M-Series Tutorial (Advanced Tutorial) | CNC Functions (2) G2, G3, G4, G17, G18, G19 Instructions and M Codes

Preliminary preparations

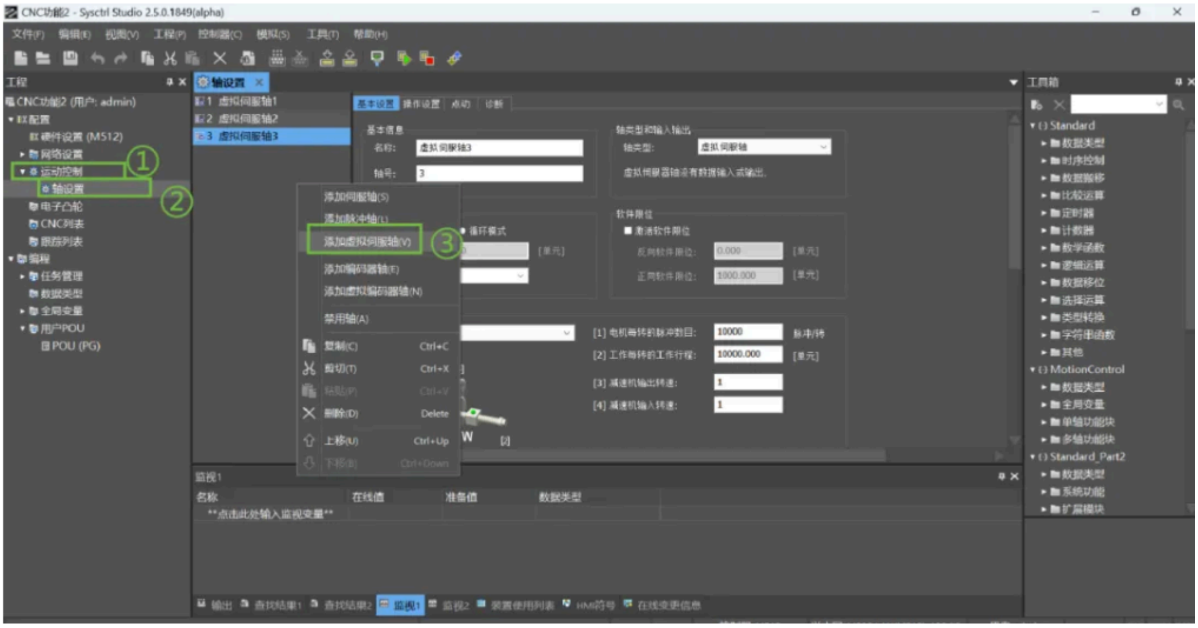
Software : Sysctrl Studio (available from Chuanhe Automation Academy)

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

Note : Only the M500 series controllers (excluding the M500S series) support CNC functions; other series do not.

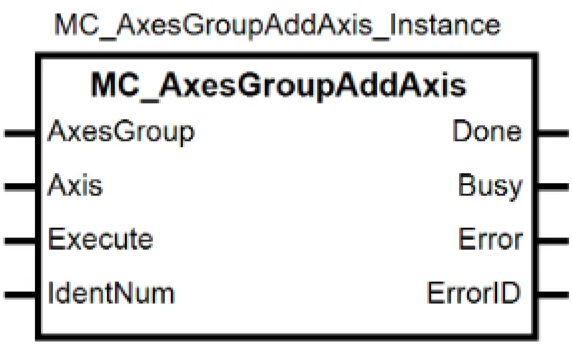
Sysctrl Studio basic configuration

Motion Control → Axis Settings: Enter the axis settings interface and add a virtual servo axis (simulating actual use).



G-Code uses function blocks

MC_ AxesGroupAddAxis (Add axis to axis group)



① **Function Description:**

● This command is used to add a shaft to a shaft group. A shaft must be added before using a shaft group.

● IdentNum indicates the axis number within the axis group, ranging from 1 to 8. 1 represents the X-axis, 2 represents the Y-axis, and so on...

② Parameter description (input and output variables):

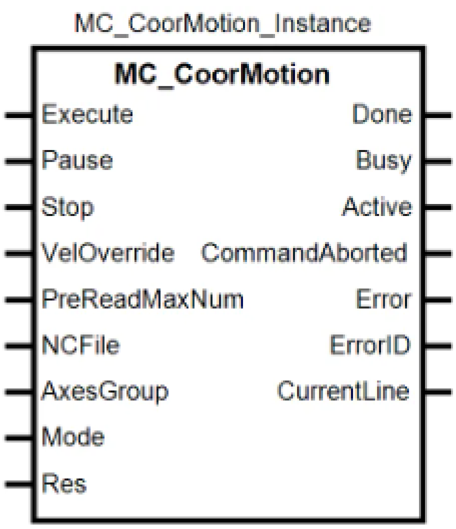
Input Variables

Variable Type	Name	Description	Data Type	Setting Range	Default Value	Explanation
Input Variable	AxesGroup	Axis Group Number	USINT	Series M500: 1~8 Other models: 1	No default value	Number of the axis group
Input Variable	Axis	Axis Number	USINT	Determined by controller model	No default value	Specify the number of the controlled axis
Input Variable	Execute	Trigger	BOOL	TRUE / FALSE	FALSE	The instruction will execute upon rising edge detection of this signal
Input Variable	IdentNum	Logical Axis Index within Axis Group	USINT	1~8	No default value	Logical number of the axis inside the axis group

Output Variables

Variable Type	Name	Description	Data Type	Output Range	Function
Output Variable	Done	Execution Complete	BOOL	TRUE / FALSE	Turns TRUE when the instruction finishes execution
Output Variable	Busy	Busy Executing	BOOL	TRUE / FALSE	Remains TRUE while the instruction is running
Output Variable	Error	Fault Occurred	BOOL	TRUE / FALSE	Turns TRUE if any exception occurs during instruction execution
Output Variable	ErrorID	Fault Code	WORD	0~65535	Output the corresponding fault code when an execution exception happens

MC_CoorMotion (CNC execution instruction)



① Function Description:

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

● Before executing this command, please add each axis to an axis group using the MC_AxesGroupAddAxis command, and then execute the command. Which axes form a group is determined by the value of AxesGroup in the MC_AxesGroupAddAxis command. Multiple axis groups can be executed simultaneously. For example, to group axes 1, 2, and 3 into one axis group, and axes 4, 5, and 6 into another axis group, use two MC_CoorMotion commands, specifying different values for AxesGroup in the two commands.

● The MC_CoorMotion command, along with other motion commands, controls the target speed. The target speed can be changed in real-time by altering the value of the VelFactor (target speed ratio). Setting the VelFactor value to 0 enables a pause function.

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

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

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

② Parameter description (input and output variables):

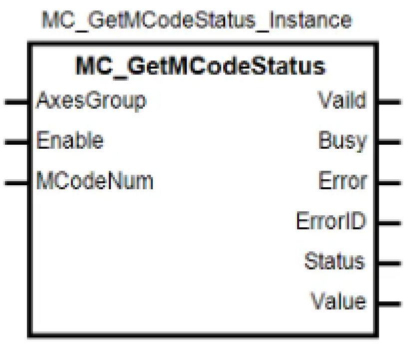
Input

Variable Type	Name	Description	Data Type	Setting Range	Default Value	Explanation
Input Variable	Execute	Trigger	BOOL	TRUE / FALSE	FALSE	The instruction executes when the rising edge of this signal is detected
Input Variable	Pause	Pause Motion	BOOL	TRUE / FALSE	FALSE	Pause all axes in the axis group, decelerate to stop at the deceleration defined in CNC commands
Input Variable	Stop	Emergency Stop	BOOL	TRUE / FALSE	FALSE	Emergency stop all axes in the axis group within one cycle
Input Variable	VelOverride	Velocity Override	LREAL	0~500	0	Velocity override factor
Input Variable	PreReadMaxNum	Max CNC Block Pre-read Count	UINT	1~50	No default value	Maximum number of CNC lines to pre-read
Input Variable	NCFile	CNC Program Number	UINT	1~64	No default value	Index number of the CNC program file
Input Variable	AxesGroup	Axis Group Index	USINT	1~8	No default value	Serial number of target axis group
Input Variable	Mode	Mode Select	INT	Reserved	/	Reserved for future use
Input Variable	Res	Reserved	LREAL	Reserved	/	Reserved for future use

Output

Variable Type	Name	Description	Data Type	Output Range	Function
Output Variable	Done	Execution Completed	BOOL	TRUE / FALSE	Turns TRUE once the instruction finishes execution
Output Variable	Busy	Executing	BOOL	TRUE / FALSE	Remains TRUE while the instruction is running
Output Variable	Active	Axis Under Control	BOOL	TRUE / FALSE	Turns TRUE when the instruction is controlling axes
Output Variable	CommandAborted	Motion Aborted	BOOL	TRUE / FALSE	Turns TRUE if the motion command is interrupted
Output Variable	Error	Fault Detected	BOOL	TRUE / FALSE	Turns TRUE if an exception occurs during execution
Output Variable	ErrorID	Fault Code	WORD	0~65535	Output the corresponding fault code upon execution error
Output Variable	CurrentLine	Current CNC Line No.	UDINT	0~65535	Line number of the CNC block currently being executed

MC_GetMCodeStatus (Read M code status)



① Function Description:

● This instruction is used to read the status of a specified M code and its corresponding value. The M code number is specified by the input variable MCodeNum. When the input variable Enable is TRUE, the output variables Status and Value are updated once per synchronization clock cycle; when Enable is FALSE, the output variables Status and Value stop updating and retain the values from the last Enable set to TRUE.

● The state of the M code and the corresponding value of the M code change according to the execution of the M code in the CNC code during CNC code execution.

② Parameter description (input and output variables):

Input

Variable Type	Name	Description	Data Type	Setting Range	Default Value	Explanation
Input Variable	AxesGroup	Axis Group Index	USINT	1~8	No default value	Serial number of target axis group
Input Variable	Enable	Enable Readback	BOOL	TRUE / FALSE	FALSE	Set TRUE to read M-code status; Set FALSE to stop reading M-code status
Input Variable	MCodeNum	Target M-code Number	USINT	Positive integer	No default value	Specified index number of target M-code

Output

Variable Type	Name	Description	Data Type	Output Range	Function
Output Variable	Valid	Data Valid	BOOL	TRUE / FALSE	Turns TRUE when the instruction is reading valid M-code status
Output Variable	Busy	Executing	BOOL	TRUE / FALSE	Remains TRUE while the instruction is running
Output Variable	Error	Fault Detected	BOOL	TRUE / FALSE	Turns TRUE if an exception occurs during execution
Output Variable	ErrorID	Fault Code	WORD	0~65535	Output corresponding fault code upon execution error
Output Variable	Status	M-code Flag Status	BOOL	TRUE / FALSE	Boolean status of the specified M-code
Output Variable	Value	M-code Numeric Value	LREAL	Positive real number	Numeric value associated with the specified M-code

MC_ResetMCode (M code reset)

① Function Description:

● This instruction is used to reset the M-code with the specified number. The state of the M-code becomes TRUE after the M-code is executed, and this instruction can be used to reset the state of the M-code to FALSE.

② Parameter description (input and output variables):

Input

Variable Type	Name	Description	Data Type	Setting Range	Default Value	Explanation
Input Variable	AxesGroup	Axis Group Index	USINT	1~8	No default value	Serial number of target axis group
Input Variable	Execute	Trigger	BOOL	TRUE / FALSE	FALSE	The function block executes upon detection of the rising edge of this signal
Input Variable	MCodeNum	Target M-code Number	USINT	Positive integer	No default value	Index number of the specified M-code

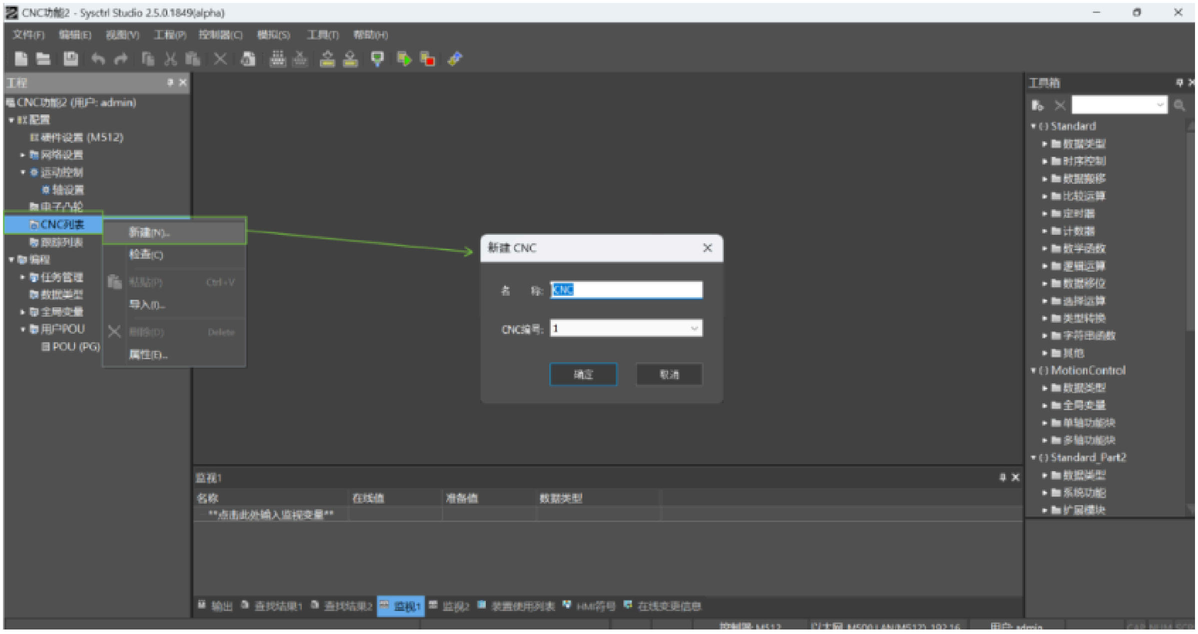
Output

Variable Type	Name	Description	Data Type	Output Range	Function
Output Variable	Done	Execution Complete	BOOL	TRUE / FALSE	Turns TRUE after the function block finishes execution
Output Variable	Busy	Executing	BOOL	TRUE / FALSE	Remains TRUE while the function block is running
Output Variable	Error	Fault Detected	BOOL	TRUE / FALSE	Turns TRUE if an abnormal condition occurs during execution
Output Variable	ErrorID	Fault Code	WORD	0~65535	Outputs the corresponding fault code when an execution exception occurs

G2/G3 Circular or Helical Interpolation

Step 1: Create a CNC list

Right-click on the 【CNC List】 on the left side of the software, select "New", and a "New CNC" window will pop up. Click "OK" to create CNC lists numbered 1 and 2.



Step 2: Writing the program

Command Format 1 - Specify the center:

① Write G-Code program files

example:

N10 G1 X100 Y100

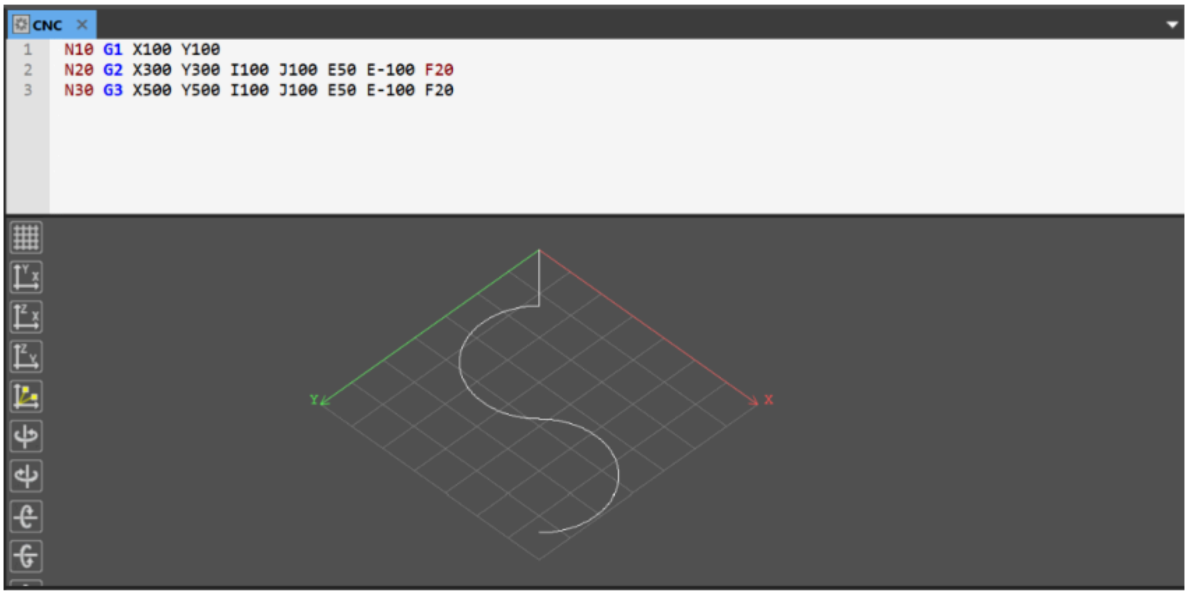
N20 G2 X300 Y300 I100 J100 E50 E-100 F20

N30 G3 X500 Y500 I100 J100 E50 E-100 F20

○ Explanation of the first line: G1 linear interpolation to the position X=100, Y=100 is used as the starting position for circular interpolation. The position in the CNC code defaults to absolute value mode.

○ Second line explanation: With the XY plane as a reference, the starting position coordinates of the clockwise circular interpolation are X=100, Y=100, and the center coordinates are X=200, Y=200. Parameters I, J, and K are the center parameters, representing the relative values between the center coordinates and the starting position.

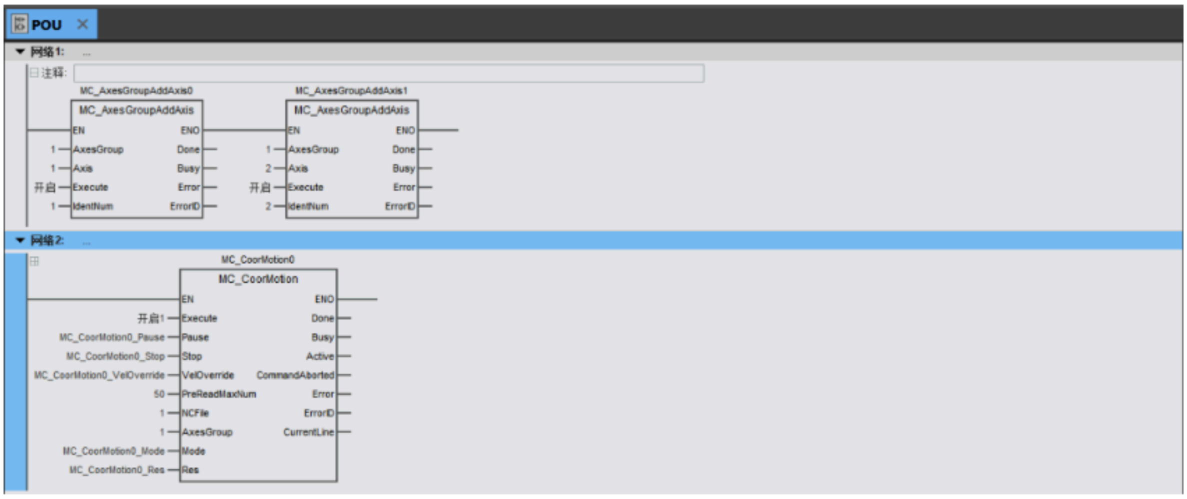
○ Third line explanation: With the XY plane as a reference, the starting position coordinates of the counterclockwise circular interpolation are X=300, Y=300, and the center coordinates are X=400, Y=400.



② Write motion control programs

● Create a shaft group (shaft group enable is not required)

● Execute G-Code program files



Command format 2 - Specify radius:

① Write G-Code program files

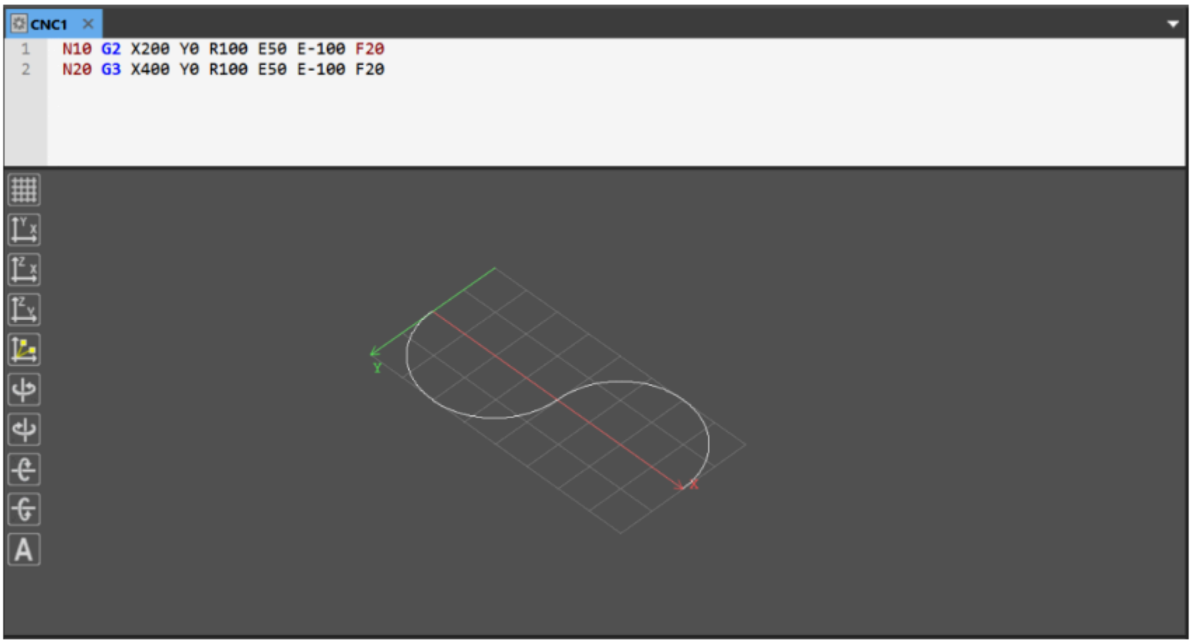
example:

N10 G2 X200 Y0 R100 E50 E-100 F20

N20 G3 X400 Y0 R100 E50 E-100 F20

○ First line explanation: With the XY plane as a reference, the target position coordinates for clockwise circular interpolation are X=200, Y=0, and the radius is 100.

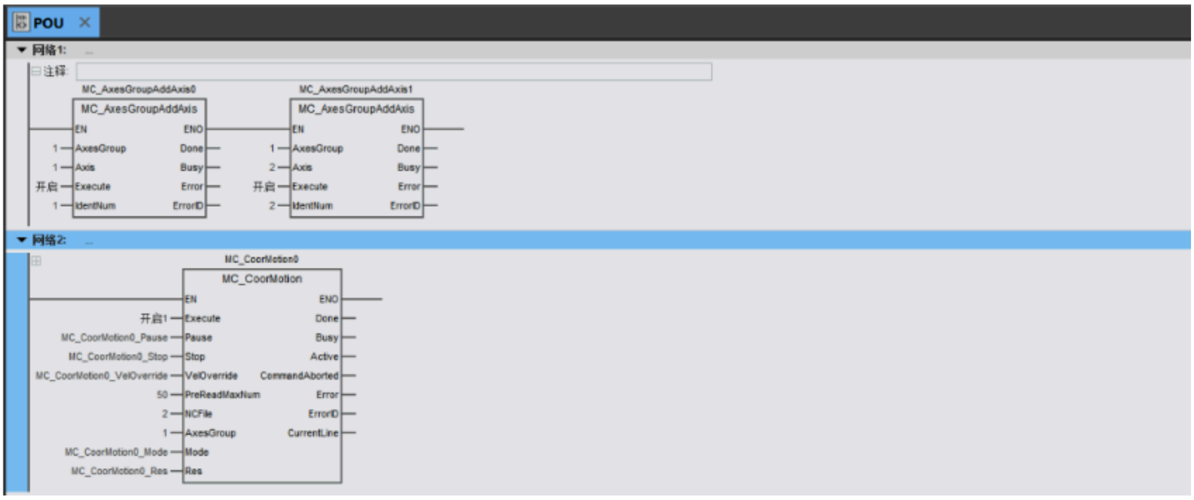
○ Second line explanation: With the XY plane as a reference, the target position coordinates of the counterclockwise circular interpolation are X=400, Y=0, and the radius is 100.



② **Write motion control programs**

● Create a shaft group (shaft group enable is not required)

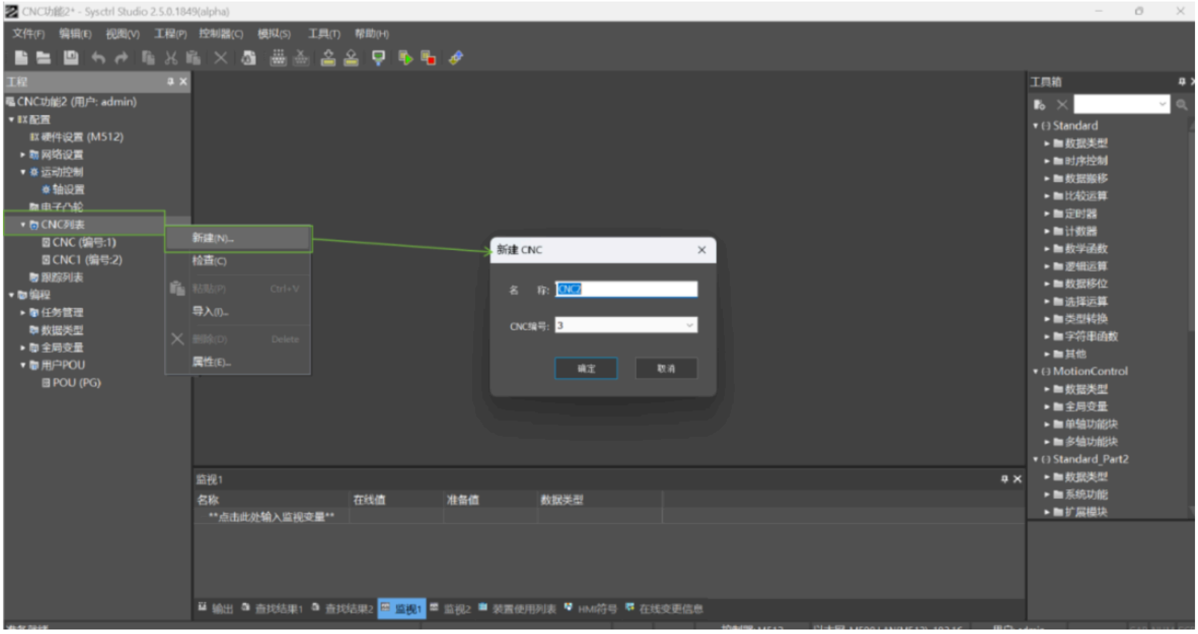
● Execute G-Code program files



G4 Delay

Step 1: Create a CNC list

Right-click on the 【CNC List】 on the left side of the software, select "New", and a "New CNC" window will pop up. Click "OK" to create a CNC list with the number 3.



Step 2: Writing the program

① Write G-Code program files

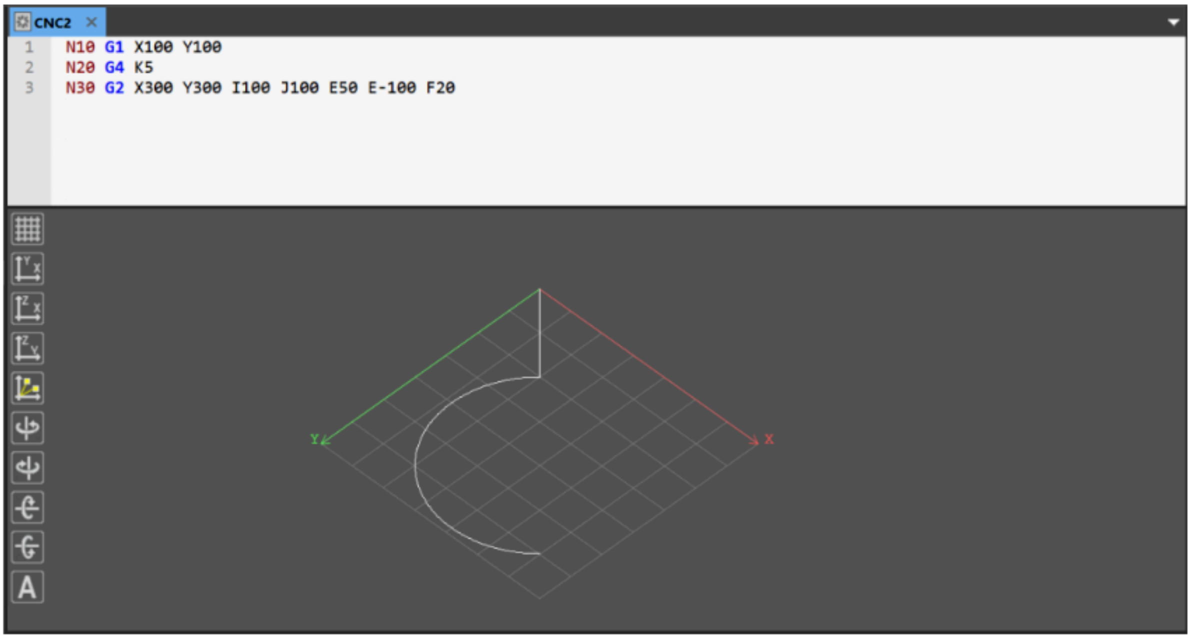
example:

N10 G1 X100 Y100

N20 G4 K5

N30 G2 X300 Y300 I100 J100 E50 E-100 F20

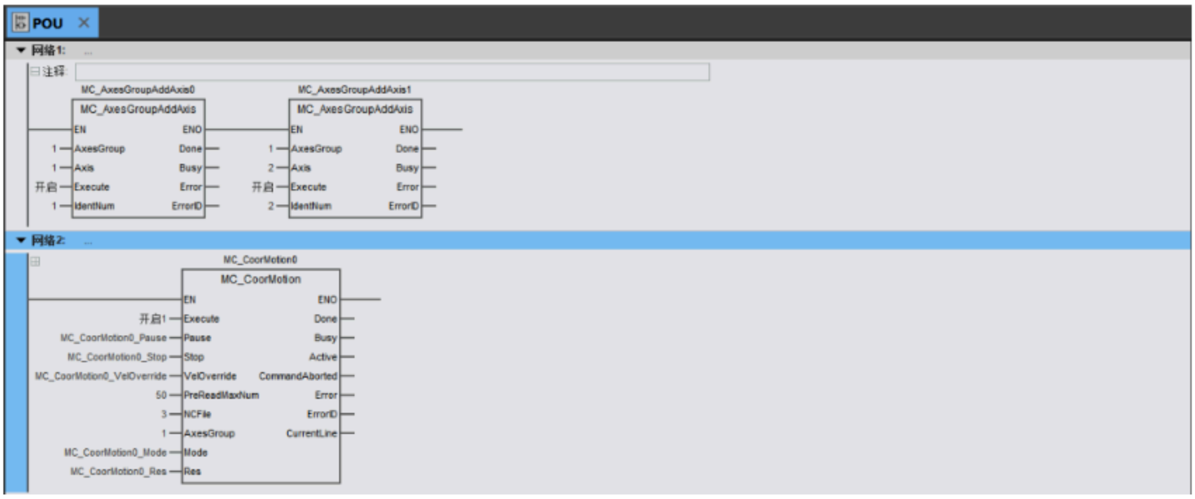
○ Second line explanation: Parameter K is the pause time, in seconds, range: 0.001~100000.



② Write motion control programs

● Create a shaft group (shaft group enable is not required)

● Execute G-Code program files



G17/G18/G19 Reference Plane Selection

Step 1: Write the program

① Write G-Code program files

◆ G17 is used to select the XY plane as the reference plane for the arc (spiral). Example:

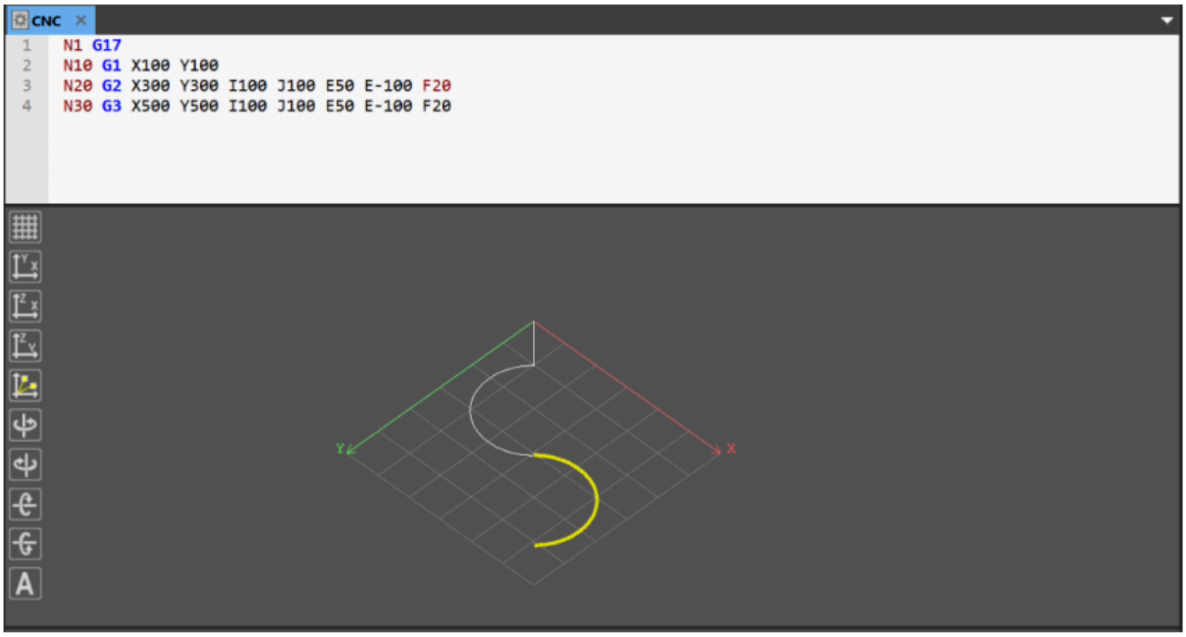
N1 G17

N10 G1 X100 Y100

N20 G2 X300 Y300 I100 J100 E50 E-100 F20

N30 G3 X500 Y500 I100 J100 E50 E-100 F20

Example Explanation: Parameters I and J are the center parameters.



◆ G18 is used to select the XZ plane as the reference plane for the arc (spiral). Example:

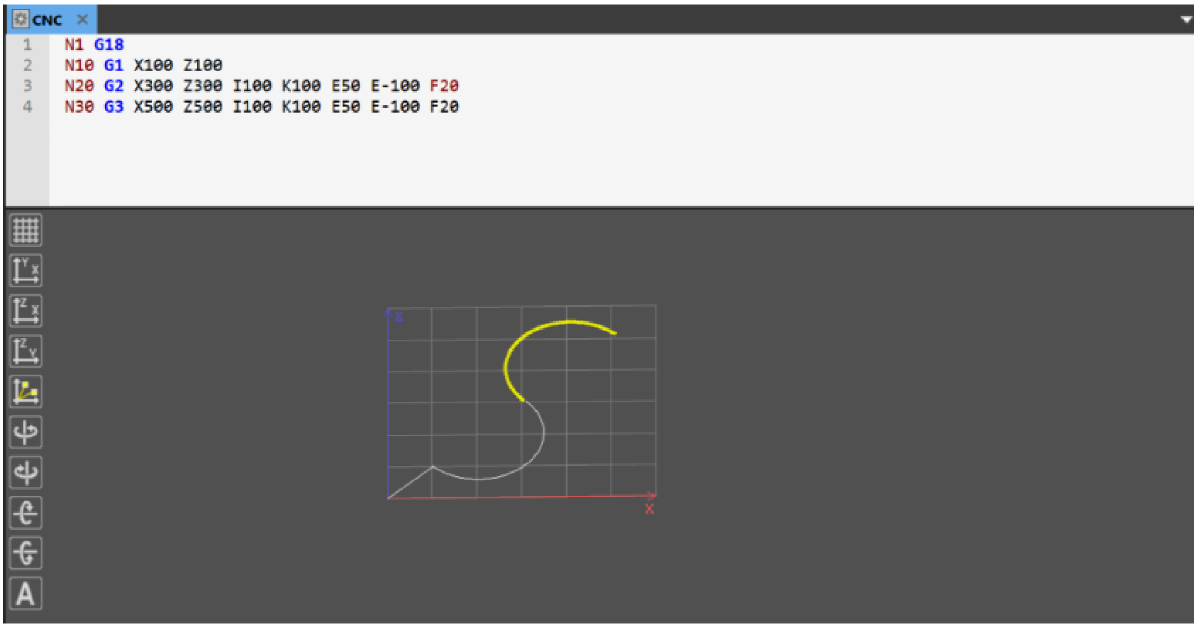
N1 G18

N10 G1 X100 Z100

N20 G2 X300 Z300 I100 K100 E50 E-100 F20

N30 G3 X500 Z500 I100 K100 E50 E-100 F20

Example Explanation: Parameters I and K are the center parameters.



◆ G19 is used to select the reference plane for the circular arc (spiral) as the YZ plane. Example:

N1 G19

N10 G1 Y100 Z100

N20 G2 Y300 Z300 J100 K100 E50 E-100 F20

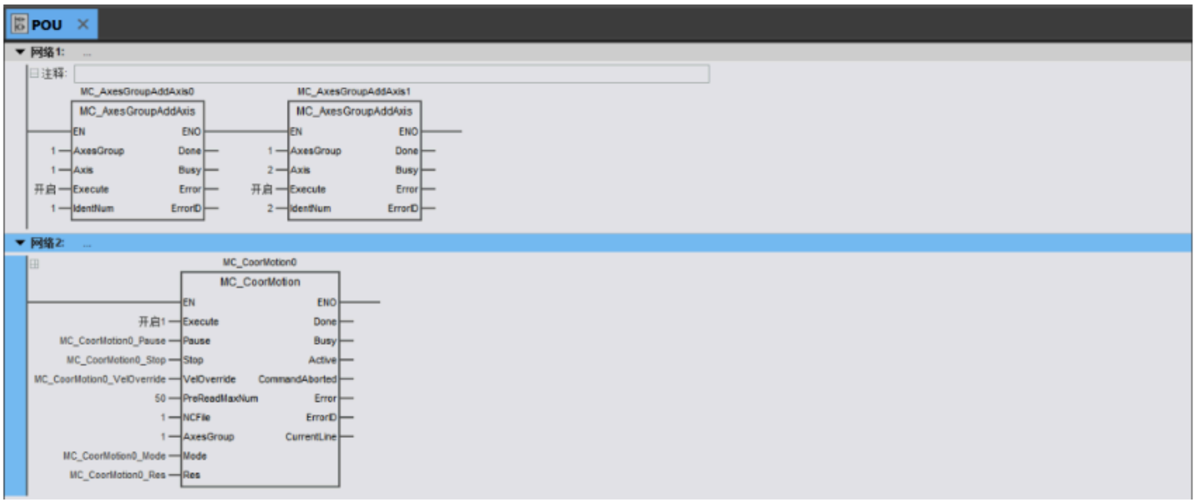
N30 G3 Y500 Z500 J100 K100 E50 E-100 F20

Example Explanation: Parameters J and K are the center parameters.

② Write motion control programs

● Create a shaft group (shaft group enable is not required)

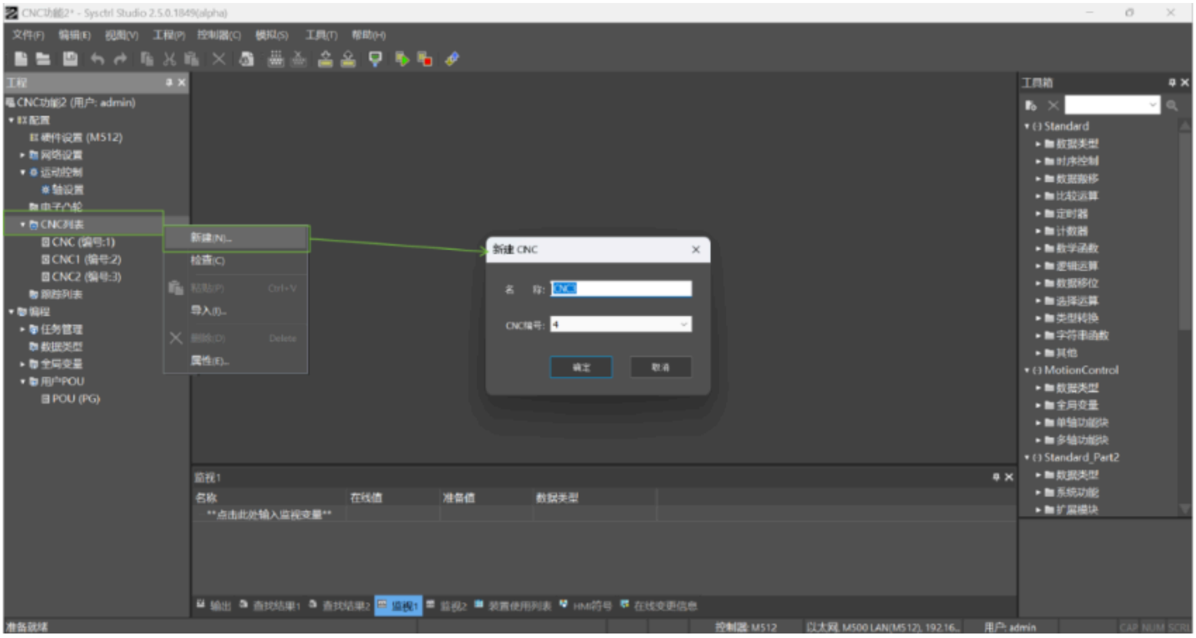
● Execute G-Code program files



M code

Step 1: Create a CNC list

Right-click on the 【CNC List】 on the left side of the software, select "New", and a "New CNC" window will pop up. Click "OK" to create a CNC list with the number 4.



Step 2: Writing the program

Instruction format 1 - **M codes and G codes are on the same line** :

① Write G-Code program files

example:

N10 G1 X100 Y100 E100 E-100 F30

N20 G1 X300 Y300 E100 E-100 F30 M0 D123

N30 G1 X500 Y500 E100 E-100 F30

Example Explanation: When a G-Code program executes to the line containing the M code, it will not pause because of the M code, but the output status and data corresponding to the M code will still be output.

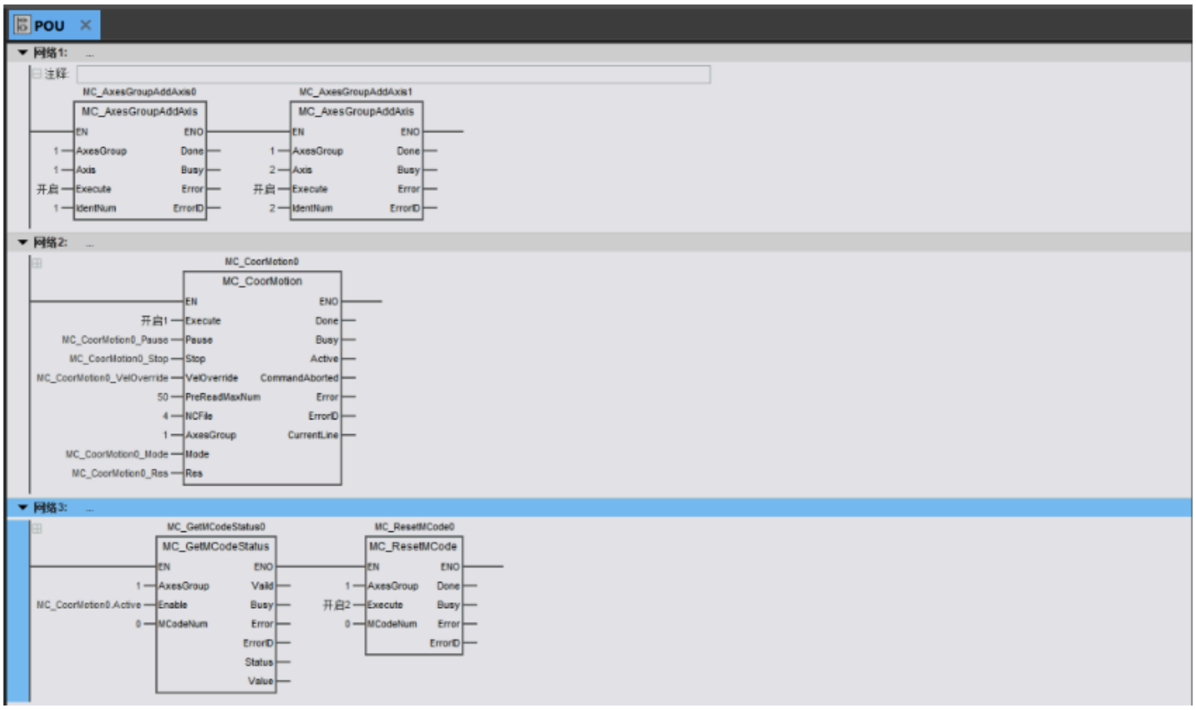
② Write motion control programs

● Create a shaft group (shaft group enable is not required)

● Execute G-Code program files

● The MC_GetMCodeStatus command reads the M-code status.

● The MC_ResetMCode instruction performs an M-code reset.



Instruction format 2 - ***M codes and G codes are not on the same line***:

① Write G-Code program files

example:

N10 G1 X100 Y100 E100 E-100 F30

N15 M0 D123

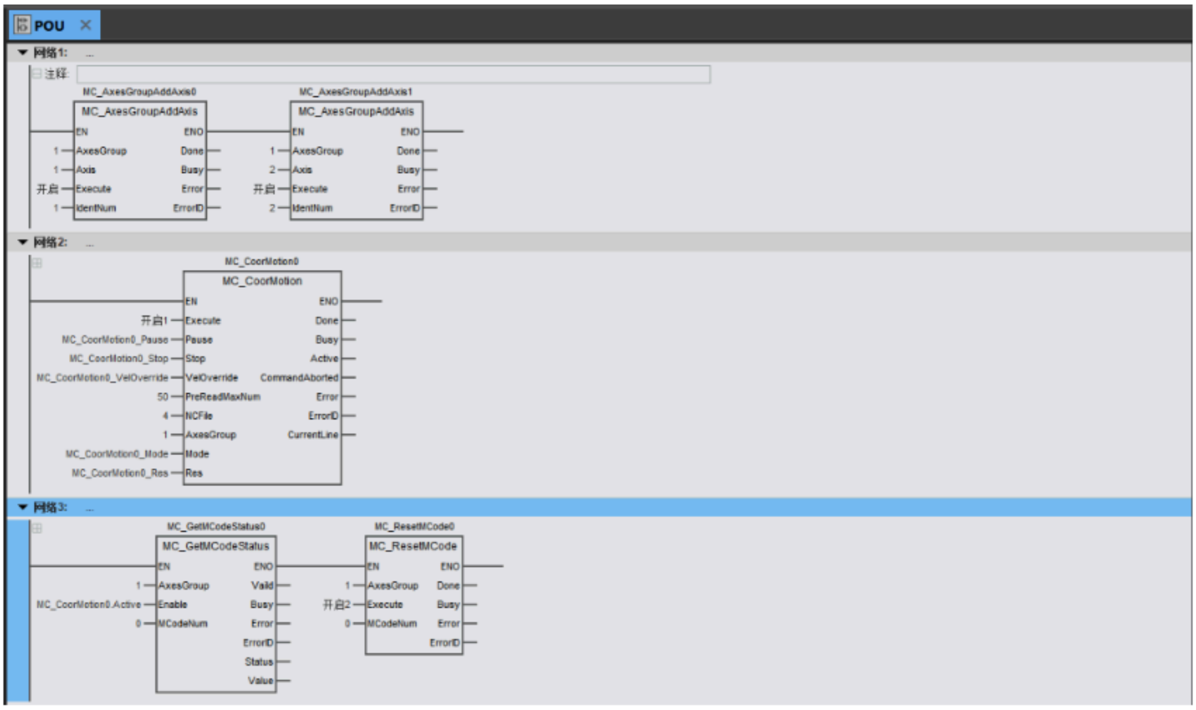
N20 G1 X300 Y300 E100 E-100 F30

N30 G1 X500 Y500 E100 E-100 F30

Example Explanation: When the CNC code reaches the line containing the M code, execution stops. The M code needs to be reset before execution can continue.

② Write motion control programs

- Create a shaft group (shaft group enable is not required)
- Execute G-Code program files
- The MC_GetMCodeStatus command reads the M-code status.
- The MC_ResetMCode instruction performs an M-code reset.

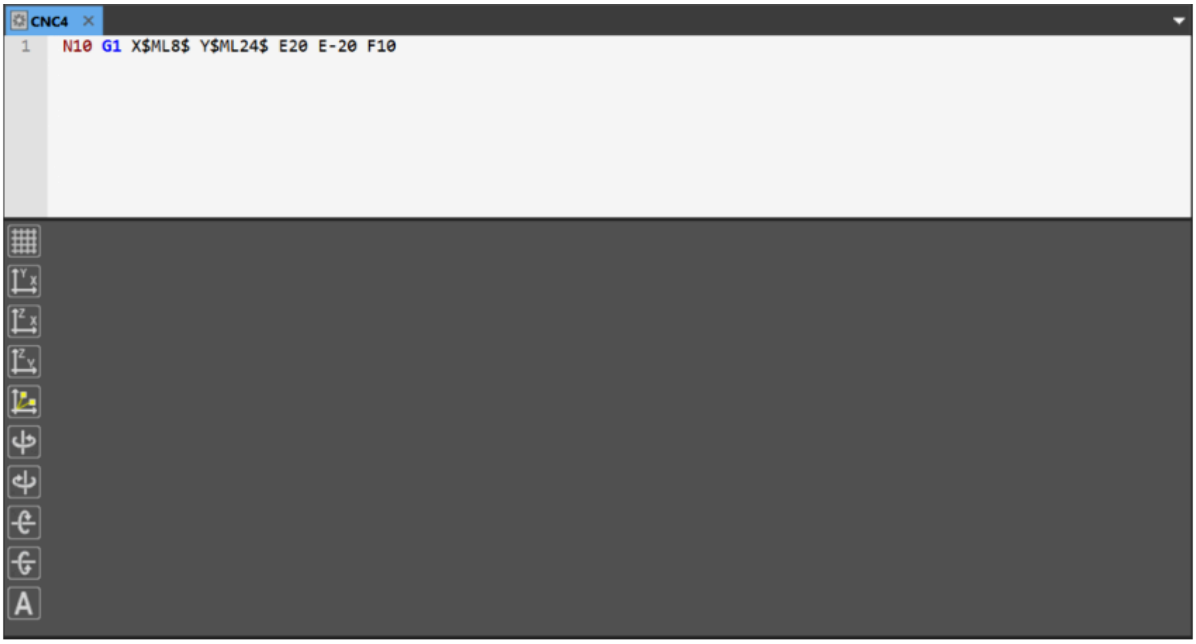


CNC code parameter usage device

Representation methods (extension)

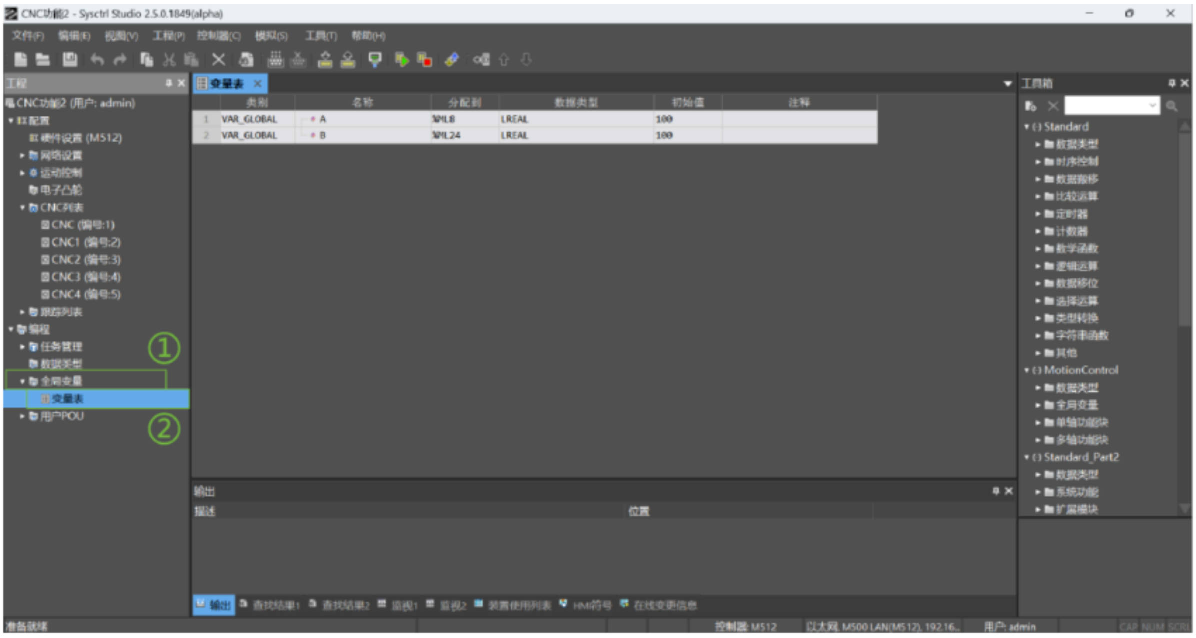
Step 1: Create a CNC list

Right-click on the 【CNC List】 on the left side of the software, select "New", and a "New CNC" window will pop up. Click "OK" to create a CNC list with the number 5.



② Define variables of the corresponding data type

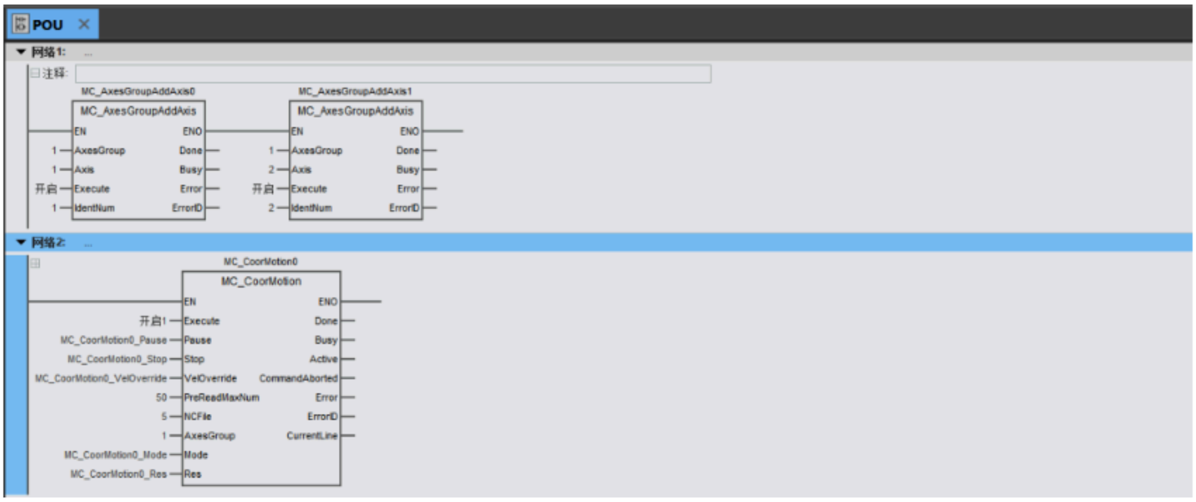
Access the variable table interface via 【Global Variables】→Variable Table on the left side of the software, define variables with corresponding data types , such as the LREAL data type for parameters after X and Y, then assign them to the devices used by the CNC code and set their initial values.



③ Write motion control programs

● Create a shaft group (shaft group enable is not required)

● Execute G-Code program files


POU
网络1:
注释:
MC_AxesGroupAddAxis0
MC_AxesGroupAddAxis
EN
ENO
AxesGroup
Done
Axis
Busy
Execute
Error
IdentNum
ErrorID
开启
MC_AxesGroupAddAxis1
MC_AxesGroupAddAxis
网络2:
MC_CoorMotion0
MC_CoorMotion
开启1
Execute
Pause
Stop
VelOverride
PreReadMaxNum
NCFile
AxesGroup
Mode
Res
Done
Busy
Active
CommandAborted
Error
ErrorID
CurrentLine
MC_CoorMotion0_Pause
MC_CoorMotion0_Stop
MC_CoorMotion0_VelOverride
MC_CoorMotion0_Mode
MC_CoorMotion0_Res