

# M Series Teaching (Advanced Tutorial) | CNC Functions (3) G50, G51, G52 Instructions

## Preliminary preparations

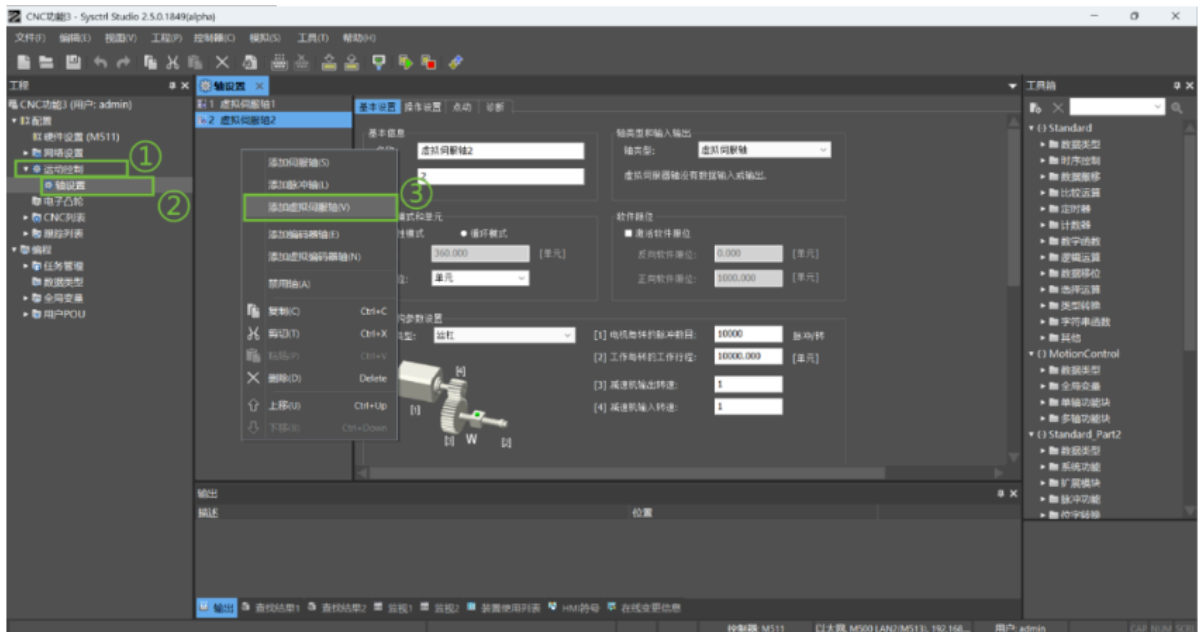
**Software** : Sysctrl Studio (available from Chuanhe Automation Academy)

**Hardware** : M-series controller (this tutorial uses the M511 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 related 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):

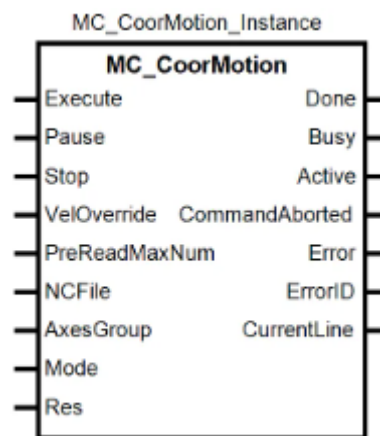
### Input

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

### Output

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

## 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

Name	Description	Data Type	Setting Range	Default Value	Explanation
Execute	Trigger	BOOL	TRUE / FALSE	FALSE	The function block runs upon detection of the rising edge of this signal
Pause	Pause Motion	BOOL	TRUE / FALSE	FALSE	Suspend all axes in the group and decelerate to stop at the CNC-defined deceleration rate
Stop	Emergency Stop	BOOL	TRUE / FALSE	FALSE	Immediately stop all axes in the group within one control cycle
VelOverride	Velocity Override Factor	LREAL	0~500	0	Global speed override percentage for CNC motion
PreReadMaxNum	Max CNC Block Pre-read Count	UINT	1~50	No default value	Maximum number of CNC program lines preloaded into buffer
NCFile	CNC Program Index	UINT	1~64	No default value	Index number of the target CNC program file
AxesGroup	Axis Group Index	USINT	1~8	No default value	Serial number of the target motion axis group
Mode	Operation Mode	INT	Reserved	/	Reserved for future function expansion
Res	Reserved	LREAL	Reserved	/	Reserved register for future use

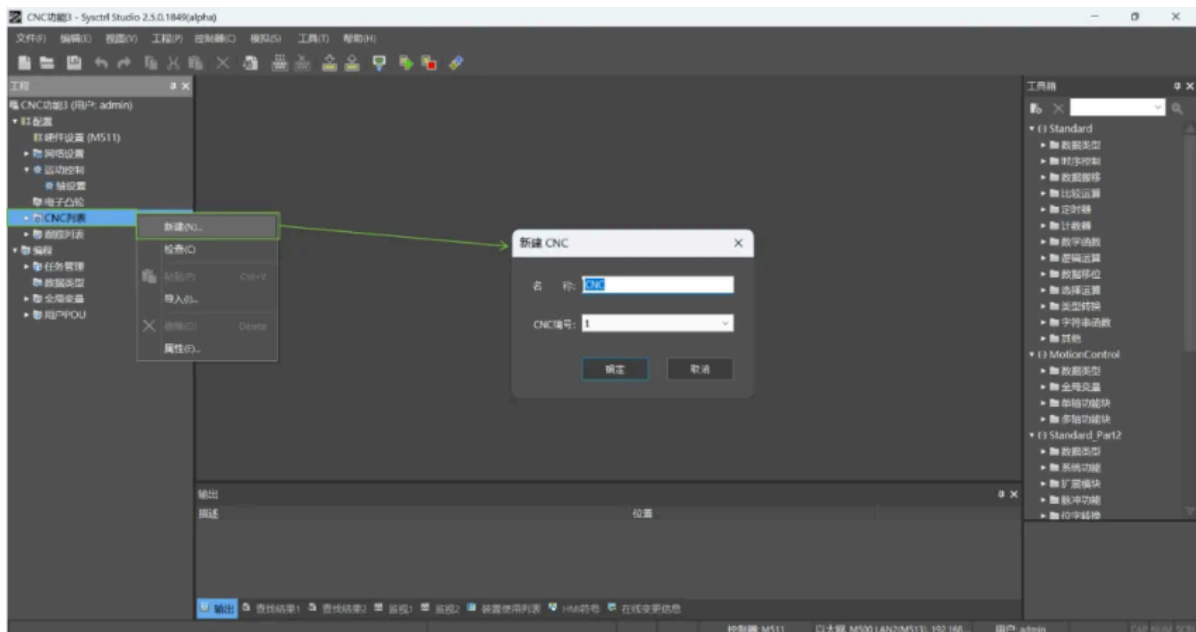
### Output

Name	Description	Data Type	Output Range	Function Description
Done	Execution Completed	BOOL	TRUE / FALSE	Turns TRUE after all CNC program blocks finish running
Busy	Program Running	BOOL	TRUE / FALSE	Remains TRUE while the CNC program is being processed
Active	Axis Under Motion Control	BOOL	TRUE / FALSE	Turns TRUE when the function block is driving axis motion
CommandAborted	Motion Aborted	BOOL	TRUE / FALSE	Turns TRUE if the CNC program is interrupted by Pause or Stop signal
Error	Fault Flag	BOOL	TRUE / FALSE	Turns TRUE when any abnormal error occurs during operation
ErrorID	Fault Code	WORD	0~65535	Output the corresponding fault code when an error is triggered
CurrentLine	Active CNC Line Number	UDINT	0~65535	Line number of the CNC block currently being executed

## G50 Precise Path

### Step 1: Create a CNC list

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



## Step 2: Writing the program

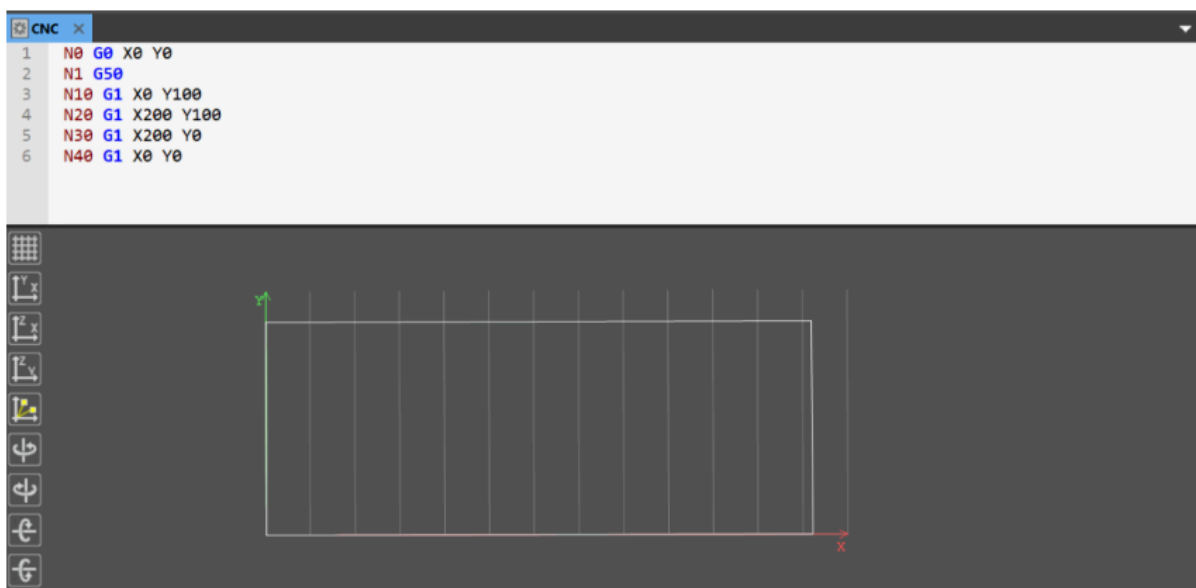
### ① Write G-Code program files

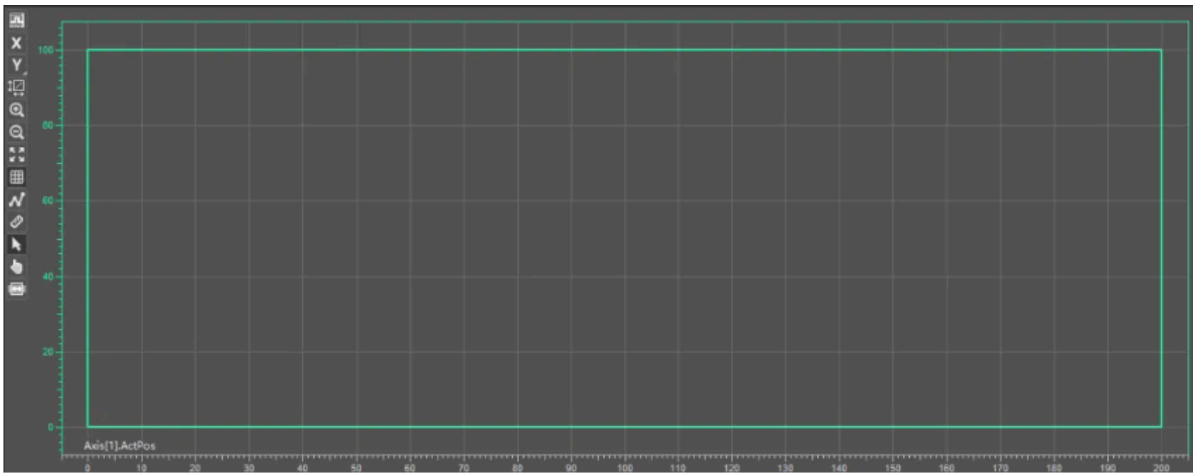
example:

```
N0 G0 X0 Y0
N1 G50
N10 G1 X0 Y100
N20 G1 X200 Y100
N30 G1 X200 Y0
N40 G1 X0 Y0
```

This code is used to set the transition mode between two adjacent CNC codes. The function of this code is to indicate that when the position of the previous line of CNC code is reached, the speed is also reduced to 0, and then the execution of the next line of CNC code is started.

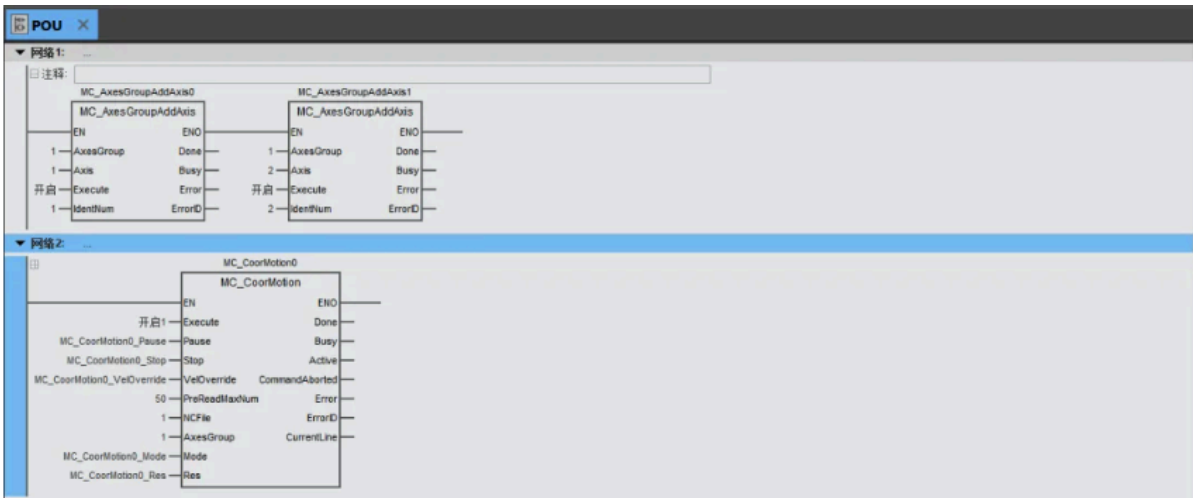
○ The default transition mode between two CNC codes is G50, which can be switched using G51 and G52 codes. All transitions between G codes can use G50.





## ② Write motion control programs

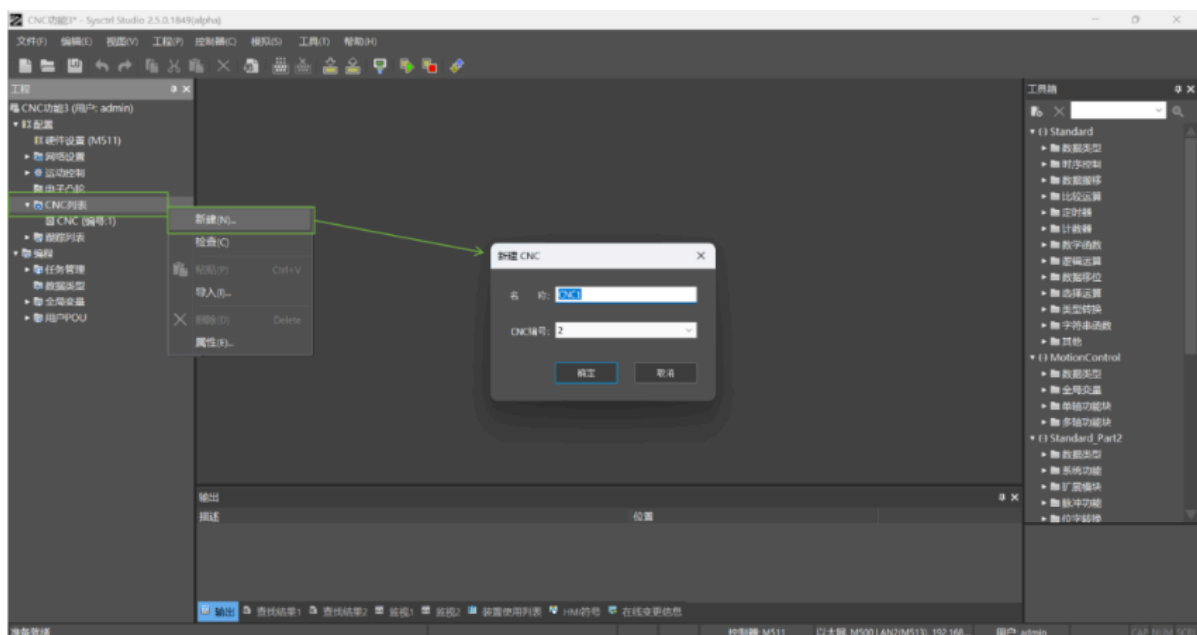
- Create a shaft group (shaft group enable is not required)
- Execute G-Code program file



## G51 Rounded transition

### 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 2.



## Step 2: Writing the program

### ① Write G-Code program files

example:

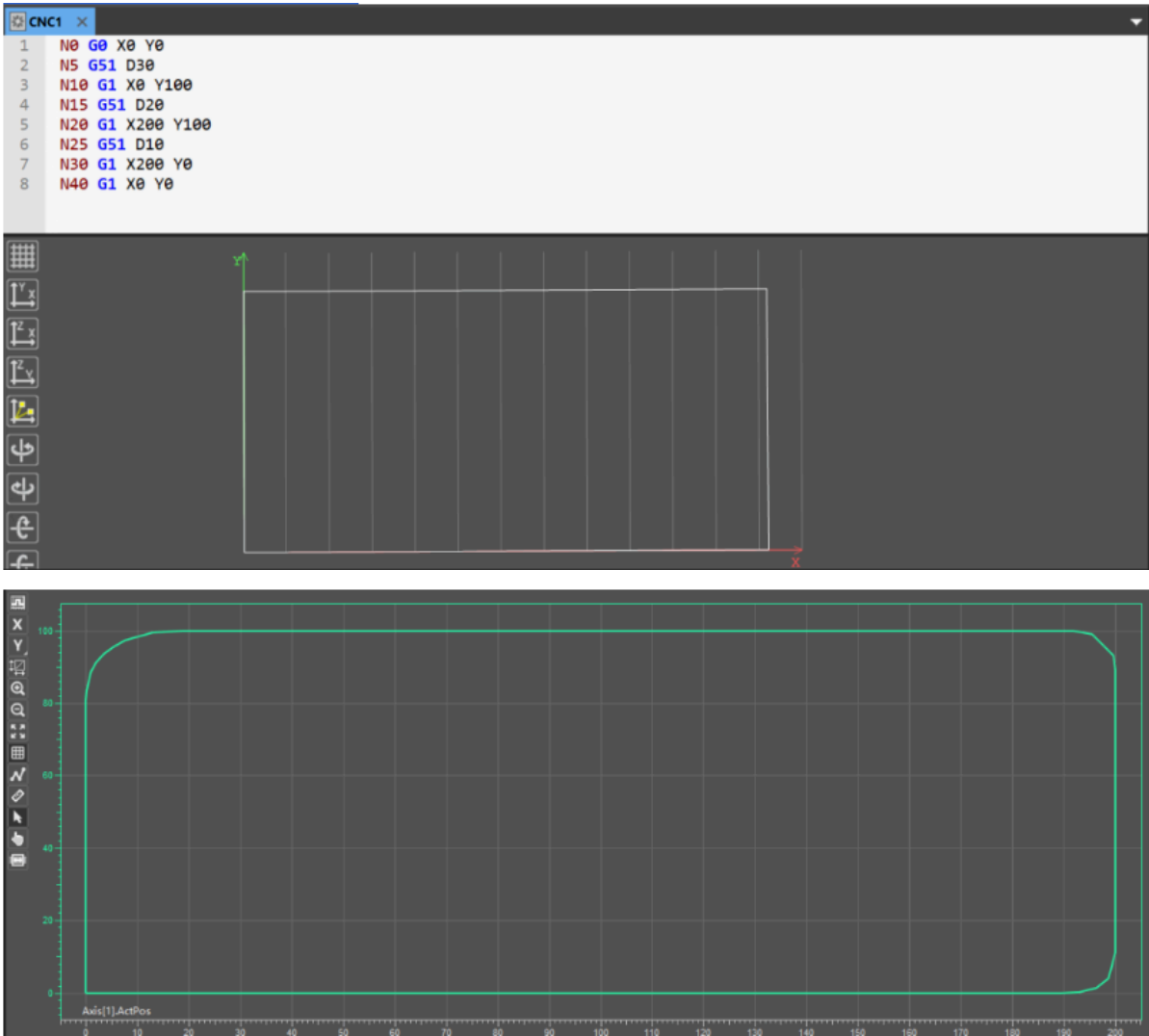
```

N0 G0 X0 Y0
N5 G51 D30
N10 G1 X0 Y100
N15 G51 D20
N20 G1 X200 Y100
N25 G51 D10
N30 G1 X200 Y0
N40 G1 X0 Y0
  
```

This code is used to set the transition mode between two adjacent CNC codes. The function of this code is to execute the next CNC code when the previous CNC code reaches the end position without the speed dropping to 0. The transition between the end position of the previous line and the start position of the next CNC code is an arc, and the interpolation speed does not drop to 0 during the transition.

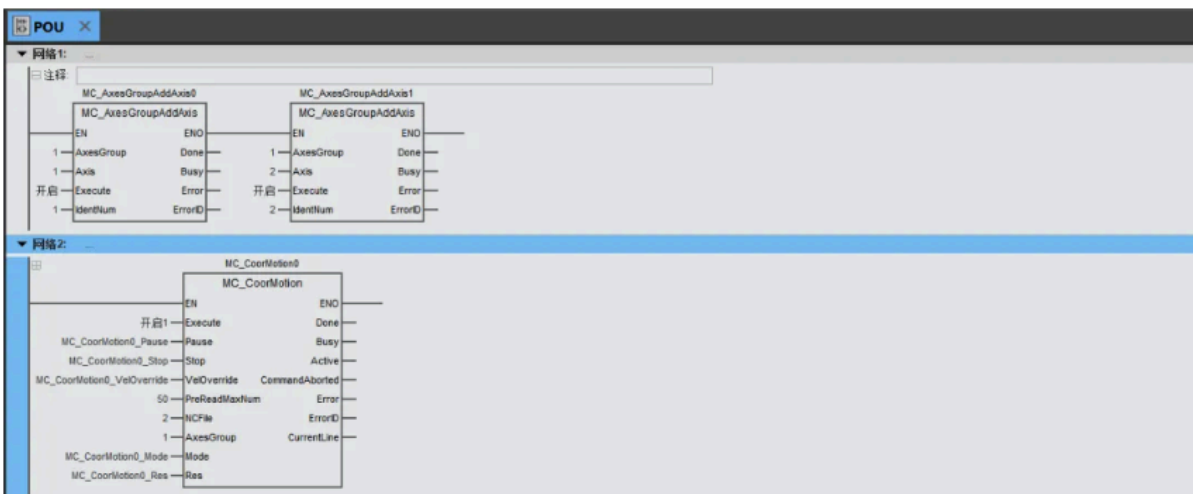
The transition starts at the end position of the previous line of G code minus the transition distance specified in that code, and ends at the end position of the previous line of G code plus the transition distance specified in that code.

- The speed of G-codes is based on the composition speed specified in the first G-code following G51. Subsequent G-codes, even if their composition speed is set via F, will still use the speed specified in the first G-code following G51 if G51 is not used again. To change the speed during G-code execution, the value of the MC\_CoorMotion directive input variable VelOverride can be modified.



## ② Write motion control programs

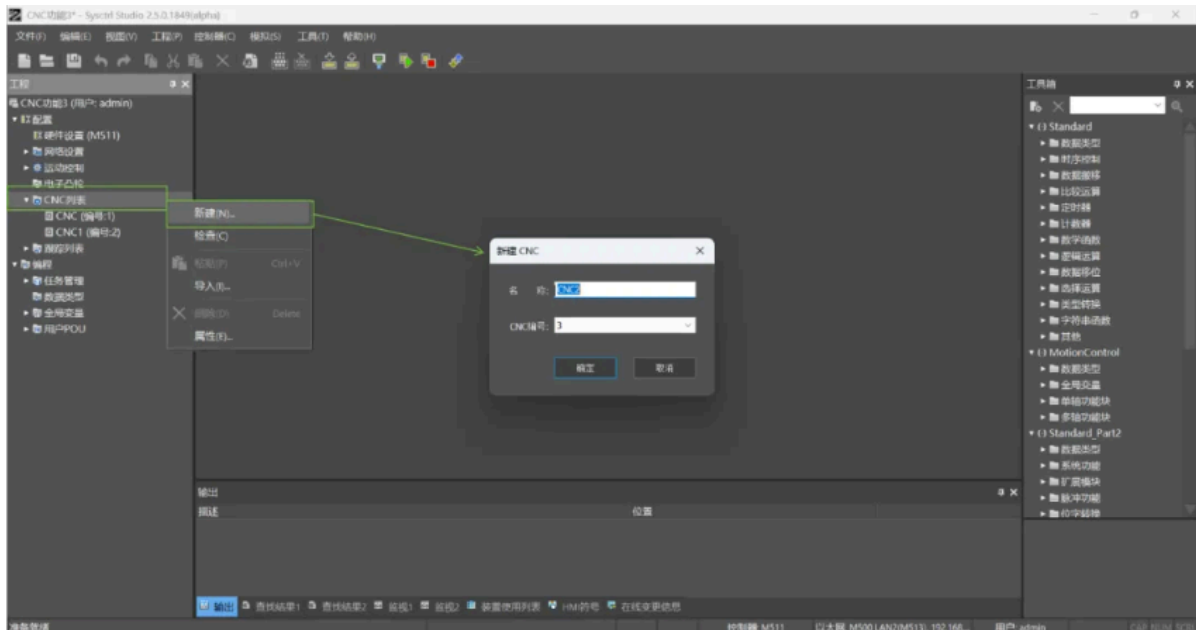
- Create a shaft group (shaft group enable is not required)
- Execute G-Code program files



# G52 Smooth transition

## Step 1: Create a CNC list

Right-click on the 【CNC List】 on the left side of the software, select "New", and the "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:

```
N0 G0 X0 Y0  
  
N1 G52  
  
N10 G1 X0 Y100  
  
N20 G1 X200 Y100  
  
N30 G1 X200 Y0  
  
N40 G1 X0 Y0
```

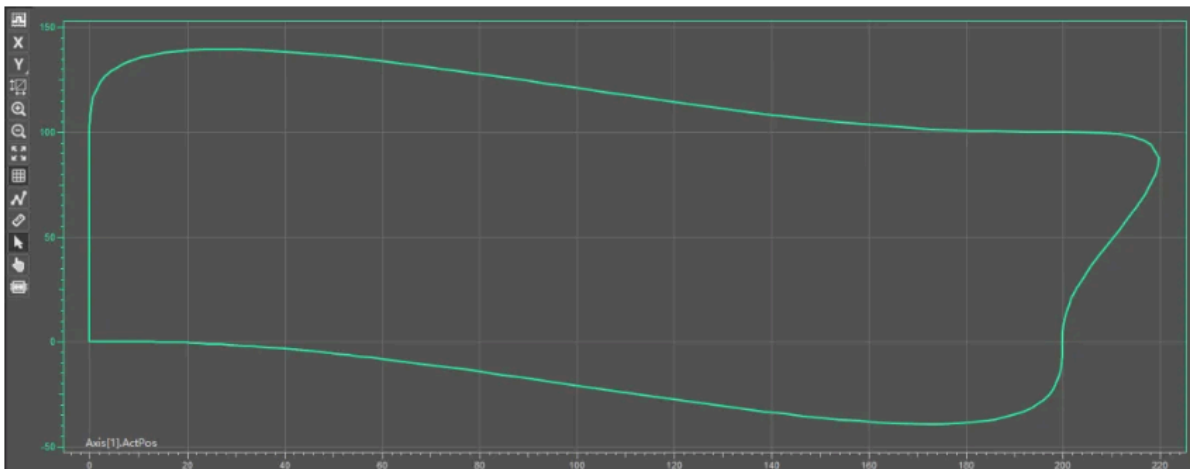
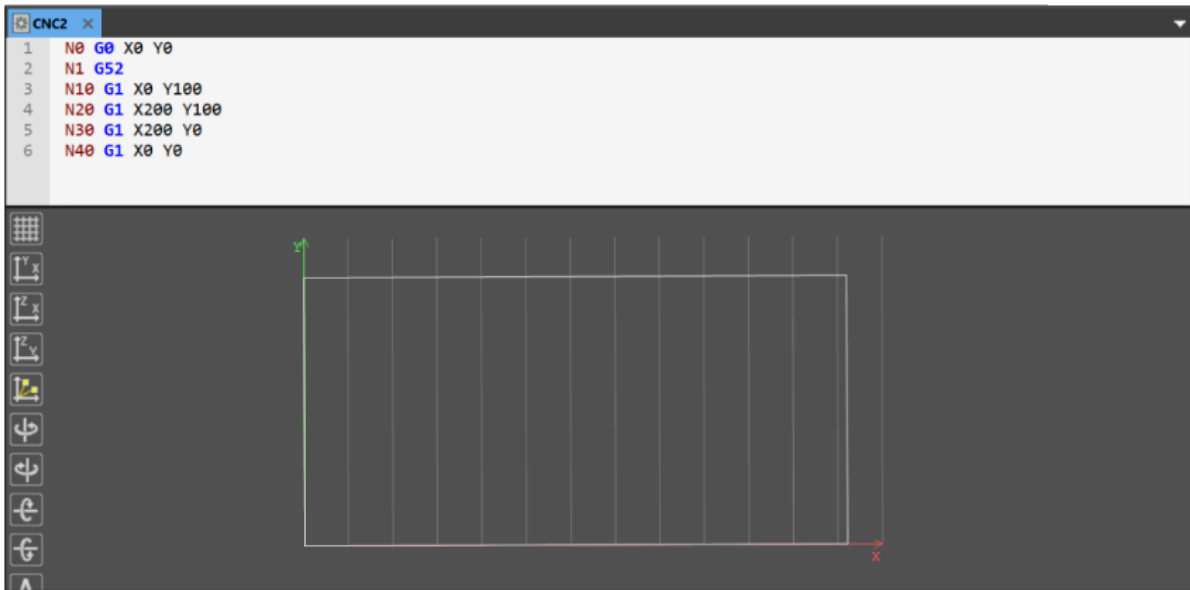
This code is used to set the transition mode between two adjacent CNC codes. The function of this code is to execute the next CNC code when the previous CNC code reaches the end position without the speed dropping to 0, and to transition at the set speed (the synthesis speed of the previous CNC code).

○ When executing only G1, G2, and G3 codes, the transition mode can be specified as G52; other G codes cannot be specified as G52.

This mode is generally used when the incremental position between multiple CNC codes is small, and the position is fitted to a curve by multiple lines of CNC code.

When using the G52 transition mode between two adjacent code snippets, the composition speed of the G-code is based on the speed specified in the first G-code snippet following the G52 snippet. Subsequent G-code snippets, even if their composition speed is set via F, will still use the speed specified in the first G-code snippet following the G52 snippet. To change the speed during

G-code execution, the composition speed can be modified by changing the value of the VelOverride directive input variable in the MC\_CoorMotion command.



## ② Write motion control programs

- Create a shaft group (shaft group enable is not required)
- Execute G-Code program files

