to the maximum speed using the values specified in the *Spindle* page of the *Setup* property sheet for Spindle speed and acceleration. When using a SmartMotor™ as a spindle, the "M3" command is equivalent to the "S-100" command. You can use the *Axis* page of *Setup* property sheet to define a SmartMotor™ to be in Spindle Mode.

If the Spindle is some other motor controlled by digital outputs originating from one of the linked SmartMotors™, the "M3" command turns on the related digital output as specified in the *Spindle* page of *Setup* property sheet.

Example:
```
G0X0Y0       * Go to origin
M3           * Turn on Spindle clockwise
```

M4 Command

This command turns the Spindle on counterclockwise. If a SmartMotor™ is being used as the Spindle, it will accelerate to the maximum speed using the values specified in the *Spindle* page of the *Setup* property sheet for Spindle speed and acceleration. When using a SmartMotor™ as a spindle, the "M4" command is equivalent to the "S100" command. You can use the *Axis* page of *Setup* property sheet to define a SmartMotor™ to be in Spindle Mode.

If the Spindle is some other motor controlled by digital outputs originating from one of the linked SmartMotors™, the "M4" command turns on the related digital output as specified in the *Spindle* page of *Setup* property sheet.

Example:
```
G0X0Y0       * Go to origin
M4           * Turn on Spindle counterclockwise
```

M5 Command

This command turns off the Spindle. If a SmartMotor™ is being used as the Spindle, it will decelerate to a stop using the acceleration value specified in the *Spindle* page of *Setup* property sheet.   When a SmartMotor™ is the spindle, the "M5" command does the same function as the "S0" command. You can use the *Axis* page of *Setup* property sheet to define a SmartMotor™ to be in the Spindle Mode.

If the Spindle is some other motor controlled by digital outputs originating from one of the linked SmartMotors™, the "M5" command turns off the digital outputs for the "M3" and "M4" commands specified in the *Spindle* page of *Setup* property sheet.

Example:
```
G0X0Y0        * Go to origin
M4            * Turn on Spindle counterclockwise
M5            * Turn off Spindle
```

WORKING WITH M-CODES

Click on the *Setup* button or select the *Setup* sub-item from the *Tools* menu. Select the *M-Codes* page in the *Setup* property sheet. (Figure 16)
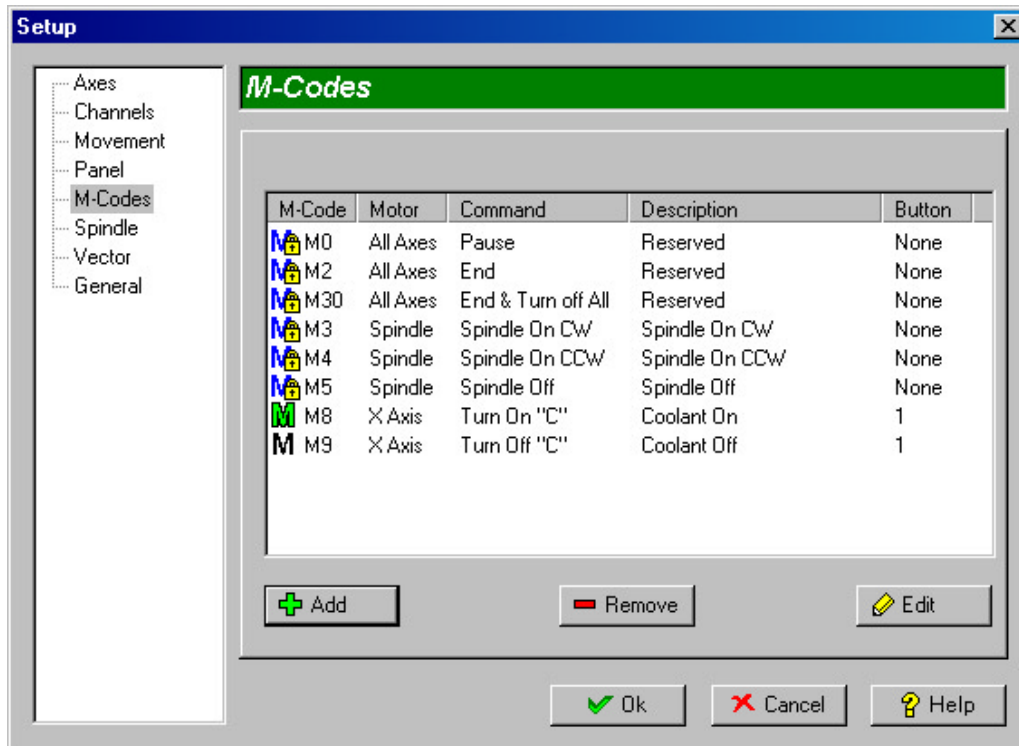

Figure 16: *The M-Codes page of Setup property sheet.*

There are eight M-Codes defined by default in **SMNC™**. The user can Add, Edit, or Remove M-Codes in this page. A Maximum of 30 M-Codes are allowed in **SMNC™**. Also, up to 6 buttons are available on the Panel View, which can be linked to the M-Codes. By default, one of these buttons is related to Coolant (M8/M9).

Double-click on one of the M-Codes in the list to view the associated properties. The M-Code properties dialog box will appear (Figure 17).
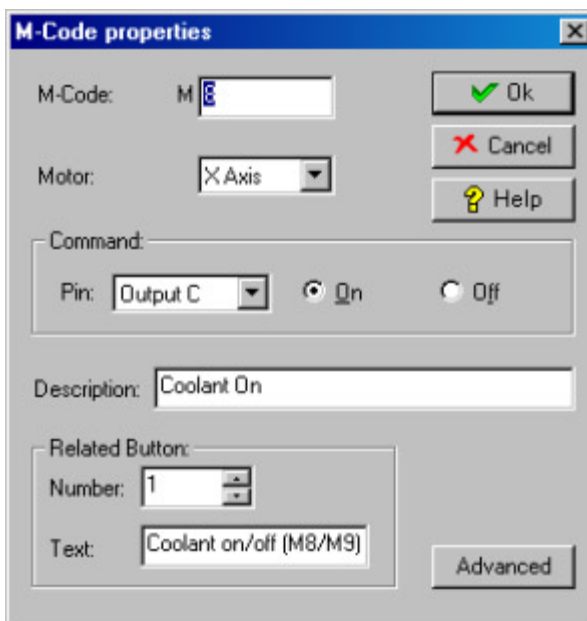

Figure 17: *The M-Code properties dialog box.*

27

Each M-Code controls one or all of the digital outputs in one or all of the motors. For example, M8, as defined by default, turns on the output 'C' on the SmartMotor™ corresponding to the X-Axis. For a two state On/Off command, you should define two different M-Codes. You can assign a descriptive text to each M-Code. If you want to reserve a button on the Panel View for the M-Code, you must enter the button number and some text to reside beside the button. A value of 0 for button number means that you don't want any button related to the M-Code.

Sometimes you may want to define an M-Code to do some specific action rather than just turn on or off an output. To define custom M-Codes, click on the Advanced button. The dialog box extends to show the advanced section (Figure 18).

You can now enter any SMI command in the SmartMotor™ Command edit box.

Click on the cancel button to close the M-Code properties dialog box and the setup property sheet. On the Panel View, and in the Digital Outputs group box, click on the button for Coolant on/off. You will see that the color of the indicator on the button changes to Green, indicating that it is in the "On" state. Please note that all SmartMotor™ digital outputs are

Figure 18: The M-Code properties dialog box - advanced.

considered Active Low. That means that in the "On" state, the output will go to 0 VDC (from 5 VDC).

Click again on the button, and you will see the indicator becomes Black - indicating the "Off" state.
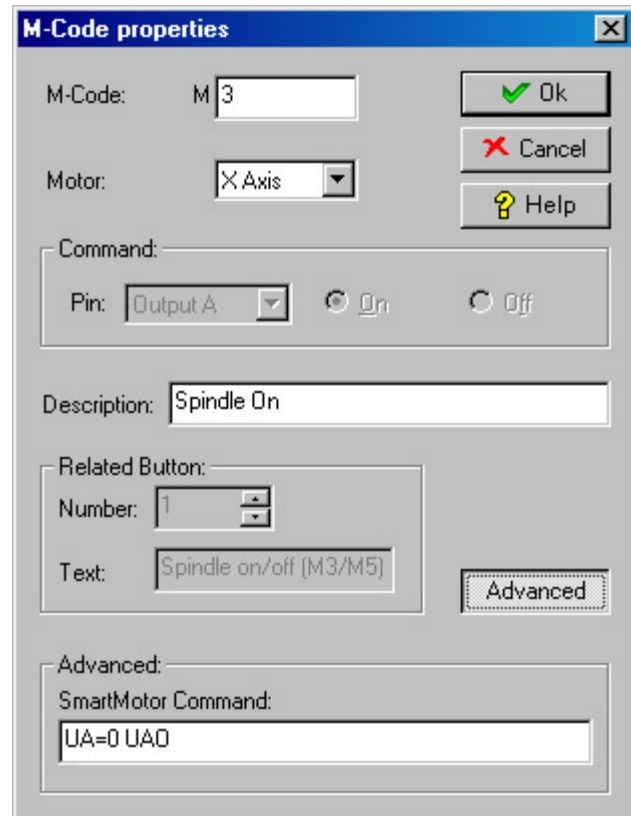
SmartMotor™ commands can include motion commands for axes setup as "uncoordinated" or even "GOSUB" commands for SmartMotors™ with programs loaded in them.

28

Change the G-Code program and add some M-Code lines, as shown in the following code:

```
M8
G1X0Y.5Z0
G2I.5J0
M9
G1Z.5
G2I.5J0
G0Z0
X0Y0
M30
```

Click the Start button to run the program. Note that when the indicator in the source view reaches the M-Code lines, their related button on the Panel View changes color to indicate the new state on the digital output.

IMPORTING DXF FILES

If you have AutoCAD® installed on your computer, start AutoCAD® and draw a number of simple objects such as lines, poly-lines, arcs and circles. Please make sure that you are drawing in the range acceptable by **SMNC™** as defined in the *Axis* page of *Setup* property sheet.

Then export the drawing to a DXF file, and select a name for your drawing. Finally, exit or minimize AutoCAD®.

Run **SMNC™** and select the *Import DXF file* … sub-item from the File menu. The *Import DXF File* dialog box will appear. (Figure 19)
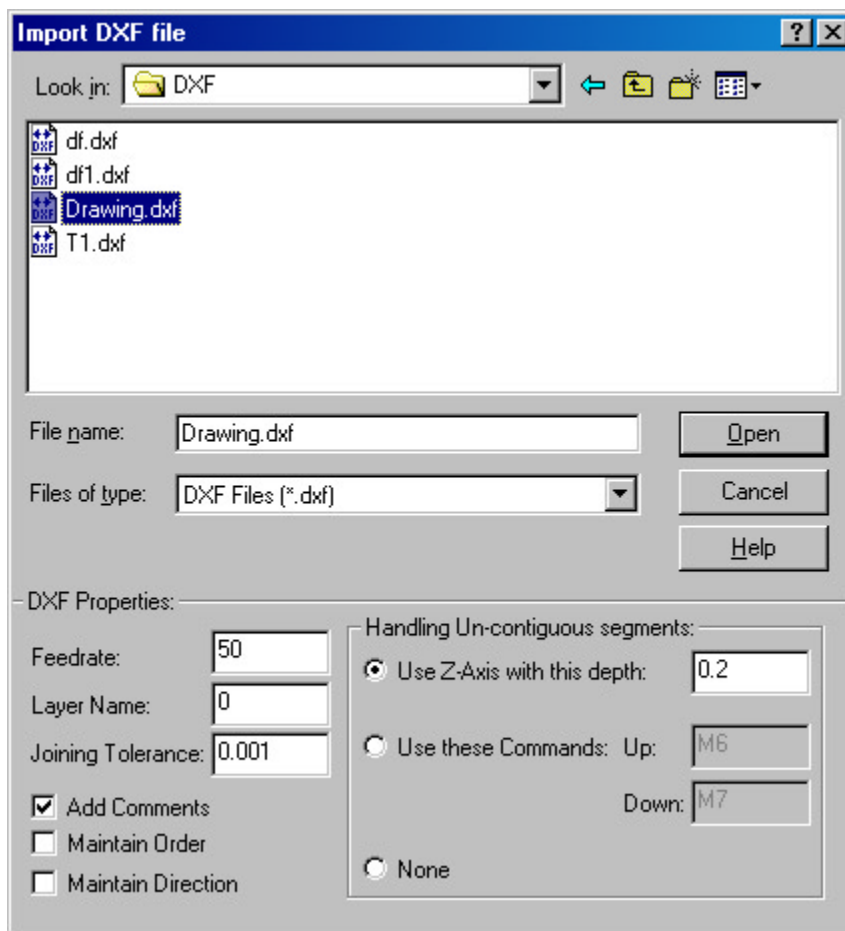

The DXF conversion delivers center line motion. The Offset function within AutoCAD® may be useful in defining proper tool path.

Figure 19: *Import DXF file dialog box*

Select the DXF file you created by AutoCAD® and click on the Open button. You will see that **SMNC™** creates an equivalent G-Code program for the DXF file. Run the created G-Code program and check the results.

ERROR HANDLING:

SMNC™ comes with robust error handling capabilities. Add some erroneous lines to the G-Code program as shown in the following code:

```
M8
G1X0Y.5Z0
G2I.5J0
M9
G1Z.5
G2I.5J0
G1X5              * Error
X0Y0Z0
G13               * Warning
G2I0.5J0.5X2      * Error
G0Z0
X0Y0
M30
```

Now click on the *Start* button to run the program. You will see that it detects two errors and one warning and shows them in the *Errors View* (Figure 20).
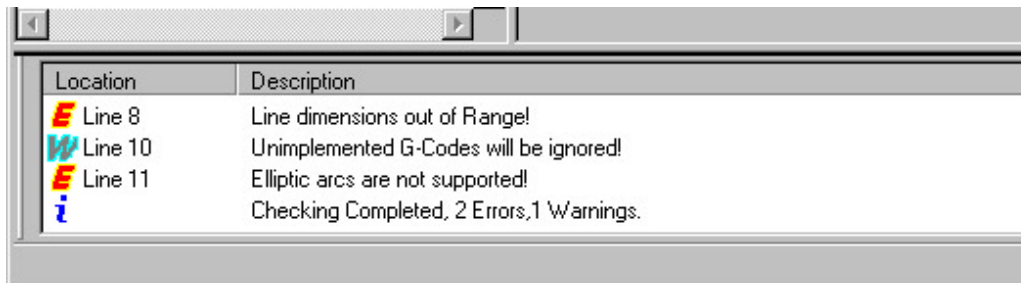

Figure 20: The Errors and Warnings in the Errors View

You can always see the results of checking and running the programs in the status group box on the top left of *Panel View* (Figure 17).

Double click on one of the errors. You will see that the line containing the error will be highlighted in the *Source View*, making it easy for you to edit the line to fix the error. Now click one of the errors to select it, and press the F1 key. You will see the error message and the guidelines to fix the error appear in the help window. You can also get help specific to an error using the context sensitive help button on the toolbar (The button with an arrow and a question mark near it). Click on the button and then click on the desired error line in the *Errors View*.

If there are one or more errors in the program, **SMNC™** won't run the program; but if there are only warnings, **SMNC™** starts running the program after checking the program and reporting the warnings in the *Errors View*.

WRITING AND RUNNING A COORDINATED MOTION FILE

If the required contouring cannot be specified by simple G-Codes, it can be specified using the Coordinated Motion file format.

Select the *New Coordinated Motion file* sub-item from the *File* menu; and type the following data in the *Source View*. Please note that the fields in each line are delimited by Tabs.

0	0	0	0
1000	1000	1000	1024
2000	2000	2000	2048
3000	3000	3000	3072
4000	4000	4000	4096
5000	5000	5000	5120
6000	6000	6000	6144
7000	7000	7000	7168
8000	8000	8000	8192
9000	9000	9000	9216
10000	10000	10000	10240

You may create coordinated motion files in a spreadsheet and then save them as tab delimited text files. The first three columns in this table are the coordinates and the fourth column is the time for each coordinate. The 11 points and their time values specify the path and speed of movement. The point values are in SmartMotor™ Counts and the time values are in SmartMotor™ Sample Periods. After typing the data, save the file using the *Save As* sub-item from *File* menu. You will need to assign it a name.

Note that when you are working with a coordinated motion file, the buttons *Pause* and *Step* are disabled. There is no pause or step mode when running coordinated motion files. Furthermore, you cannot change the speed of movement on the fly using the *Feedrate* scroll bar, because the speed of movement is specified by the data in the table and cannot be altered in real time.

Click the *Start* button and see the resulting movement. Try again, studying the *Run Time Parameters* group box. Note that *Current Mode* and *Current Origin* are not available for coordinated motion files, because they are G-Code-specific parameters.

Each time you run the file, the device moves to the point (0,0,0) and then runs the file. This movement to the start point in the table is called position initialization. Different types of position initialization can be defined in the *General* page of the *Setup* property sheet.

35

USING SMNC™ HELP

You can use the **SMNC™** Help in different ways:

1. Context-Sensitive Help using F1:

Dialog and Message box:
Just use the F1 key while the box is displayed.

Errors View:
Select the line and press the F1 key. After pressing the F1 key, you will see the description of selected error.

Menu command:
Just press the F1 key when you are on the menu item that you want help on.

2. Context-Sensitive Help using the mouse:
There is a context sensitive help button on the tool bar. When you click on that (or press Shift+F1 on keyboard) the program enters the Help Mode and the cursor shape changes to the context sensitive help cursor, which is an arrow with a question mark near it. In Help Mode you can use the mouse or keyboard to select a menu command, or an object on the *Panel View* or *Toolbar*, or even an error message in the *Errors View*. Instead of executing the command, Help on that command or object is displayed.

3. Help Buttons:
You can also click on the Help button available on all dialog boxes to get help about that dialog box.

4. Table of Contents:
To see the list of topics of **SMNC™** Help use the *Contents* command in the *Help* menu.

5. Index:
To see the index of **SMNC™** Help, use the *Index* command in the *Help* menu.