M Series Tutorial CNC Functions (1) G0, G1 Commands

Software : Sysctrl Studio

Hardware : M series controller (this tutorial uses M512 as an example)

Note : Among the M series controllers, only the M500 series controllers (excluding the M500S series) support the CNC function. Other series do not support it.

Sysctrl Studio Basic Configuration

Motion Control → Axis Setting, enter the axis setting interface, and add three virtual servo axes (simulating actual use).
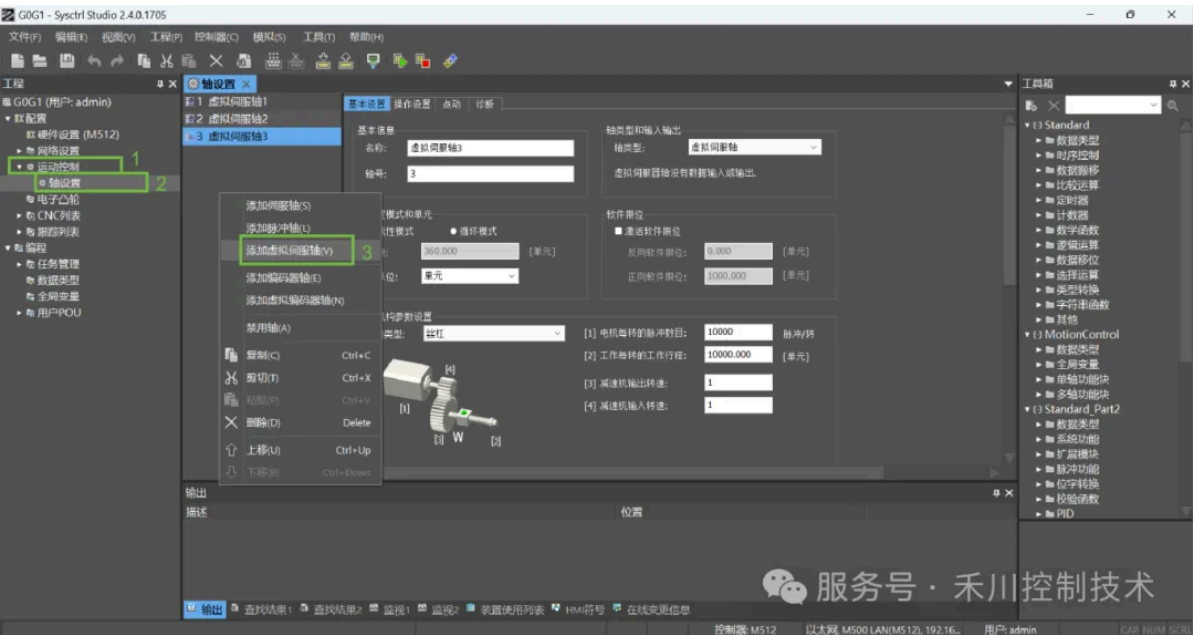
G-Code using function blocks

MC_AxesGroupAddAxis (Add an axis to an axes group)

① Functional description:

● This command is used to add an axis to an axis group. Before using the axis group, the axis must be added.

● IdentNum indicates the axis number in the axis group, ranging from 1 to 8. 1 represents the X axis, 2 represents the Y axis, etc.

② Parameter description (input and output variables):

■ Input variable

| Name	Meaning	Data type	Valid range	Default	Description
AxesGroup	Axis group number	USINT	Depend on model	Required field	Axis group number
Axis	Axis number	USINT	Depend on model	Required field	Specify axis number in the axis group
Execute	Execute	BOOL	TRUE or FALSE	FALSE	Executing when detecting the rising edge
IdentNum	Logical axis number in axis group	USINT	1~8	Required field	Logic axis number in axis group

■ Output variable

Name	Meaning	Data type	Valid range	Description
Done	Completed	BOOL	TRUE or FALSE	TRUE when instruction execution is completed
Busy	Executing	BOOL	TRUE or FALSE	TRUE when instruction is executed
Error	Error	BOOL	TRUE or FALSE	TRUE when there is an error
ErrorID	Error code	WORD	0~65535	Refer to "instruction error code description" for the meaning of the output error code value when an instruction execution error occurs.

MC_CoorMotion

(CNC execution command)
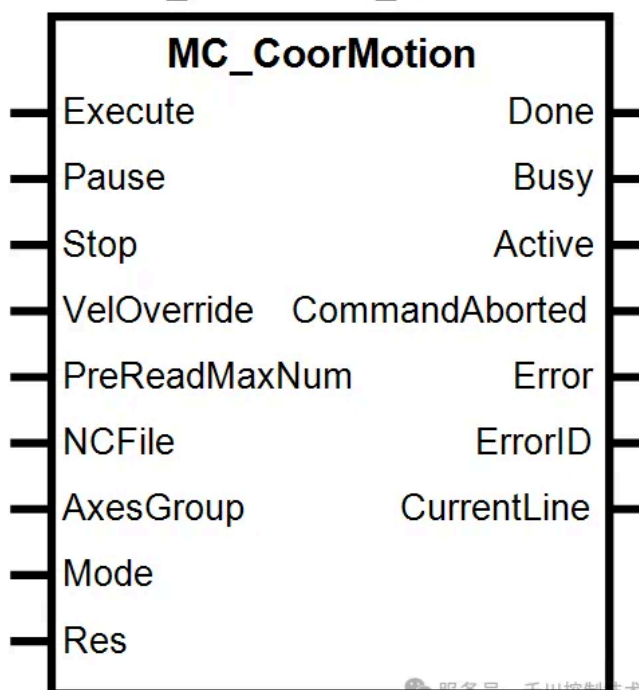
① Functional description:

● This command is used to execute the CNC code downloaded to the controller from the software. Before executing this command, the state of each axis in the axis group must be in StandStill state, otherwise the command execution will report an error.

● Before executing this instruction, first add each axis to an axis group using the MC_AxesGroupAddAxis instruction, and then execute this instruction. The axes that form a group are determined by the value of AxesGroup in the MC_AxesGroupAddAxis instruction. Multiple axis groups can be executed simultaneously. For example, axes 1, 2, and 3 can be grouped together, and axes 4, 5, and 6 can be grouped together. This can be achieved by using two MC_CoorMotion instructions with different AxesGroup values specified in each instruction.

● The MC_CoorMotion instruction controls the target speed together with the target speeds of other motion instructions. The target speed can be changed in real time by changing the value of the VelFactor (target speed ratio) of the instruction. Setting the VelFactor value to 0 can achieve the pause function.

● PreReadMaxNum indicates the number of pre-read CNC instructions. The setting range is 1~50.

● The NCFile input variable is used to specify the CNC number to be executed. This number can be viewed through the CNC number established in the software.

● The output variable CurrentLine of this instruction is used to display the number of lines executed by the CNC code specified by AxesGroup.

②Parameter description (input and output variables):

■ Input variable

Name	Meaning	Data type	Valid range	Default	Description
AxesGroup	Axis group number	USINT	1~8	Required field	Axis group number
Execute	Execute	BOOL	TRUE OR FALSE	FALSE	Executing when detecting the rising edge
Pause	Pause	BOOL	TRUE OR FALSE	FALSE	Pause the motion of all axes in the axis group and decelerate and stop according to the deceleration of the command executed in the CNC.
Stop	Stop	BOOL	TRUE OR FALSE	FALSE	Stop all axes in axis group in one cycle
VelOverride	Velocity override	LREAL	0~500	0	Velocity override
PreReadMaxNum	Number of pre-read CNC commands	UINT	1~5	Required field	Number of pre-read CNC commands
NCFile	CNC number	UINT	1~64	Required field	CNC number
AxesGroup	Axis group number	USINT	1~8	Required field	Axis group number
Mode	Mode	INT	Reserve	Reserve	Reserve
Res	Reserve	LREAL	Reserve	Reserve	Reserve

■ Output variable

Name	Meaning	Data type	Valid range	Function
Done	Done	BOOL	TRUE OR FALSE	Change to TRUE after execution of the instruction is complete
Busy	Executing	BOOL	TRUE OR FALSE	Change to True when instruction is executing.
Active	Under control	BOOL	TRUE OR FALSE	Change to TRUE when the axis is under control
CommandAborted	Abortion	BOOL	TRUE OR FALSE	Change to TRUE when the instruction is aborted
Error	Error	BOOL	TRUE OR FALSE	Change to True when error occurs.

ErrorID	Error code	WORD	0~65535	Output error code when instruction execution error occurs Please refer to the 'Instruction error code description'
CurrentLine	Currently executing lines	UDINT	0~65535	Currently executing lines

MC_SetMoveDirectParm

(CNC individual positioning motion parameters setting)

① Functional description:

● The speed, acceleration, deceleration and jerk parameters during G0 motion can be set using the MC_SetMoveDirectParm instruction.

● The parameter Velocity is an array containing 8 elements, which correspond to the velocities of the 8 dimensions (X~Q) in sequence.

● The parameter Acceleration is an array containing 8 elements, which correspond to the acceleration of 8 dimensions (X~Q) in sequence.

● The parameter Deceleration is an array containing 8 elements, which correspond to the deceleration of 8 dimensions (X~Q) in sequence.

■ **Input variable**

Name	Meaning	Data type	Valid range	Default	Description
AxesGroup	Axis group number	USINT	1~8	Required field	Axis group number
Execute	Execute	BOOL	TRUE OR FALSE	FALSE	Executing when detecting the rising edge
Velocity	Fast positioning velocity	ARRAY [1..8] OF LREAL	Positive	Required field	Fast positioning velocity
Acceleration	Fast positioning acceleration	ARRAY [1..8] OF LREAL	Positive	Required field	Fast positioning acceleration
Deceleration	Fast positioning deceleration	ARRAY [1..8] OF LREAL	Positive	Required field	Fast positioning deceleration
Jerk	Fast positioning jerk	ARRAY [1..8] OF LREAL	Positive	Required field	Fast positioning jerk

■ **Output variable**

Name	Meaning	Data type	Valid range	Function
Done	Done	BOOL	TRUE OR FALSE	Change to TRUE after execution of the instruction is complete
Busy	Executing	BOOL	TRUE OR FALSE	Change to True when instruction is executing.
Error	Error	BOOL	TRUE OR FALSE	Change to True when error occurs.
ErrorID	Error code	WORD	0~65535	Output error code when instruction execution error occurs Please refer to the 'Instruction error code description'

MC_SetMovelinearParm

(CNC interpolation motion default parameter setting)

① Functional description:

● This command is used to set the default velocity, acceleration, deceleration, and jerk parameters used in G1, G2, and G3 in the CNC. This command is used to input the variables Velocity, Acceleration, Deceleration, and Jerk to set them. If interpolation velocity, acceleration, and deceleration are not specified using E and F in G1, G2, and G3, the interpolation velocity,

acceleration, and deceleration set in this command are used. The Jerk value in G1, G2, and G3 can only be set using this command.

②Parameter description (input and output variables):

■ Input variable

| Name	Meaning	Data type	Valid range	Default	Description
AxesGroup	Axis group number	USINT	1~8	Required field	Axis group number
Execute	Execute	BOOL	TRUE OR FALSE	FALSE	Executing when detecting the rising edge
Velocity	Interpolation velocity	LREAL	Positive	Required field	Interpolation velocity
Acceleration	Interpolation acceleration	LREAL	Positive	Required field	Interpolation acceleration
Deceleration	Interpolation deceleration	LREAL	Positive	Required field	Interpolation deceleration
Jerk	Interpolation jerk	LREAL	Positive	Required field	Interpolation jerk

■ Output variable

Name	Meaning	Data type	Valid range	Function
Done	Done	BOOL	TRUE OR FALSE	Change to TRUE after execution of the instruction is complete
Busy	Executing	BOOL	TRUE OR FALSE	Change to True when instruction is executing.
Error	Error	BOOL	TRUE OR FALSE	Change to True when error occurs.
ErrorID	Error code	WORD	0~65535	Output error code when instruction execution error occurs Please refer to the 'Instruction error code description'

G0 rapid positioning

Step 1: Create a CNC list

Right-click on the [CNC List] on the left side of the software to create a new one. A "New CNC" window will pop up. Click OK to create a CNC list numbered 1.
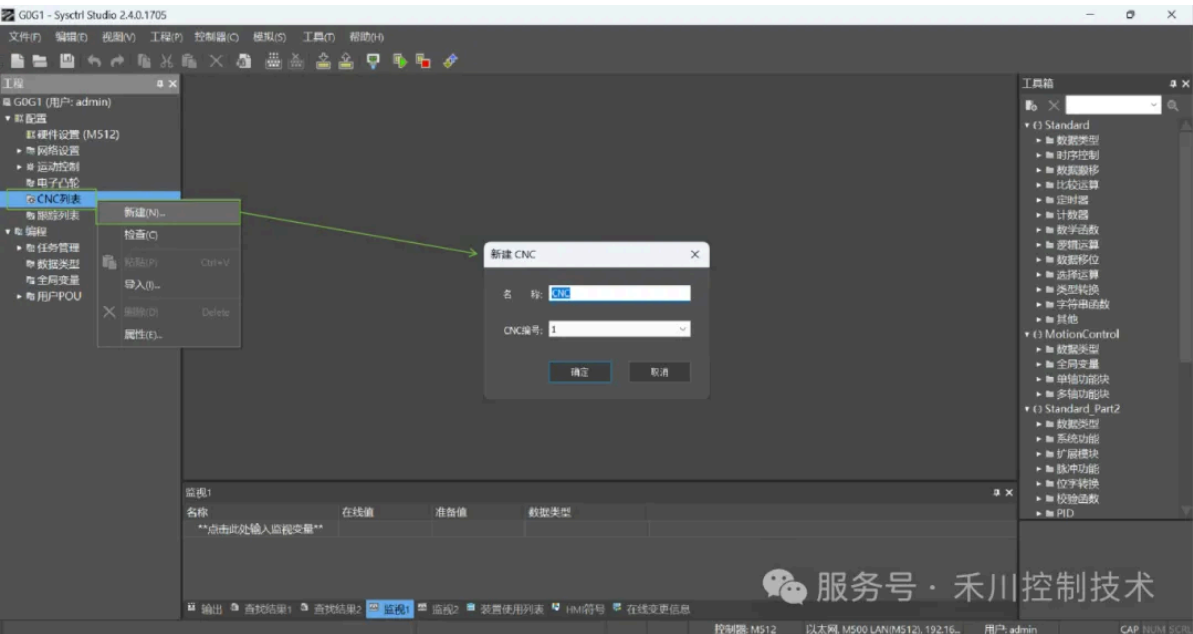
Step 2: Write the program

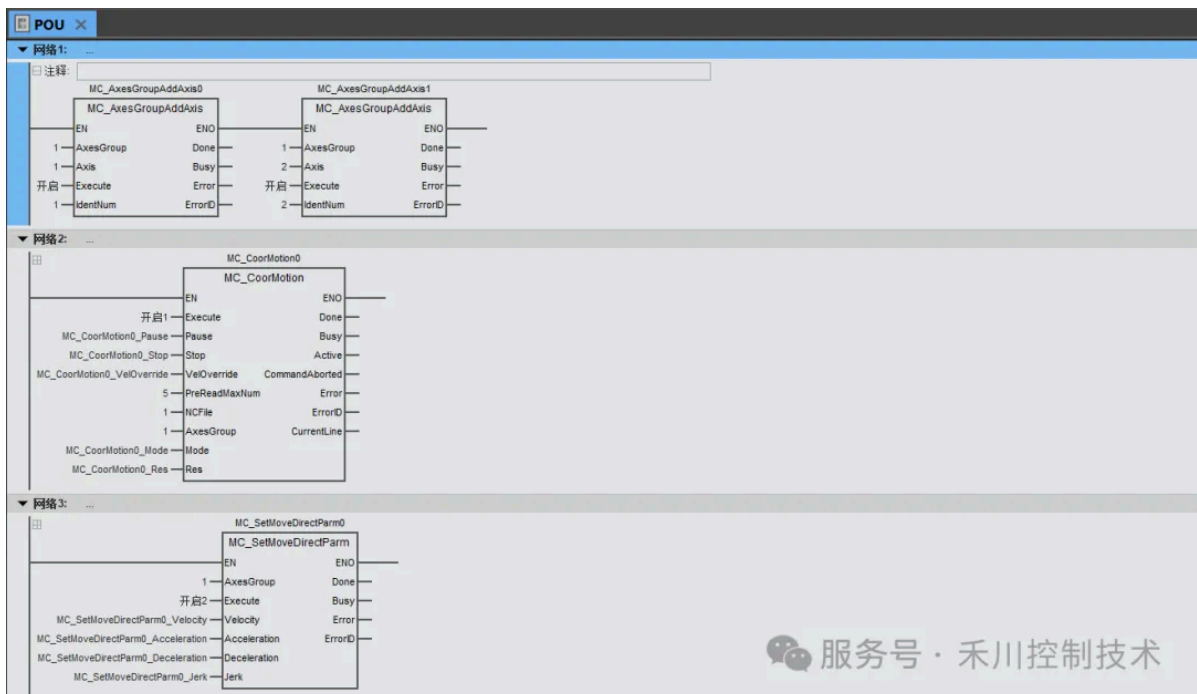**① Write G-Code program file**

Example: N10 G0 X1000 Y1000

② Write motion control program

● Create an axis group (no axis group enable required)

● MC_SetMoveDirectParm instruction sets the required motion parameters
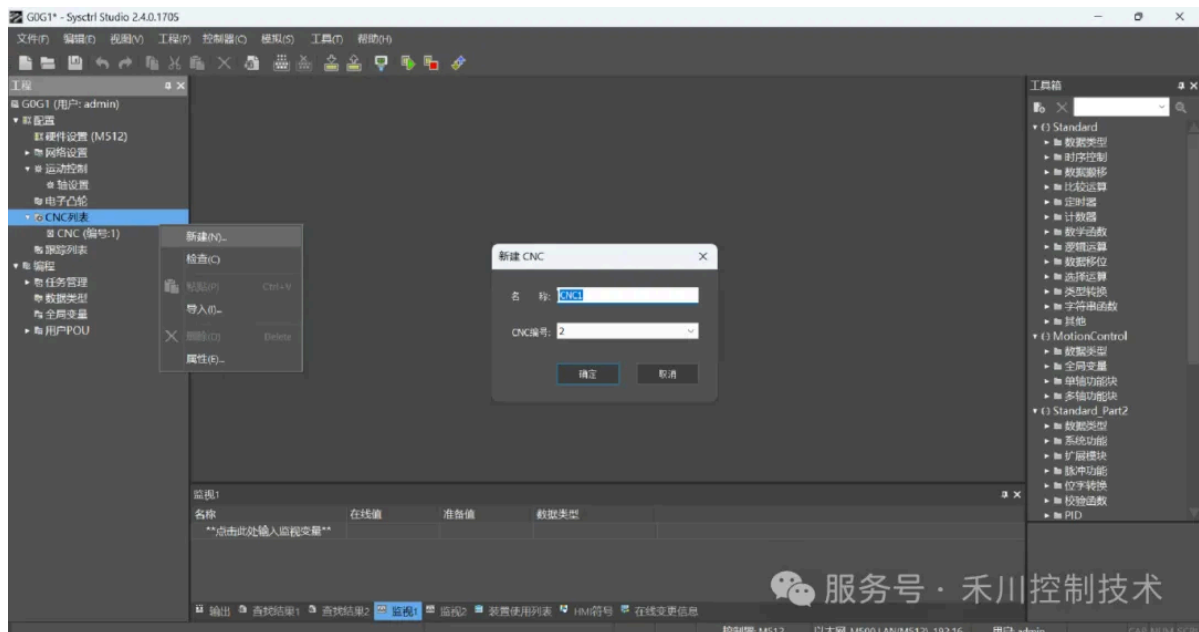
● Execute G-Code program files

Note : The G0 instruction cannot directly set the speed, acceleration, or deceleration (it runs at the default speed of 1000 units per second). The required motion parameters for each axis must be set using the MC_SetMoveDirectParm instruction.

G1 linear interpolation

Step 1: Create a CNC list

Right-click on [CNC List] on the left side of the software to create a new one. A "New CNC" window will pop up. Click OK to create a CNC list numbered 2.

Step 2: Write the program

Method 1 (setting the composite speed and composite acceleration and deceleration through the G1 command):
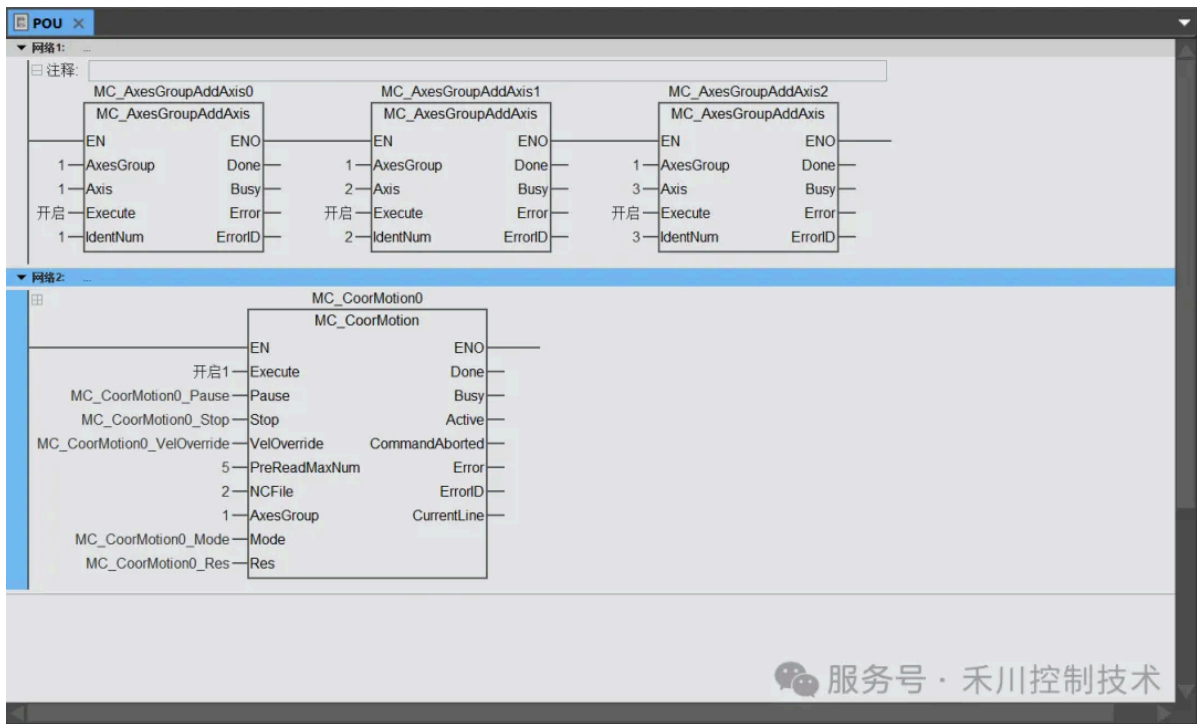
① Write G-Code program file

Example: N10 G1 X100 Y200 Z100 E20 E-50 F20

② Write motion control program

● Create an axis group (no axis group enable required)

● Execute G-Code program files

Method 2:

(using the MC_SetMoveLinearParm instruction to set the composite speed and composite acceleration and deceleration):

①Write G-Code program file

Example: N10 G1 X100 Y200 Z100

② Write motion control program

● Create an axis group (no axis group enable required)

● MC_SetMovelinearParm instruction sets the required motion parameters

● Execute G-Code program files