



SYNTEC

新代科技股份有限公司

CNC Milling/Lathe interface Introduction

SYNTEC TECHNOLOGY CO.,LTD

Topics

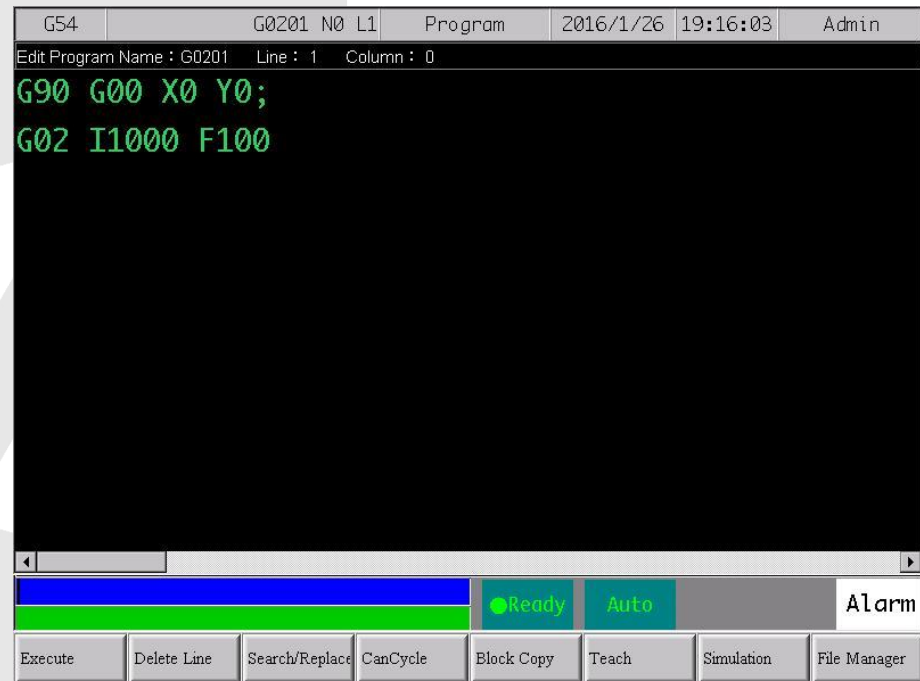
- CNC Home screen (Milling)
- Program Edit
- Simulation/Step
- Execute
- Can Cycle
- Offset setting
- Lathe G-92

CNC Home Screen (Mill)

G54	G0201 N0 L1	Coordinate	2016/1/26	19:15:46	Admin		
Machine X 0.000 Y 0.000 Z 0.000			Relative X 0.000 Y 0.000 Z 0.000 Absolute X 0.000 Y 0.000 Z 0.000 Dist. To Go X 0.000 Y 0.000 Z 0.000				
F 1000.000 mm/min 100 %	S 0 RPM 100 %						
0.0 mm/min (Actual)	0 RPM (Actual)						
Run Time 0 : 0 : 0	PartCou 0	T 0					
			● Ready	Auto	Alarm		
Coordinate	Program	Offset/Setting	Monitor	Maintain	User Param Setting	Fast Diagnostic	System Info

Program Edit

1. Press F2 in the Home screen to write a new program. Please refer App manual for examples.
2. Alternatively, one can also load or modify existing NC file from File manager.
3. Then press F7 to watch the simulation.



Program Edit

- F1/F2 to watch stepwise / continuous simulation
- F3/F4 to zoom in/reset the graph
- F5 to change settings of your simulation

G54 G0201 N0 L1 Program 2016/1/26 19:16:31 Admin

X=(2, 0) Y=(1, -1) Z=(0, 0)

Absolute

X 0.000
Y 0.000
Z 0.000

G90 G00 X0 Y0;
G02 I1000 F100

Syntax Error

Ready Auto Alarm

Step Continue Zoom Graph Reset Simu. Setting

Parameter Setting

Color

0 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15

Path 13 Cursor 14

R 255 G 0 B 255 R 255 G 255 B 0

Draw Mode XYZ

Simu Mode XYZ

View Angle

Vertical 0.000 Horizontal 0.000

Scope

X Min. 0.000 X Max. 0.000

Y Min. 0.000 Y Max. 0.000

Z Min. 0.000 Z Max. 0.000

Range

Start No. End No.

Simulation setting

The setting window adjusts these graph parameters:

- Draw Mode

Select view angles base on XYZ, XY, YZ, or XZ.

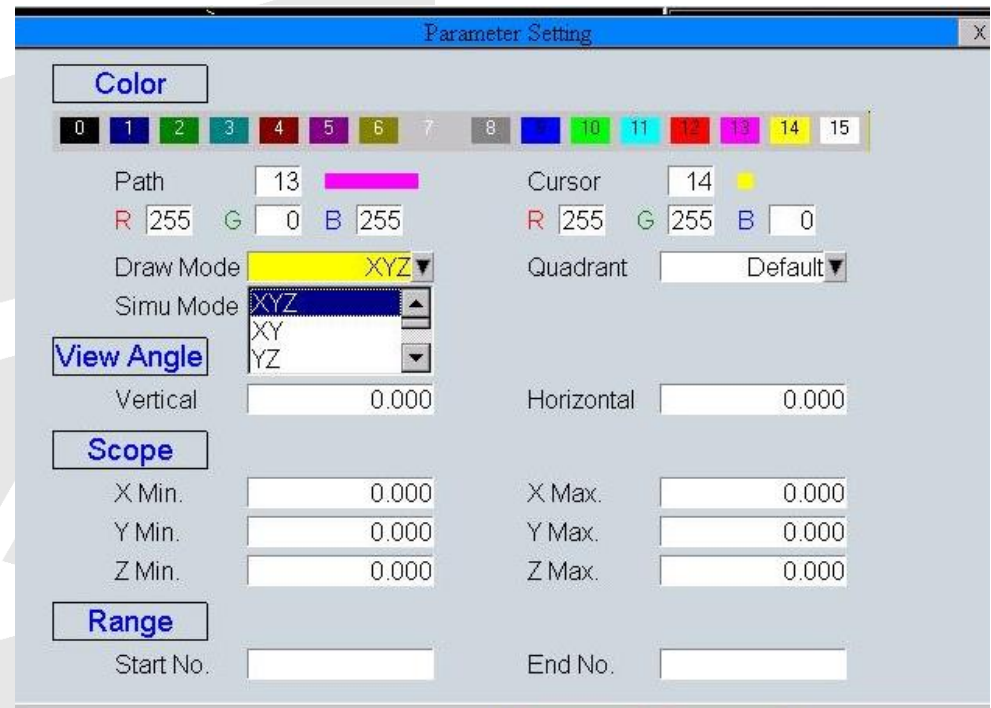
- Simu Mode

- Color

- View Angle

- Scope

- Range



Simulation Modes

Three Simulation Modes:

1. **Simulation** : When operator changes his main screen to F4 "Monitor" , CNC would automatically display simulation on the screen.
2. **Direct draw** : In this mode, cursor will show up but simulation will not execute automatically. User need to define the simulation boundary first. When the machining starts, cursor will follow up as well.
3. **No Simulation** : Stops the simulation function.



Simu Mode **Simulation** ▼

Execute

1. Press F1 in the program edit screen to execute the code.
2. Press F4 for Parameter Set to switch the status window in right-top corner
3. **Start Block** : In the lower screen , we can enter value in Start Block no. to begin the code execution directly from that particular line.
4. Press F5 for Tool wear set.

The screenshot shows the CNC control interface with the following data:

G54	G0201 N0 L1	Monitor	2016/1/26	19:19:14	Admin
Absolute	Dist. To Go	G Code G1	Run Time 0 : 0 : 0	Accum 0 : 4 : 31	
X 0.000	X 0.000	G17 G90 G94	Run Time		
Y 0.000	Y 0.000	G71 G40 G49	G00 Over 100 %		
Z 0.000	Z 0.000		G01 Over 100 %		
F mm/min	S 0 %	TotalAcu 2	T 0 D 0 H 0		
0.0	0 RPM	Part Count 0	Start Block No. 1		
G90 G00 X0 Y0;		0.0 1.0 2.0			
G02 I1000 F100		-1.0 0.0 D=1.0 mm			
		Ready Auto		Alarm	
Open file to edit	Simulation Switch	MDI Input	Parameter Set	Tool Wear Set	Start MPG Coordinate
				Work Record	Clear Acum Cycle Time

Input Mode(A)bsolute (I)crement (Z)Measure

	Absolute		Length(H)	
	Geometry	Wear	Geometry	Wear
1	0.000	0.000	0.000	0.000
2	0.000	0.000	0.000	0.000
3	0.000	0.000	0.000	0.000
4	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	0.000

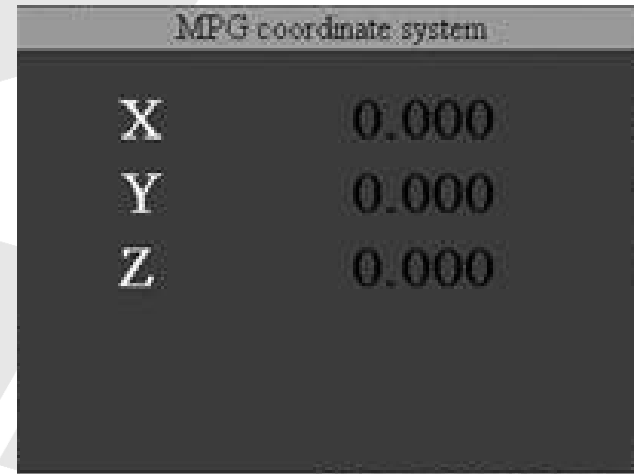
Part Count

1. We can input **Required part count number** so when the total number of part count is reached , a message window will pop up to notify user.
2. In the **Part count box**, user can see an increment each time one part is completed.
3. Also make sure parameter 3804 is set to designate a specific M code for counting.

Part Count		Part	
TotalAcumPa	<input type="text" value="0"/>	Count	<input type="text" value="0"/>
Required Par	<input type="text" value="0"/>		

MPG Coordinate

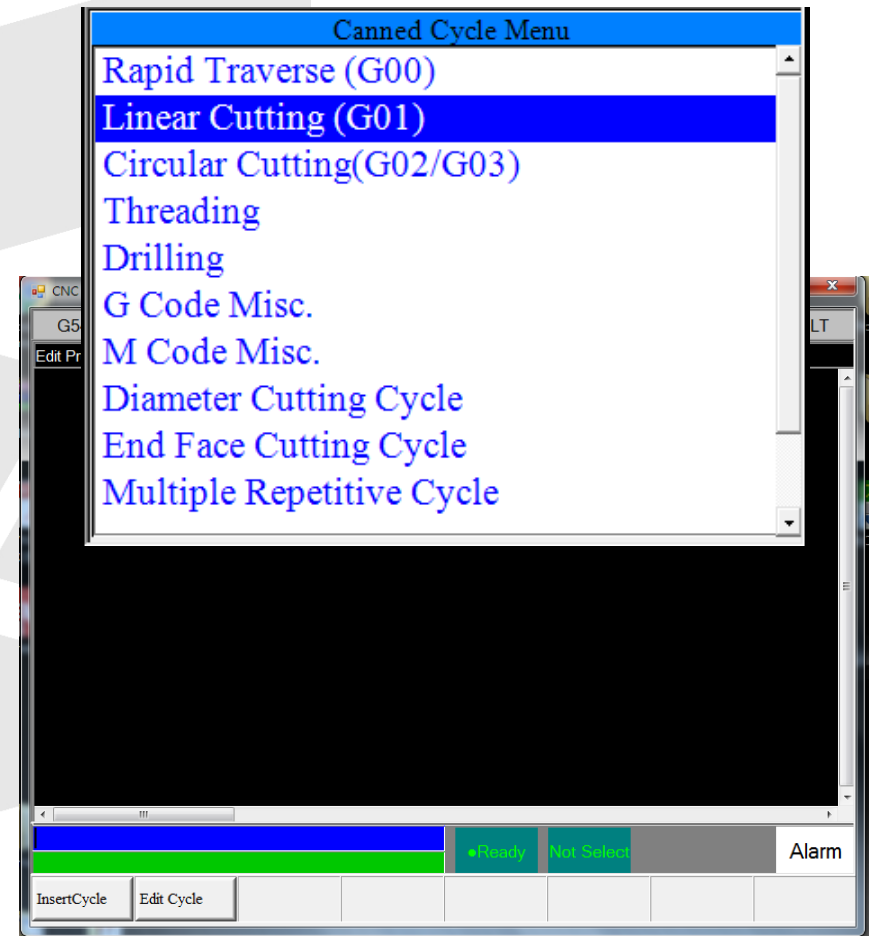
1. In Execute , **press F6 to start MPG Coordinate**
2. Modifying the coordinate system while processing.
3. In a certain block, if user inserts a displacement by hand wheel, the remaining blocks will process with this displacement vector until process finishes.
4. After pressing, current state of MPG shift coordinate system will be shown.
5. While turning the wheel, the displacement will add up into MPG shift coordinate system and program coordinate remains unchanged.



MPG coordinate system	
X	0.000
Y	0.000
Z	0.000

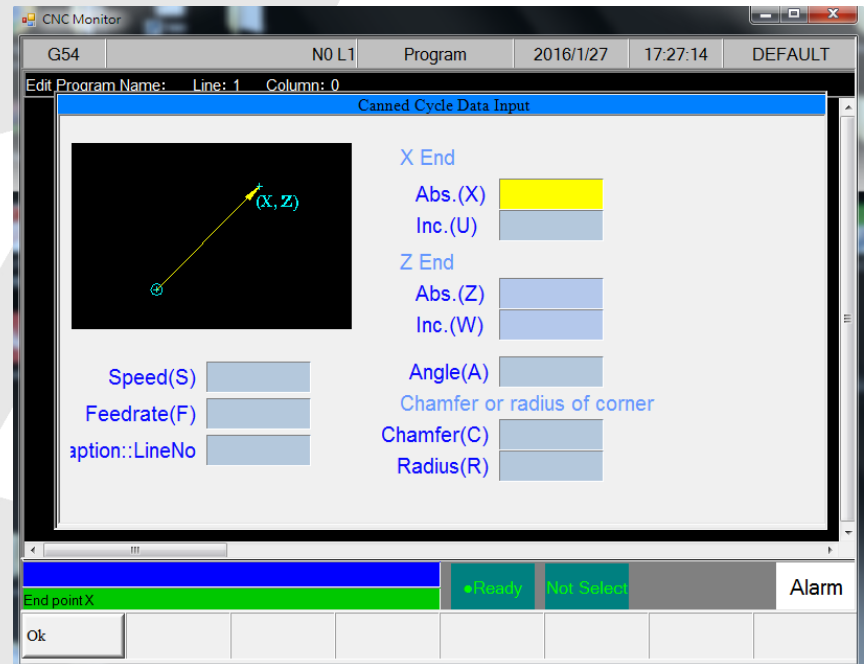
Can Cycle

1. F2 Program→F4 Can Cycle
2. To insert a required G code from list of codes.
3. Press F1 to select InsertCycle. We then choose Linear Cutting. We then choose Linear Cutting(G01)



Can Cycle

- As shown in the adjoining figure, we can add a G code for linear cutting in our existing code by just providing input parameters as shown in the adjacent figure.



Offset setting

Home → F3 Offset/Settings

- In the offset setting, we need to frequently use WorkPiece Cord. Or Tool Tip measurement
- WorkPiece Cord is used for workpiece coordinate setting purpose.
- After setting the workpiece coordinate, user need to check the tool length again.

G54	G0201 N0 L1	Coordinate	2016/1/26	19:20:03	Admin
External Shift	G54P1(G54)	G54P2(G55)	Machine		
X 0.000	X 0.000	X 0.000	X	0.000	
Y 0.000	Y 0.000	Y 0.000	Y	0.000	
Z 0.000	Z 0.000	Z 0.000	Z	0.000	
MPG Shift	G54P3(G56)	G54P4(G57)	Relative		
X 0.000	X 0.000	X 0.000	X	0.000	
Y 0.000	Y 0.000	Y 0.000	Y	0.000	
Z 0.000	Z 0.000	Z 0.000	Z	0.000	
			Aux. Coord		
			X	0.000	
			Y	0.000	
			Z	0.000	
			Ready Auto Alarm		
WorkPiece Cord.	Tool Set	Tool Tip Measure	User Param Setting		

Workpiece Coordinate

1. Press F1 in offset setting screen and press F1 again to choose **Mach. Cord. Teach**
2. Set current mechanical coordinate value into work piece coordinate
3. For eg. say current mechanical coordinate of X axis is 5.000 and current value of X axis of G54 is 0.000
4. Move the cursor to G54 X axis, press 「Mach. Coord. Teach」, the value of X axis of G54 becomes 5.000

G54		N0 L1		Coord.		2016/1/29		15:52:48		DEFAULT	
External Shift		G54P1(G54)		G54P2(G55)		Machine					
X	0.000	X	0.000	X	0.000	X	0.000	Y	0.000	Z	0.000
Y	0.000	Y	0.000	Y	0.000	Y	0.000	Z	0.000	C	0.000
Z	0.000	Z	0.000	Z	0.000	Z	0.000				
C	0.000	C	0.000	C	0.000	C	0.000				
MPG Shift		G54P3(G56)		G54P4(G57)		Relative					
X	0.000	X	0.000	X	0.000	X	0.000	Y	0.000	Z	0.000
Y	0.000	Y	0.000	Y	0.000	Y	0.000	Z	0.000	C	0.000
Z	0.000	Z	0.000	Z	0.000	Z	0.000				
C	0.000	C	0.000	C	0.000	C	0.000				
						Aux. Coord					
						X		0.000			
						Y		0.000			
						Z		0.000			
						C		0.000			
						●Ready		Auto		Alarm	
Apply Mach. Coord.		Apply Rel. Coord.		Apply Aux. Coord.		Inc. Input		Middle Func.		Apply Mach. Coord. Inc.	
										Next Coord. Page	

Workpiece Coordinate

1. Similarly , **Rel. Coord. Teach** sets the current relative coordinate value into work piece coordinate.
2. **Aux. Coord. Teach** sets the current cursor located work piece coordinate value as the aux. coordinate value. Aux. value appears after using middle function.
3. **Inc. Input** adds the work piece coordinate value with the manual input value and restores into work piece coordinate again.

G54		N0 L1		Coord.		2016/1/29		15:52:48		DEFAULT		
External Shift				G54P1(G54)		G54P2(G55)		Machine				
X	0.000	X	0.000	X	0.000	X	0.000	X	0.000	Y	0.000	
Y	0.000	Y	0.000	Y	0.000	Y	0.000	Y	0.000	Z	0.000	
Z	0.000	Z	0.000	Z	0.000	Z	0.000	Z	0.000	C	0.000	
C	0.000	C	0.000	C	0.000	C	0.000	C	0.000	Relative		
MPG Shift				G54P3(G56)		G54P4(G57)		X	0.000	Y	0.000	
X	0.000	X	0.000	X	0.000	X	0.000	Y	0.000	Z	0.000	
Y	0.000	Y	0.000	Y	0.000	Y	0.000	Z	0.000	C	0.000	
Z	0.000	Z	0.000	Z	0.000	Z	0.000	Aux. Coord				
C	0.000	C	0.000	C	0.000	C	0.000	X	0.000	Y	0.000	
								Z	0.000			
								Ready		Auto		Alarm
Apply Mach. Coord.	Apply Rel. Coord.	Apply Aux. Coord.	Inc. Input	Middle Func.	Apply Mach. Coord. Inc.			Next Coord. Page				

Middle Function

1. In the workpiece coordinate , press F5 to choose Middle func.
2. This function can help to correlate the middle point of the work piece.
3. User controls the machine by MPG, and then moves the tip of 3D machine to the X&Y end point of the workpiece.
4. This system will calculate the center point, and save to Aux. Coord
5. Return to WorkPiece Coord. To teach Aux. Coord. to G54.

G54 | G0201 N0 L1 | Coordinate | 2016/1/26 | 19:24:05 | Admin

Manual Center **0** (0:Manual, 1:Auto) **Machine Coord**

0 (0:4Pts, 1:3Pts)

Px1	0.000	Machine	X	0.000
Px2	0.000		Y	0.000
Pxm	0.000		Z	0.000
Py1	0.000	Relative	X	0.000
Py2	0.000		Y	0.000
Pym	0.000		Z	0.000

Manual Center Operation SI

Step1 : take the tool to the point Px1 and press Px1 to set

Step2 : take the tool to the point Px2 and press Px2 to set

Step3 : take the tool to the point Py1 and press Py1 to set

Step4 : take the tool to the point Py2 and press Py2 to set

Step5 : back to Workpiece screen to do Auxiliary point set

Aux. Coord

X	0.000
Y	0.000
Z	0.000

Ready Auto Alarm

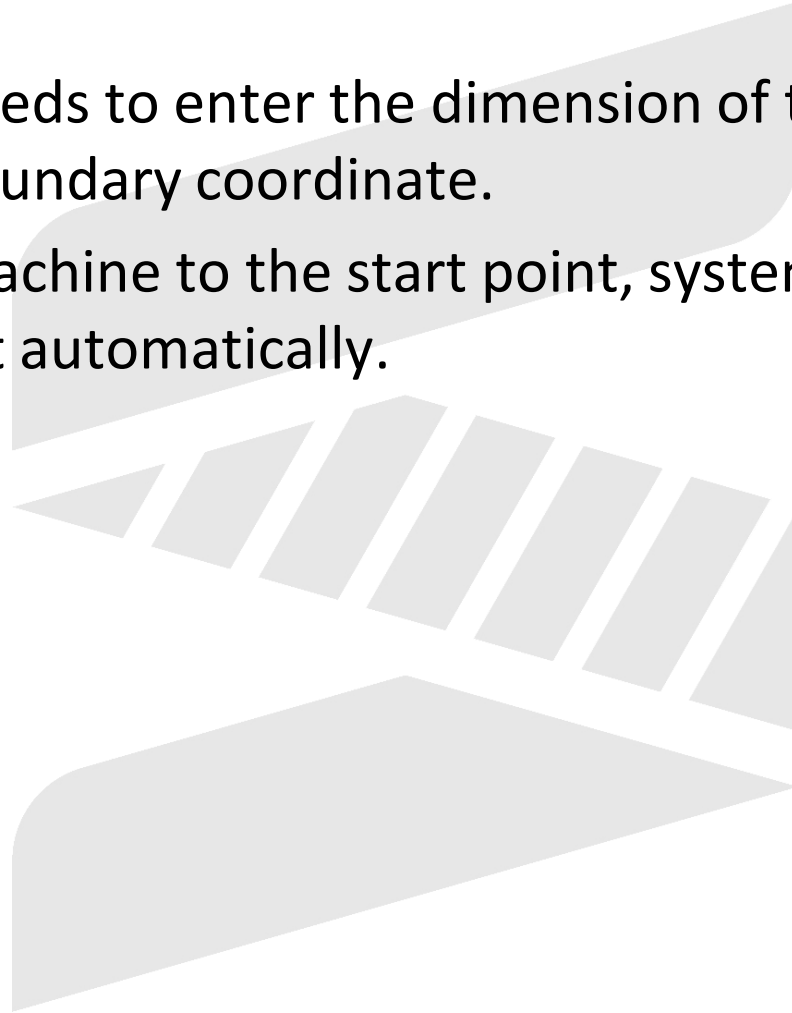
PX1 Set PX2 Set PY1 Set PY2 Set

Mid point calculation (Manual)

1. Move the machine by MPG, touching Px1 point in the previous slide figure and then press PX1 Set. Similarly set PX2.
2. The system will record the current mechanical coord. to Px1 and other to Px2.
3. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm. Similarly calculate Pym.
4. Now, the values of Pxm and Pym are the middle point of the workpiece.

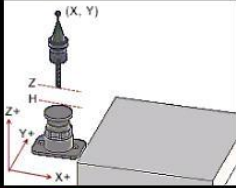
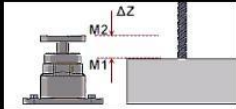
Mid point calculation (Auto)

- User only needs to enter the dimension of the workpiece and enter the boundary coordinate.
- Move the machine to the start point, system will find out the middle point automatically.



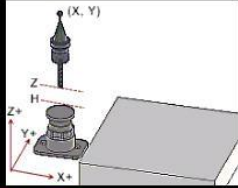
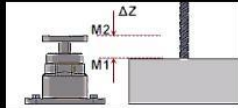
Tool Tip measure

1. We need to measure the length of the tool again when a new tool is loaded in order to compensate the correct tool length to the processing path.
2. Delta z set is mainly used to measure the distance between the surface of the work piece and the tool calibrator when a new work piece is replaced and the system will note the value to external shift.

G54	G0201 N0 L1	Coordinate	2016/1/26	19:22:44	Admin
Auto Tool Function		WorkPiece No. P	1		
AutoTool 1		Feedrate F	0.000		
1:Single tool Single workpiece		Use Reference	0		
2:Single tool Multi-workpiece		Ref Coord. X	0.000		
3:Multi-tool Multi-workpiece		Ref Coord. Y	0.000		
		Ref Coord. Z	0.000		
		Min. Z Mach. H	0.000		
		Safe Z After Measure	0.000		
		Select if use Ref Point			
		1:Set All measure parameter			
		2:If not use Ref, Take tool tip to upper of measurement			
		3:Press F1, Measure Start			
Delta Z Set		Delta Z Set	0.000		
		Do tool tip measure before do Delta Z			
		1:Take tool tip to top of good			
		2:Press F3, Delta Z Set			
		Gauge Air Blow: ON: M 0 OFF: M 0			
			Machine	X	0.000
				Y	0.000
				Z	0.000
			Relative	X	0.000
				Y	0.000
				Z	0.000
			Aux. Coord	X	0.000
				Y	0.000
				Z	0.000
			Ready	Auto	Alarm
[0-1] 0:Not use Ref, 1:use Ref					
Start	Reset	Z Delta Set	XY Ref Coord. Teach	Z Mach. Coord. Teach	

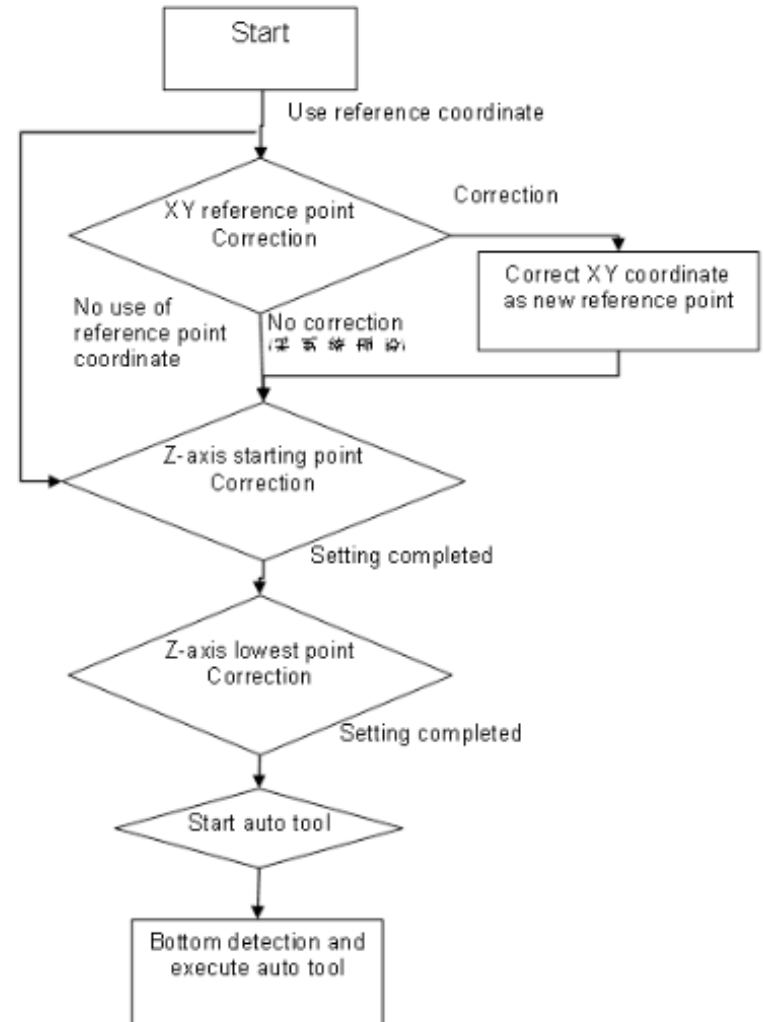
Tool Tip measure

1. For **Single tool Single workpiece**, Tool length is stored in G54 while delta z is saved in the external shift column. This can be checked in Offset setting screen.
2. For **Single tool Multi-workpiece**, the tool length will be saved in external shift while delta z in G54.
3. For **Multi tool Multi-workpiece**, tool length will be stored in Tool table while delta z in G54.

G54	G0201 N0 L1	Coordinate	2016/1/26	19:22:44	Admin
Auto Tool Function		WorkPiece No. P	1	Machine	
AutoTool 1		Feedrate F	0.000	X	0.000
1:Single tool Single workpiece		Use Reference	0	Y	0.000
2:Single tool Multi-workpiece		Ref Coord. X	0.000	Z	0.000
3:Multi-tool Multi-workpiece		Ref Coord. Y	0.000	Relative	
		Ref Coord. Z	0.000	X	0.000
		Min. Z Mach. H	0.000	Y	0.000
		Safe Z After Measure	0.000	Z	0.000
		Select if use Ref Point		Aux. Coord	
		1:Set All measure parameter		X	0.000
		2:If not use Ref, Take tool tip to upper of measurement		Y	0.000
		3:Press F1, Measure Start		Z	0.000
Delta Z Set		Delta Z Set	0.000		
		Do tool tip measure before do Delta Z			
		1:Take tool tip to top of good			
		2:Press F3, Delta Z Set			
		Gauge Air Blow: ON: M 0 OFF: M 0			
(0-1) 0:Not use Ref, 1:Use Ref			● Ready	Auto	Alarm
Start	Reset	Z Delta Set	XY Ref Coord. Teach	Z Mach. Coord. Teach	

Tool tip measurement

1. Reference Cord. X : This field can first move spindle to new auto tool position coordinate and by using 【 XY Ref. Cord. teach 】 function, correct the location of the auto tool reference point in X-axis
2. Similarly for Ref Coord. Y.



Milling- Incremental coordinate input

1. If you wish to set Incremental coordinate system, input character 'I' and then press Enter
2. In this system, all of the coordinates are measured from its previous or current coordinate point.



Coordinate

Input Mode(Absolute (I)Increment (Z)Measure

Incremental

	Diameter(D)		Length(H)	
	Geometry	Wear	Geometry	Wear
1	0.001	0.000	8.500	0.000
2	0.000	0.000	0.000	0.000
3	0.000	0.000	-3.000	0.000
4	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	0.000

Knedy

Clear Z Coord Set Tool Mach. Coord Set Tool Set Coord

Milling- Absolute coordinate input

1. If you wish to set Absolute coordinate system , input character 'A' and then press Enter
2. In this system , all of the coordinates are measured from one fixed point.



Input Mode(Absolute) (Increment (Z)Measure

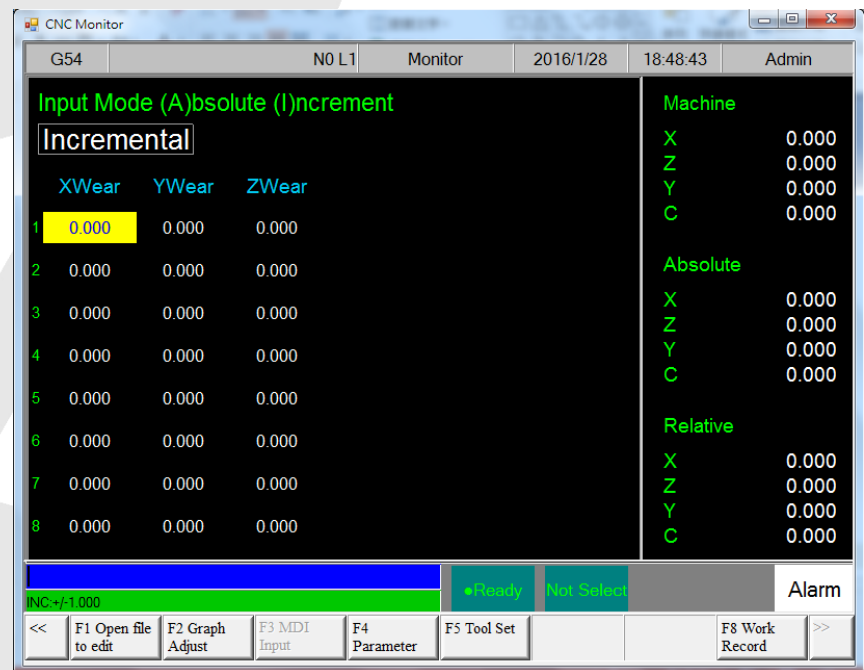
Absolute

	Diameter(D)		Length(H)	
	Geometry	Wear	Geometry	Wear
1	0.001	0.000	8.500	0.000
2	0.000	0.000	0.000	0.000
3	0.000	0.000	-3.000	0.000
4	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	0.000

Clear Z Coord Set Tool Match Coord Set Tool Rel Coord

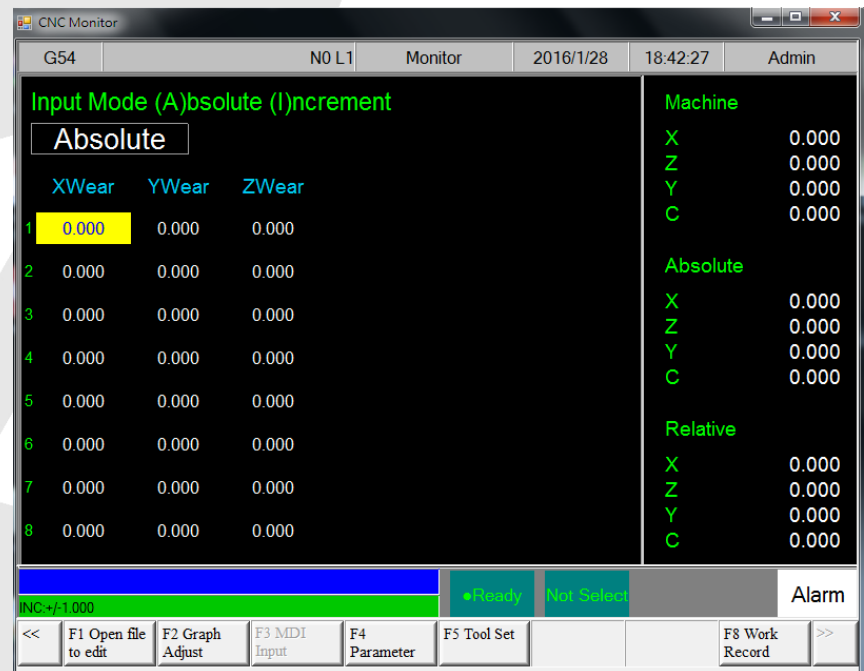
Lathe- Incremental coordinate input

1. If you wish to set Incremental coordinate system , input character 'I' and then press Enter
2. In this system , all of the coordinates are measured from its previous or current coordinate point.



Lathe- Absolute coordinate input

1. If you wish to set Absolute coordinate system , input character 'A' and then press Enter
2. In this system , all of the coordinates are measured from one fixed point.



Lathe dual Program editor

- During lathe operation , since there are two axis groups , we need to use dual program editor to edit programs for different axis groups.
- Adjoining figure displays dual program editor

The screenshot displays two side-by-side program editors. The left editor is titled 'Edit Program Name: *uuu,\$1' and shows the following code:

```
Line: 1 Column: 0
$1
S1 = 150
M103 // spindle 1 CW on
G04 X0.4 // wait spindle
G114.1 R0. // enable spindle
Mxx // wait spindle synchro
S1 = 200 // change speed.
```

The right editor is titled 'Edit Program Name: *uuu,\$2' and shows the following code:

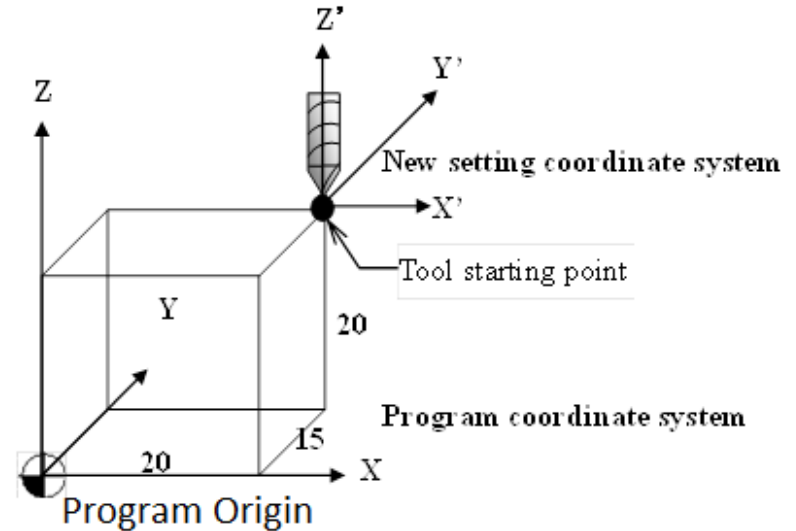
```
Line: 10 Column: 0
$2
S2 = 100
M203 // spindle 2 CW on
G04.1 P1 // wait sync. $1
M99 // end.
```

Below the editors is a control bar with a status indicator showing 'Ready' (green) and 'Not Select' (grey), and an 'Alarm' indicator. A row of function keys is located at the bottom:

<<	F1 Execute	F2 Delete Line	F3 CanCycle	F4 Select Program	F5 Half Screen	F6 SubEdit Menu	F7 Simulation	F8 File Manager	>>
----	------------	----------------	-------------	-------------------	----------------	-----------------	---------------	-----------------	----

G92 : SETTING OF WORK COORDINATE SYSTEM

1. When we design a program, we must set another program coordinate origin for the tool, we can use G92 to set a new coordinate system.
2. This command is used to set a new origin of coordinate system when the tool is at any position, after setting tool will start to perform machining from this point, absolute mode is used by this new coordinate system.





Thank you