INCON CNC Series

INCON-ME3 CNC Controller Guide for Milling

1.01 edition〈suitable for Version 1.00D and upward〉 2009/07/07.

INTEK Technology Co., LTD.
No.23, Alley 24, Lane 711, Bo-ai St., Chu-Pei City 30265, Hsin-chu County ,Taiwan
TEL：886-3-555-5623
FAX：886-3-555-5604
E-mail：info@intek.com.tw
Website：www.intek.com.tw
Support Website：support.intek.com.tw

PREFACE

IINCON-ME3 is a new generation CNC controller of INTEK for CNC milling.

INCON-ME3 adopts the new generation electronic technology, so that it is so fine in dimension and electronic layout. We use high density IC to reduce a lot of electronic parts of INCON-ME3, and make INCON-ME3 have high steady feature.

INCON-ME3 is an economical controller, but it has complete functions for a normal milling. It has complete operation board and the interface suitable for users. For example: pull down menu, dialog box, the sensitive help, compelte cutting option, manual feed program operation, tool compensation, linear interpolation, arc interpolation, cancycles, FTP file transfer, and USB.

Use〝**Help→List System Parameters...**〞item on the pull down menu and print system parameter when you first operate a CNC milling. The list is the most important backup information for maintenance.

INTEK will take the responsibility for maintenance of the INCON-ME3 controller during warranty time. Because machine is not INTEK product, do'nt warrant the precision of cutting by the CNC milling with INCON-ME3.

INTEK provides limited technology support in a business contract. But all users can log in the message board of website: support.intek.com.tw and ask your questions, INTEK will answer you on line.

CONTENT

Chapter 1 Quick Guide

1.1 Starting and Home

The main screen will display when power applies to ME3 system if ME3 has no error and alarm.

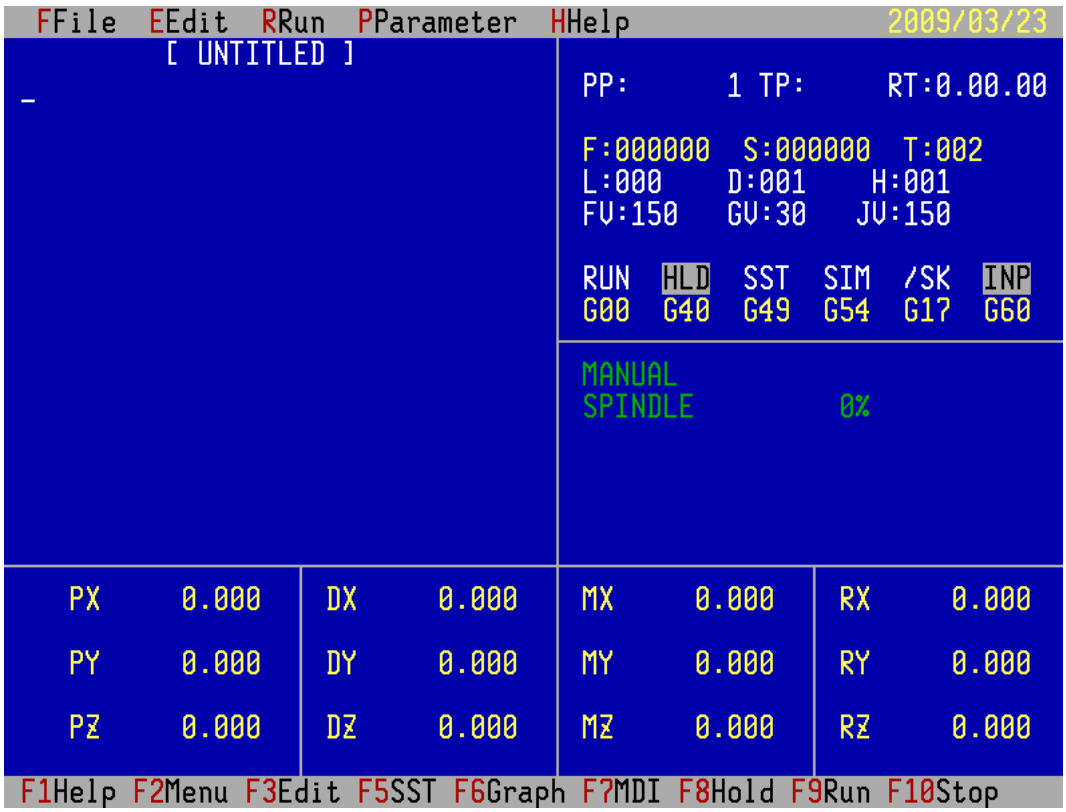


Figure 1-1 The main screen of INCON-ME3

The screen changes as below figure when switch to the Home manual mode by the manual button on 2nd operating panel, GMPU.

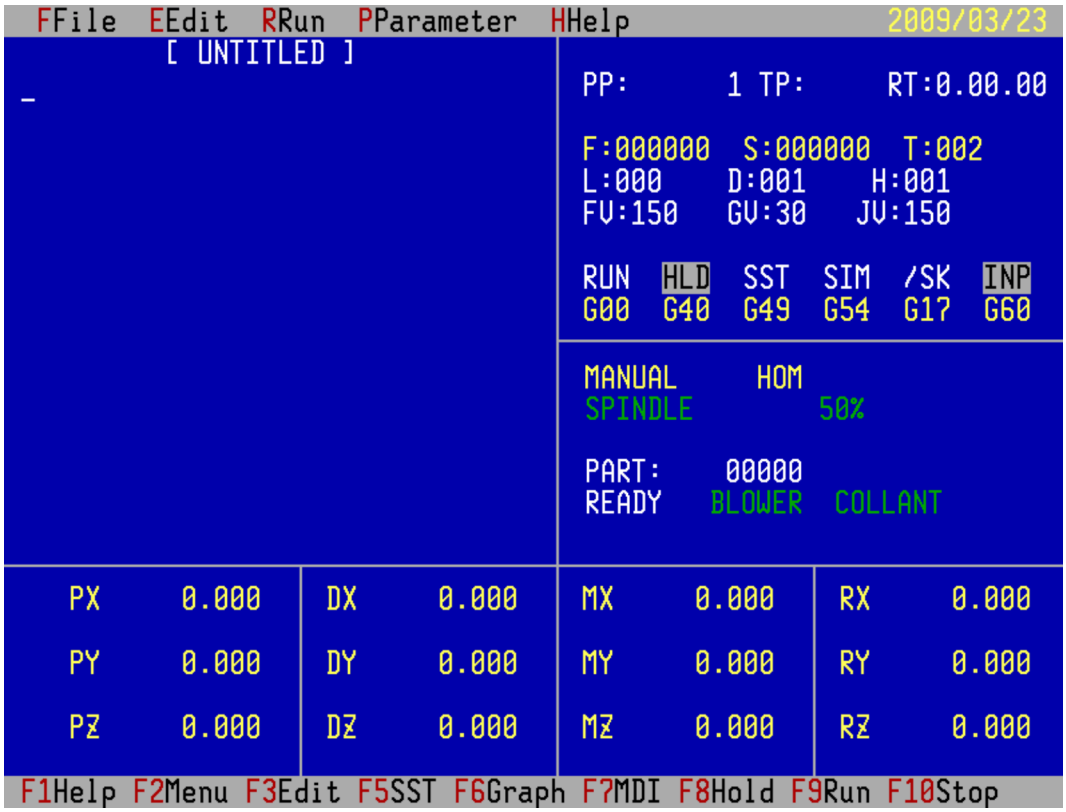


Figure 1-2 The screen of the home manual mode

Press the Home button to make axes Home. For safety, please make Z axis Home first, and then X or Y axis Home. Before Home, make sure the relative position between tool and parts won't occur collision. Press the Emergency button on the 2nd panel when occurring danger.

The machine comes to a machine position which is the same as the one when power off last after Home process.

Press the **Manual Off** button on the 2nd panel to leave the manual mode, and operate other functions further.

1.2 Edit the first program

Press the **F2** function key ,then **F** key, and select the item 〝**NEW**〞 on the **File** pull down menu after leaving the Home mode.

〝**F2→F→NEW**〞 means the operating step above in this user manual. The other operating steps of the pull down menu show through the same mode.

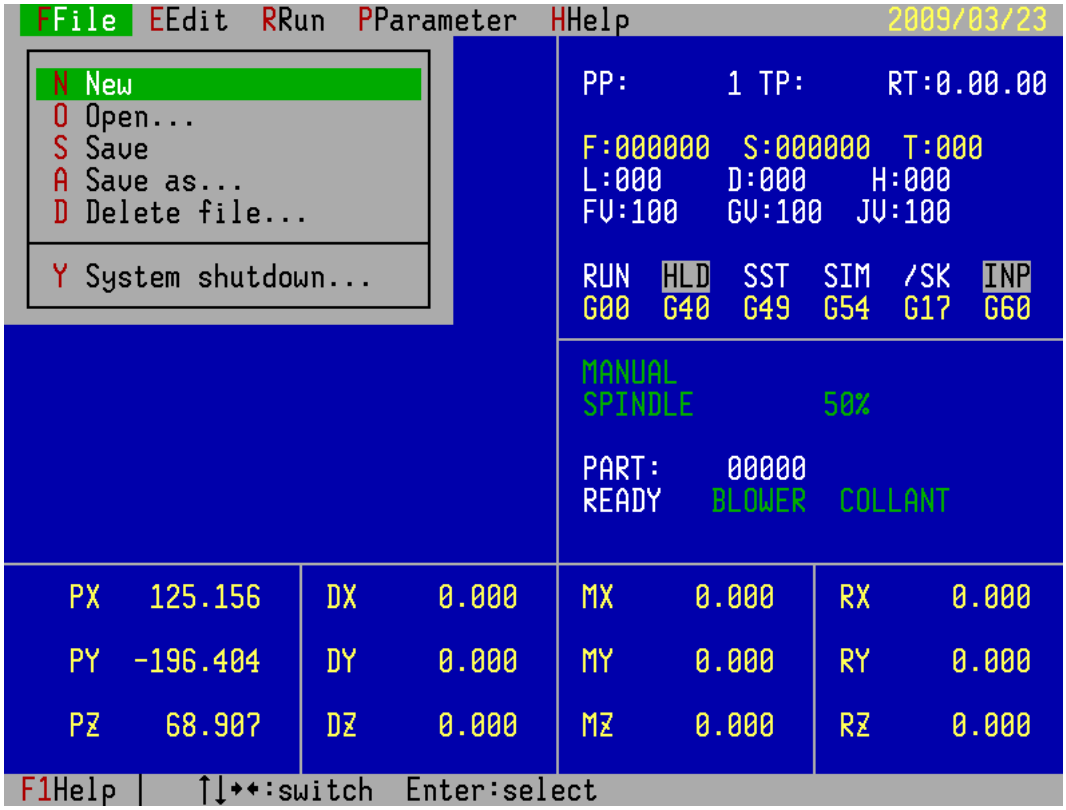


Figure 1-3 Open a new file through the File pull down menu

The empty editor screen is showing on the left half of main screen then.
And the headline "< **UNTITLED** >" is showing on the top of the editor.

The symbol "< >" means that the editor can edit programs. The figure shows below.

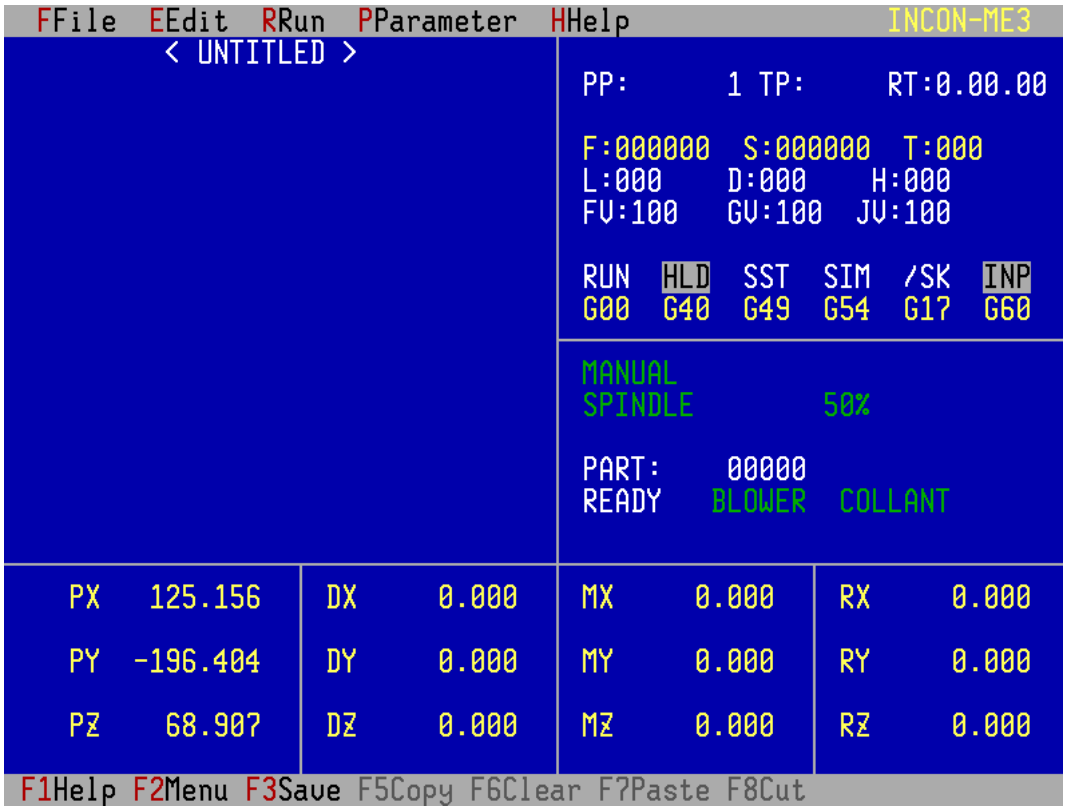


Figure 1-4 The new empty program in the editor

Input the following CNC program in the editor directly.

```
G90 G0 XY Z10.
G95
M3 S1000
G84 Z-10. F0.5 P2.
G0. X10.
M05
G94
M30
```

Press the **Esc** key to leave the **Edit** mode after inputing CNC program. Then a dialog box pops up on the screen and ask users 〝**Save untitled file?**〞 .

Figure 1-5 A dialog box: Save untitled file or not.

Please press the **Y** key and then pop up the other dialog box below to input program file name.

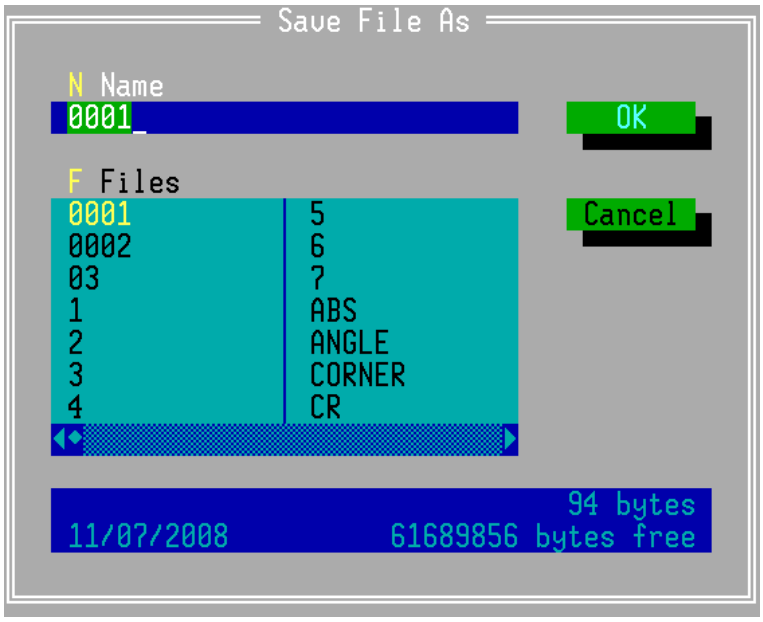


Figure 1-6 A dialog box: input file name

The first CNC program finishes after inputing file name 〝0001〞, and then the figure shows below.

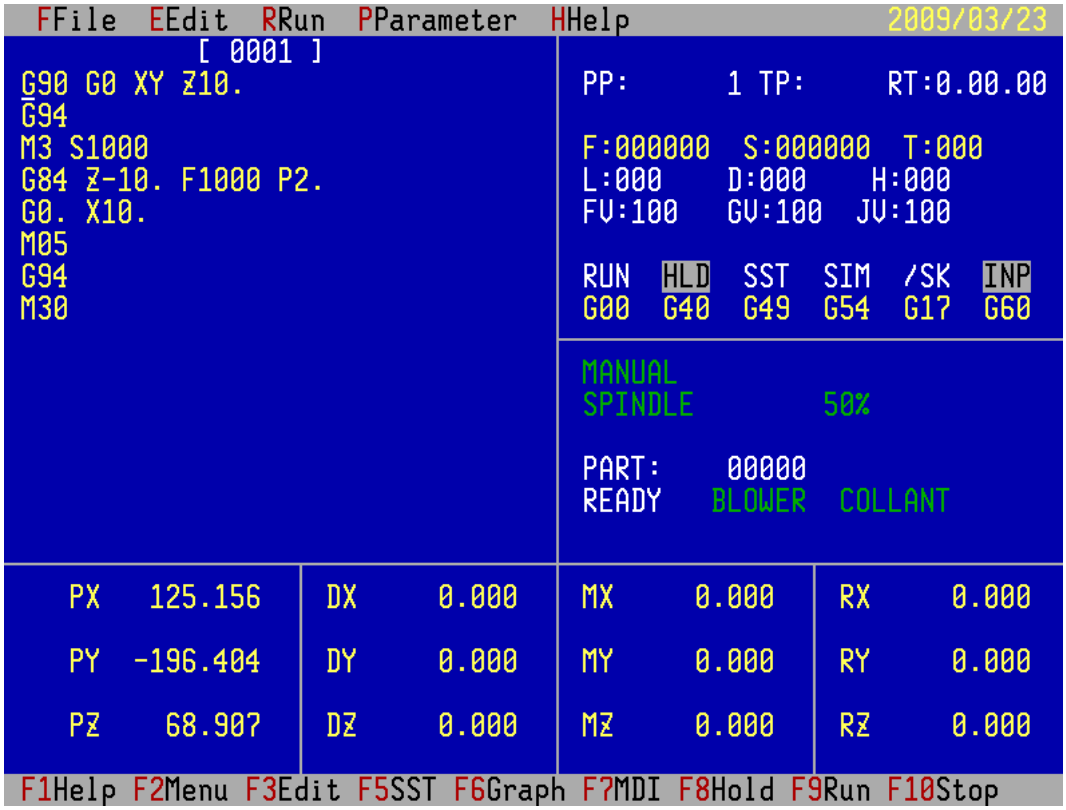


Figure 1-7 The program can execute in the **Ready** mode.

The CNC program name is 〝[0001]〞, and the symbol 〝[ ]〞 means the editor has been ready to run CNC program.

Press the **F3** key to enable editor to modify the CNC program and then the CNC program title becomes 〝< 0001 >〞 again.

1.3 Set Program Position

The machine position gets absolute position after Home process. MX, MY and MZ in the bottom screen are current machine position.
Editing the CNC program refers to part original drawing. We call it program position, PX, PY and PZ , shows on left bottom of screen.

We can execute cutting according to this CNC program correctly after adjusting the offsets of machine position and program position.

The offsets of machine position and program position are relative to the two following things:

- The fix position of a part.
- The fix position of a tool for cutting.

Fix a part and a tool first. Then switch the mode to **Manual**. Use **Jog** and Signal **Step** to shift the tool to **`PX`**、**`PY`**、**`PZ=0`** desired〔the location is the origin of X, Y and Z axis〕.

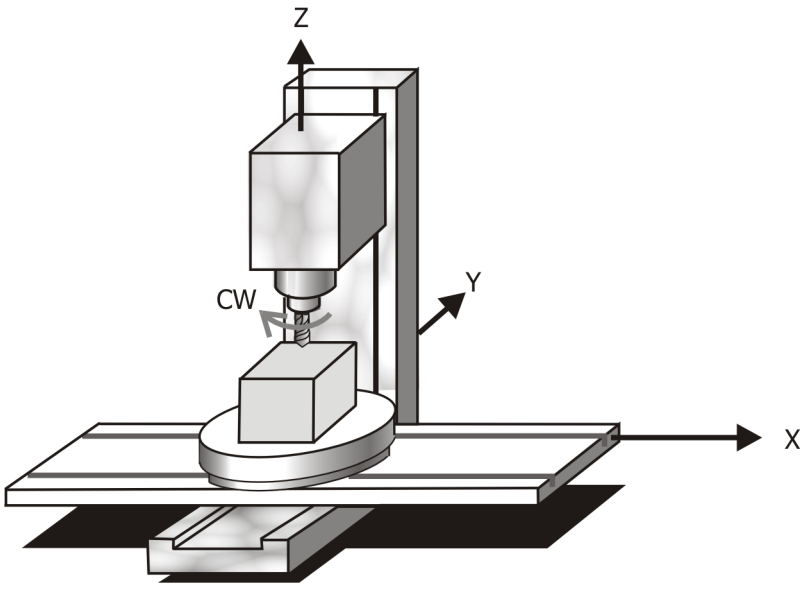


Figure 1-8 A diagram of milling

Press **ALT+X**〔**Y** or **Z**〕and input the offset into **Coordinate Offsets** automatically. Press the **Enter** key to finish the adjustment of X〔Y or Z〕axis〔Press the **ESC** key when leaving the dialog box.〕

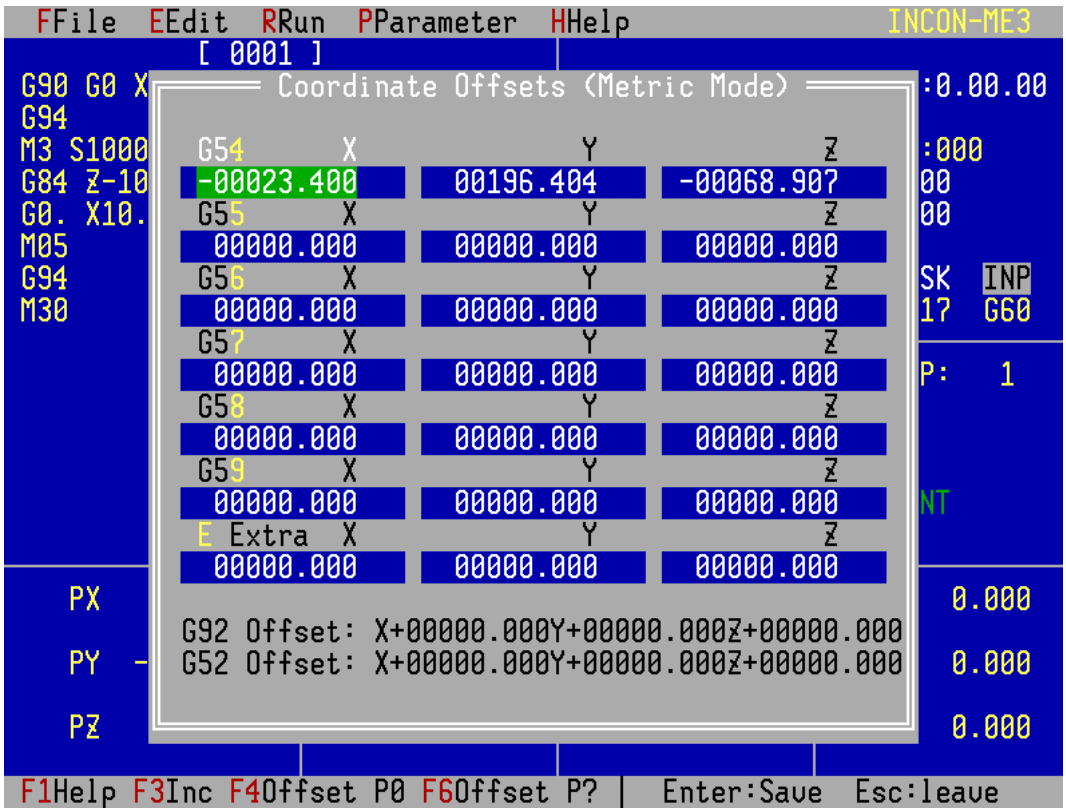


Figure 1-9 Press **ALT+X**〔**Y** or **Z**〕and input the offset into **Coordinate Offsets** automatically.

By the same way, measure a part by a vernier caliper. The tool is located at PX=18.2 when we get the value 18.2 mm. We press **Shift+X** and then pop up a dialog box to input the program position.

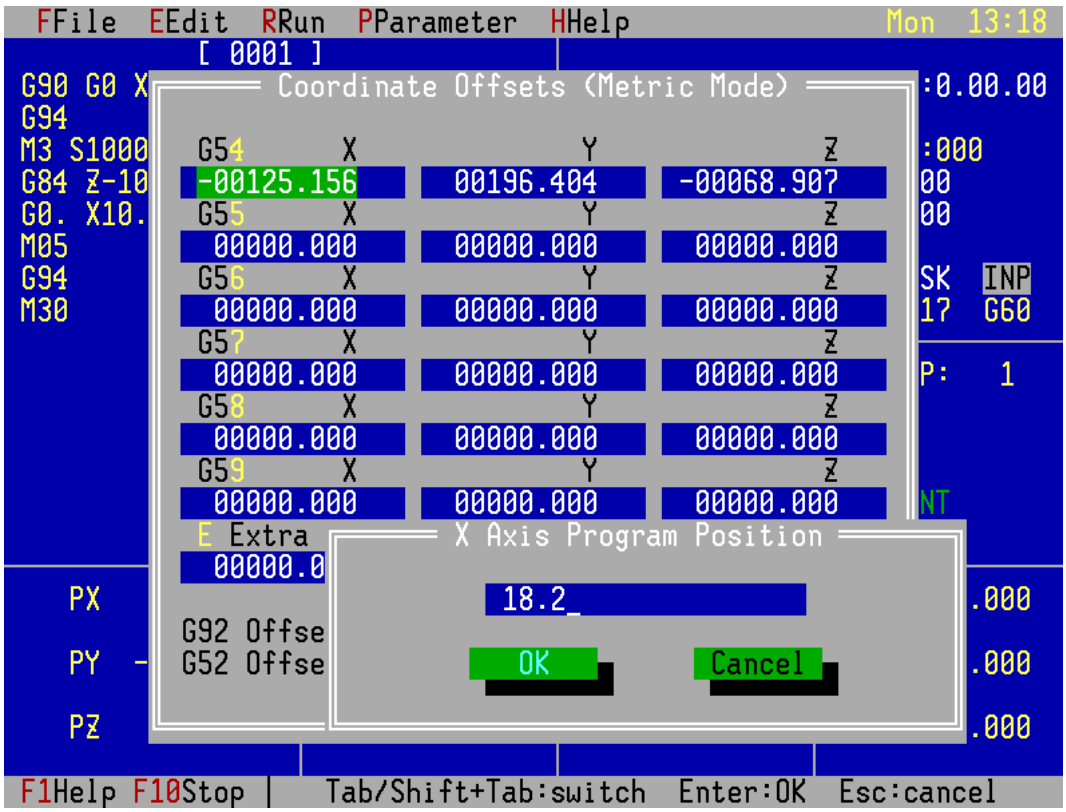


Figure 1-10 Press Shift+X and pop up a dialog box to input the program position

Input the value 18.2 and then press the **Enter** key. The controller will fill the offset an user inputs in **Coordinate Offsets** automatically. Make Sure the value is correct and press the **Enter** key to input the Offset of X axis in the controller.

That's all to finish the adjustment of X axis program position.〔〔Press the **ESC** key when leaving the dialog box.〕

1.4 Run Program

We may cut the first work piece after input CNC program and adjust the program position.

The use of millings may be dangerous. We must know the safety so well that we don't crash the parts of the millings to hurt our bodies.

The program runs for the first time, and so it is possible something happens to the program or the position. We can do that then:

- Press the function **F5** key to enable **Single Step**.
- Switch the percentage of FV, GV and spindle feed rate override to the lower one.

When everything is ready, press **Cycle Start** to run program.

As soon as a block ran the program paused by **Single Step**. Then we may check if the running program is what we expect, the spindle works or not and tool position reach where we expect, for example. Press the **F10** key to stop running program and modify program, or adjust the program position when something happens.

Cancel **Single Step** after every step running correctly. And switch the percentage of FV, GV and spindle feed rate override to the normal one and start to work.

Chapter 2 Screen and Operation

2.1 Keyboard

The ME3 controller can work in coordination with normal monitors and keyboards on the market, or optional GKYU〔one 8 inch monitor and one keyboard〕.

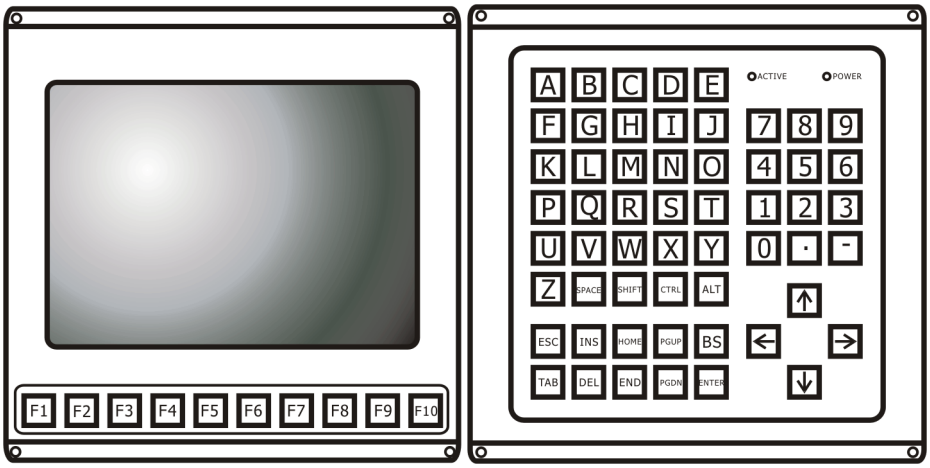


Figure 2-1 One monitor and keyboard

There are 10 function keys〔F1~F10〕below the monitor, and 56 keys on the keyboard:

26 character keys : A to Z.
12 digital keys : 0 to 9, 〝.〞 and 〝-〞.
4 arrow keys : up〔↑〕, down〔↓〕, left〔←〕and right〔→〕arrows
7 editor group keys : Ins, Del, Home, End, PgUp, PgDn and BS
3 auxiliary keys : Shift, Ctrl and Alt
4 else keys: Esc, Tab, Space and Enter

Note: Use the alcohol or cleaning naphtha to clean the panel. Don't use the strong solvent such as banana oil or toluene!

The meaning of some abbreviation is as below:

Ins : Insert	**Del** : Delete	**PgUp** : Page up
PgDn : Page down	**Ctrl** : Control	**Alt** : Alter
Esc : Escape	**Tab** : Tabulate	**BS** : Back Space

3 auxiliary keys can't be used individually, they must be used with other keys. For example, 〝**Shift+ A**〞 means press and hold the **Shift** key first and the **A** key next. 〝**Ctrl+F1**〞 and 〝**Alt+F3**〞 are both the same usage.

A Dialog box is used to set data to adjust system parameters. When modify or input data in a dialog box, press the following keys:

Alt + highlight character	: select a setting item
Tab	: switch a setting item forward
Shift + Tab	: switch a setting item backward
Esc	: leave
Enter	: save the setting

With a Check List, [ ] / [√], press **Up** or **Down arrow** key to switch to the desired sub-item, and then press **Space** key to toggle it.
With a Radio Button, ( ) / (●), press **Up** or **Down arrow** key to select the desired sub-item.

2.2 Statusline and Hot key listing

At any time, there is a statusline on the bottom of the screen, which consists of some hot keys which provide easy and quick access to system functions.

Case 1: program window while no program is running

F1：Help
F2：Pull down menu
F3：Switch editor to EDIT mode
F5：Single Step
F6：Switch to trace window
F7：MDI〔Manual Direct Input〕
F8：Hold
F9：Run〔program execution〕
F10：Emergency stop

Case 2: program window while program is running

F1：Help
F2：Pull down menu
F5：Single Step
F6：Switch to trace window
F7：MDI〔Manual Direct Input〕
F8：Hold
F9：Run〔program execution〕
F10：Emergency stop

Case 3:program window when editor is in EDIT mode

F1：Help
F2：Pull down menu
F3：Save
F5：Copy
F6：Clear
F7：Paste
F8：Cut
Esc：Back to Ready mode

Case 4: MDI -- Manual Direct Input

F1：Help
F2：Pull down menu
F8：Hold
F9：MDI continue
F10：Emergency stop
Esc：Stop MDI only

Other keys：

Alt+F8：Switch between system state window ／position state window and I／O state window.
Alt+F1 ： Help for code listing
Shift+F1： Help for topic listing
Ctrl+F1： Back to previous page

Pull down menu：

F2→F：File
F2→E：Edit
F2→R：Run program
F2→P：Parameter setting
F2→H：Help

Summary of the operation keys:

Operation Key	Function	Operation Key	Function
→	Shift cursor right	**PgUp**	Page up
←	Shift cursor left	**PgDn**	Page down
↑	Shift cursor up	**BS**	Delete a character forward
↓	Shift cursor down	**Delete**	Delete a character
Ctrl + →	Shift right a word	**Ctrl + T**	Delete a word
Ctrl + ←	Shift left a word	**Ctrl + Y**	Delete a line
Shift +↑↓→←	Block an area	**Ctrl + Delete**	Delete a block
Home	Shift cursor to line start	**Ins**	Insert Mode
End	Shift cursor to line end	**Ent**	New Line
Ctrl + Home	Shift cursor to page top		
Ctrl + End	Shift cursor to page bottom		

2.3 Screen display

There are six windows of information displayed on the ME3 screen:

1. Program Window
2. System State Window

3. I / O State Window
4. Position State Window
5. Trace Window
6. MDI Edit Window

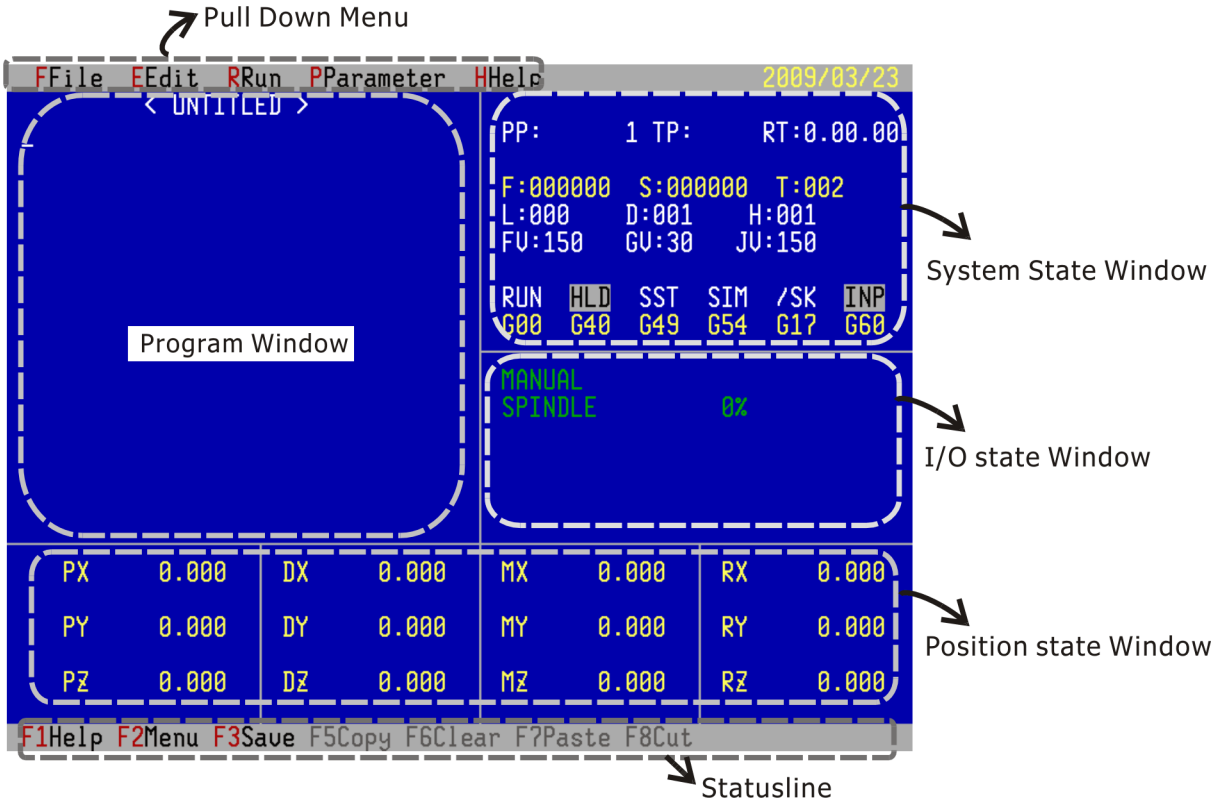


Figure 2-2 The main screen of ME3

Upon turning on the power, the main screen displays. It consists of **Pull down menu**, **Program Window**, **System State Window**, **Position State Window**, **I / O State Window** and **MDI Edit Window**, such as the Figure 2-2.

There are 5 functions of Pull Down Menu on the top of the screen : 〝**File**〞 , 〝**Edit**〞 , 〝**Run**〞 , 〝**Parameter**〞 and 〝**Help**〞 . There are 10 function keys on the bottom of the screen, **F1** to **F10**. When you press the key **F7**, the screen will pop up a **MDI Edit Window**. Press **Esc** to leave **MDI Edit Window**.

2.3.1 Program Window

Program window contains current program listing.
In this window you can:

- enter menu item 〝**File→new**〞 to create another CNC program.
- enter menu item 〝**File→open**〞 to open another existed CNC program.

We explain the variations of Program Window on every conditions, as the following figures.

Before new program doesn't save, the top of Program Window shows 〝**< UNTITLED >**〞.

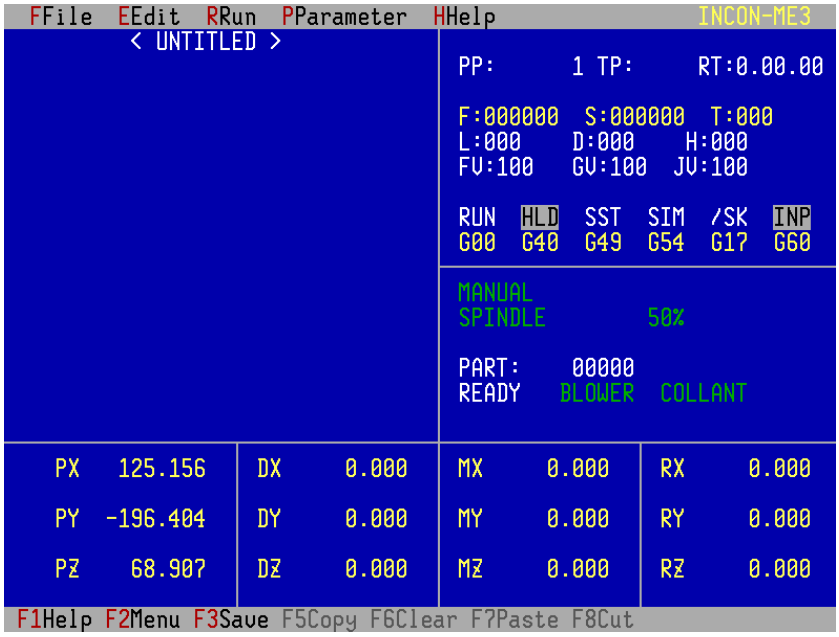


Figure 2-3 A new CNC program

The top of **Program Window** shows 〝**[filename]**〞 means **Editor** is disable and ready to run a program. The program now can run but can't edit. When you want to edit it, press **F3** to switch the mode to edit.

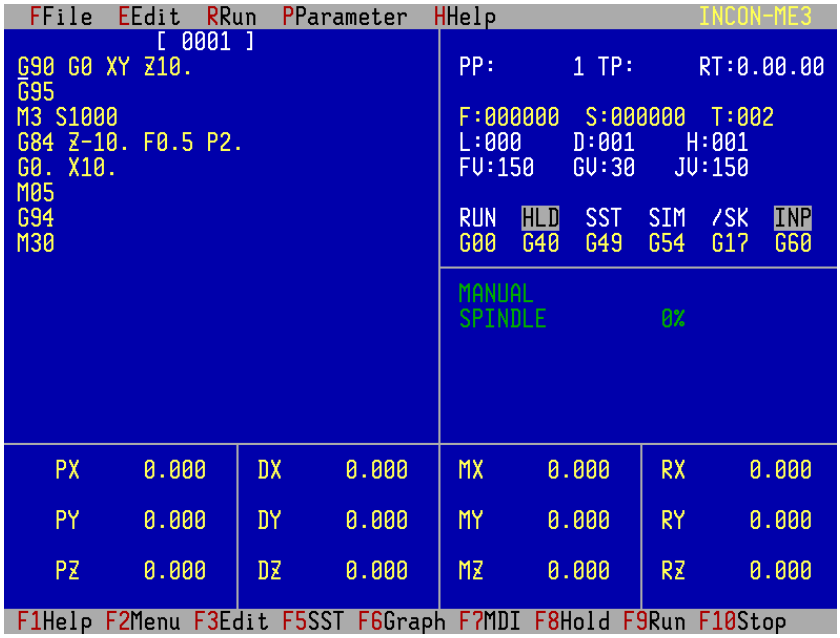


Figure 2-4 "**[filename]**" means Editor is disable and ready to run program.

The top of **Program Window** shows "**< filename >**" means **Editor** can edit the current program. Press **Esc** to finish editing and switch the mode to run program.

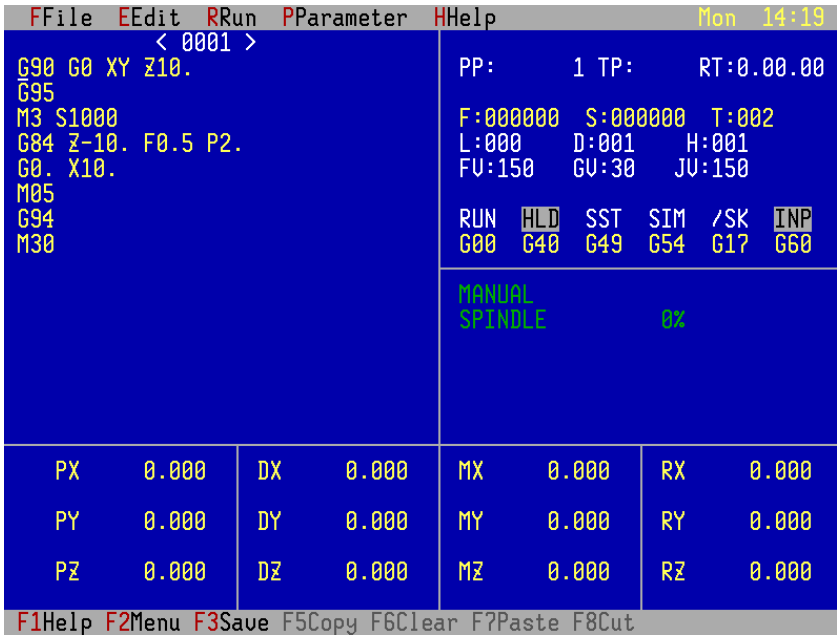


Figure 2-5 "**< filename >**" means Editor can edit the program.

The top of Program Window shows 〝< **filename *** >〞 means the program has modified but does not save yet.

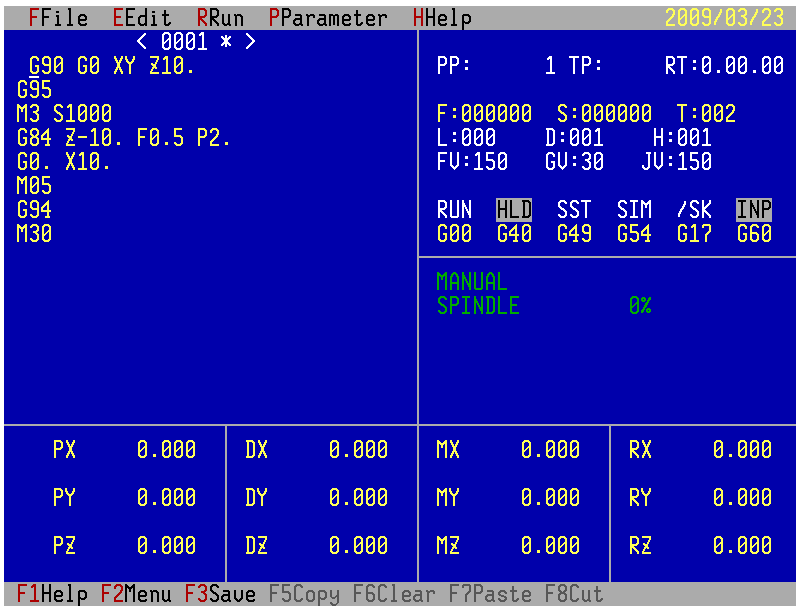


Figure 2-6 〝**< filename* >**〞 means the program modified and does not save.

The filename of the Program Window shows the figure 〝 〞 means current program opens, edits or modifies in an USB device.

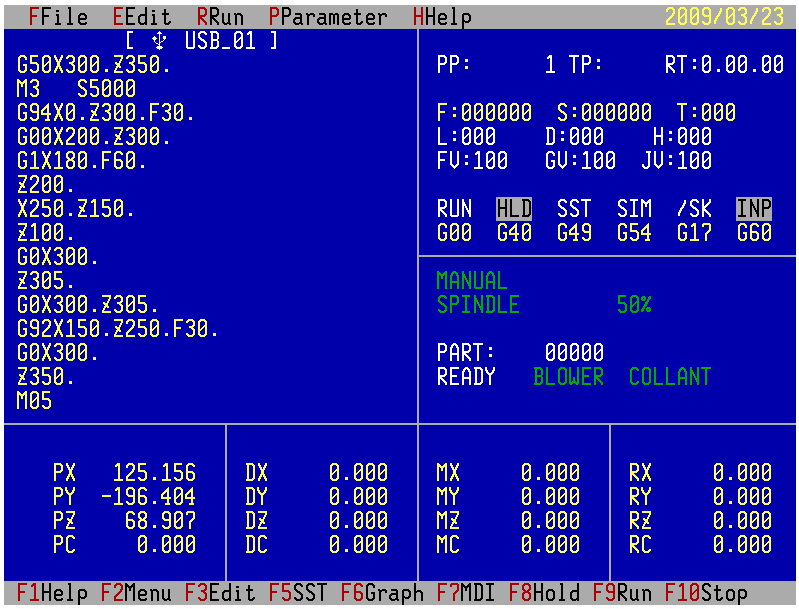


Figure 2-7 〝 〞 means current program opens, edits or modifies in an USB device.

2.3.2 Position State Window

Position state window displays 4 positions for user to operate controller. It is located on the bottom of the screen and update at once.

PX/PY/PZ: Program position for all axes **MX/MY/MZ:** Machine position for all axes
DX/DY/DZ: Distance to go for all axes **RX/RY/RZ:** Relative position for all axes

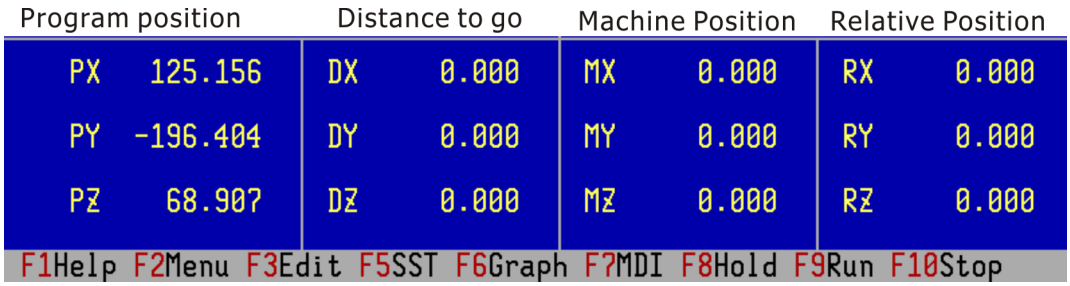


Figure 2-8 Position state window

1. **Program position** keeps a registration of system motion movement in unit of FU〔Feed unit〕.
2. **Distance to go** for all axes shows the remain distance to go of current executing command in unit of FU〔Feed unit〕.
3. **Machine position** keeps registration of the real pulses generated in unit of FU〔Feed unit〕.
   Machine position keeps unchanged as system is in **Machine Lock State**, but **Program position** keeps on recording motion movement.
   Position soft limit use **Machine position** to compare with soft limit data.
   Machine position is reset to machine position preset as Home Return completes.
4. **Relative Position** shows the distance traveled after last menu item **Relative Position Reset** has been executed.

Note:

1. The least significant digit of position data of INCON-ME3 system is equal to 1 FU〔Feed unit〕. This FU is the basic unit of distance, 0.001 mm.
2. During tool length compensation, the Z value in program position will be the no-compensated position. Whereas in tool radius compensation, the X / Y value in program

position will be the compensated position.

2.3.3 System State Window

The System State Windows is located on the right top of the screen, and it displays current system states.

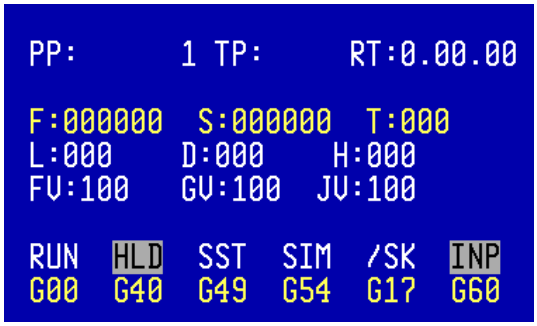


Figure 2-9 System State Windows

The bottom of the **System State Window** lists some toggled state for execution control. If the state is blocked in white reverse, this state is active. The meaning of these states lists as the following:

PP ：Displays current running program line
TP ：task number of program execution line
RT ： Displays time for running the program
F ：F_ value of the program
S ：Spindle speed setting
T ：Tool selection
L ：Loop count
D ：tool radius compensation number
H ：tool length compensation number
FV ：Feedrate override
GV ：**G00** speed override
JV ：Jog speed override

RUN：The program is running
HLD：program hold
SST：Single Step
SIM：Simulate
/SK：Block Skip
INP：in position
G00：run **G01/G02/G03/G33**
G40：run **G40/G41/G42**
G49：run **G43/G49**
G54：run **G54~G59**
G17：run **G17/G18/G19**
G60：run **G60/G61**

2.3.4 I / O State Window

The I / O State Window is under the System State Window. It displays the current situation of the machine.

The first two lines display fixed items: **manual** and **spindle**. The others display items depending on PLC.

Manual

Upon using 〝**Manual**〞 of GMPU〔 〝**Manual**〞 includes **Single Step**, **Jog**, **Hand Wheel** and **Home**〕, the first line highlights with yellow color. The last item of this line means 〝**STEP**〞.

Under the **Manual** mode, press Alt+X〔Y, Z〕or Shift+X〔Y, Z〕to pop up a **Coordinate Offsets** dialog box to set parameters for geometry offset. Don't have to select this menu item: **Parameter→Coordinate Offsets ....** to set parameters again.

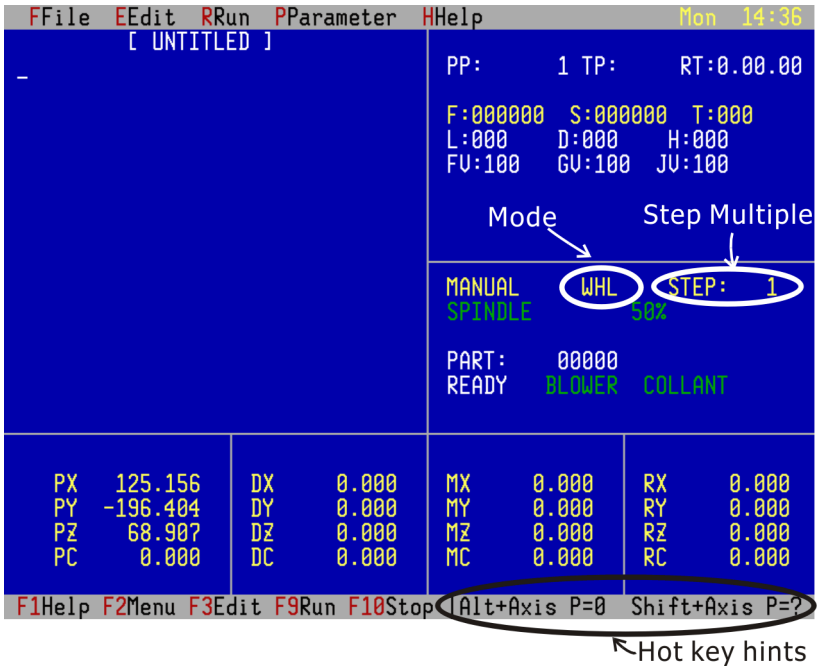


Figure 2-10 Manual display

Spindle

"**SPINDLE**" displays direction, override percentage and speed of spindle.

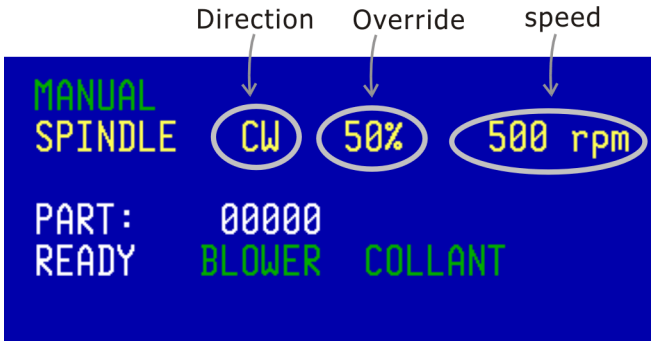


Figure 2-11 Spindle display

2.3.5 Trace Window〔F6〕

When a program simulates or runs, press **F6** and then the **Trace Window** is located on the right bottom of the screen.

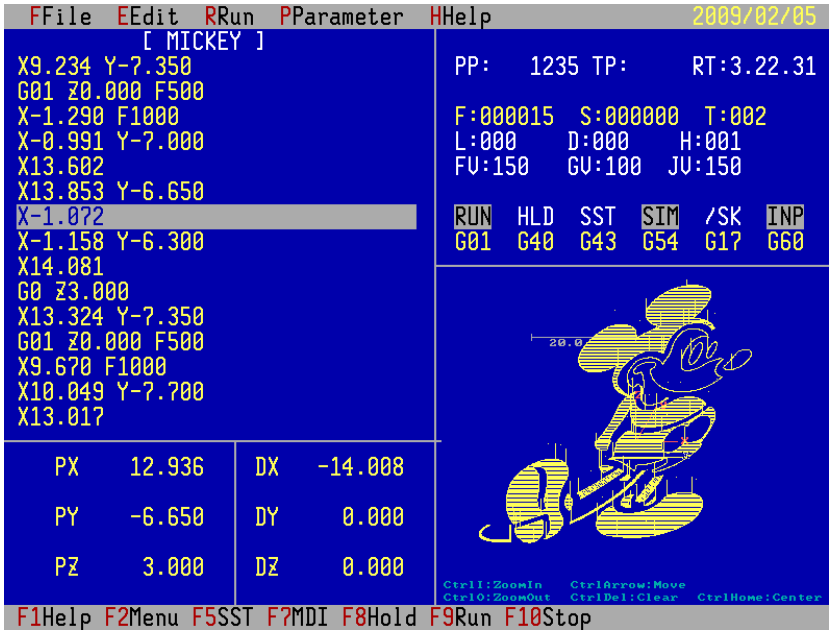


Figure 2-12 the **Trace Window** is located on the right bottom of the screen.

There are some keys can be used to configure the graphic environment:

- **Arrow keys:** Move the graphics up / down / left / right
- **Ctrl+I / Ctrl+O:** Zoom in / out the X and Z axis of graphics
- **Ctrl+Home:** To center the graphic cursor
- **Ctrl+Del:** Clear the graphic view
- **Esc :** Leave the graphic mode.

2.4 MDI Edit Window

This is a dialog box for user to enter one line of command.

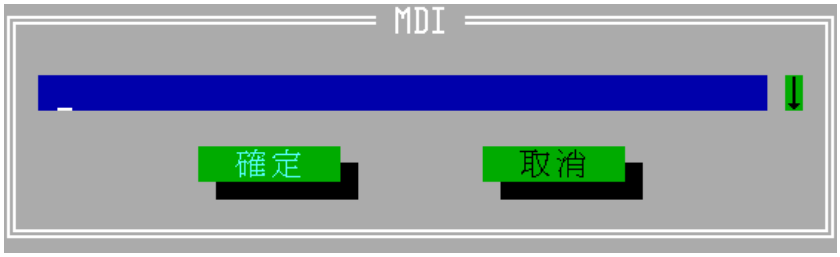


Figure 2-13 A MDI dialog box

You can also use **Down arrow** key to recall the commands that was entered previously. After making sure that the command line is correct, press **Enter** to execute. **Esc** to abort **MDI** motion, **F10** to stop the whole system〔including the program that is running〕.

Warning : Press **Enter** key will execute the command in the input line immediately. To make sure all machine limits, before pressing **Enter** key!

2.5 Manual Control

The manual control is conjunctive with machine panel, your machine marker shell proved more detail of operation.
The following operation is conjunctive with GMPU.
There are 4 modes of **Manual Control**:

〝**INC**〞: Increment
〝**JOG**〞:Jog
〝**WHL**〞:Hand Wheel
〝**HOM**〞:Home Return

Four adjust override and their ranges are list as below:

Set Jog speed override: Total 16 steps, in 10% interval.〔0%~150%〕
Set G00 speed override: Total 16 steps, in 10% interval.〔0%~150%〕
Set Feedrate Override: Total 16 steps, in 10% interval.〔0%~150%〕
Set spindle Override: Total 11 steps, in 10% interval.〔50%~150%〕

〝INC〞: Increment

Press the specified button to switch the mode to **INC**. Then moves the axis by the button +X, -X,+Z or -Z. Pressing the rotary encoder can select the **INC** strided distance.〔There are 4 selections of strided distance, 1, 10, 100 and 1000.〕

〝Jog〞: jog

Press the specified button to switch the mode to JOG. Then moves the axis by the button +X, -X,+Z or -Z. Adjust jog speed override by rotating the rotary encoder.

Note: **JOG** speed is set by 〝**Machine parameter→Axis→X/Y/Z axis→Jog speed**〞.

〝WHL〞: Hand Wheel

Press the specified button to switch the mode to **WHL**. The default PLC setting is using rotary encoders to operate **Hand Wheel** function. The machine can be set in single hand wheel or multiaxial hand wheel mode by 〝**Machine parameter→Motion and Speed→[ ]Multi hand wheel**〞.Pressing the rotary encoder can select the **Hand Wheel**

strided distance.〔There are 3 selections of strided distance, 1, 10 and 100.〕

〝HOM〞:Home Return

Press the specified button to switch the mode to **Home**.

The origin of Home is set by machine makers, users execute **Home** can return automatically. For safety, users had better **Home Return** with Z axis first, and then do with X or Y axis. 〔Home sequence can be set by 〝**machine parameter→Motion and Speed→Home sequence**〞.〕Because the motion of **Home Return** is automatic, press the **Emergency Stop** button to stop working when emergency occurs.

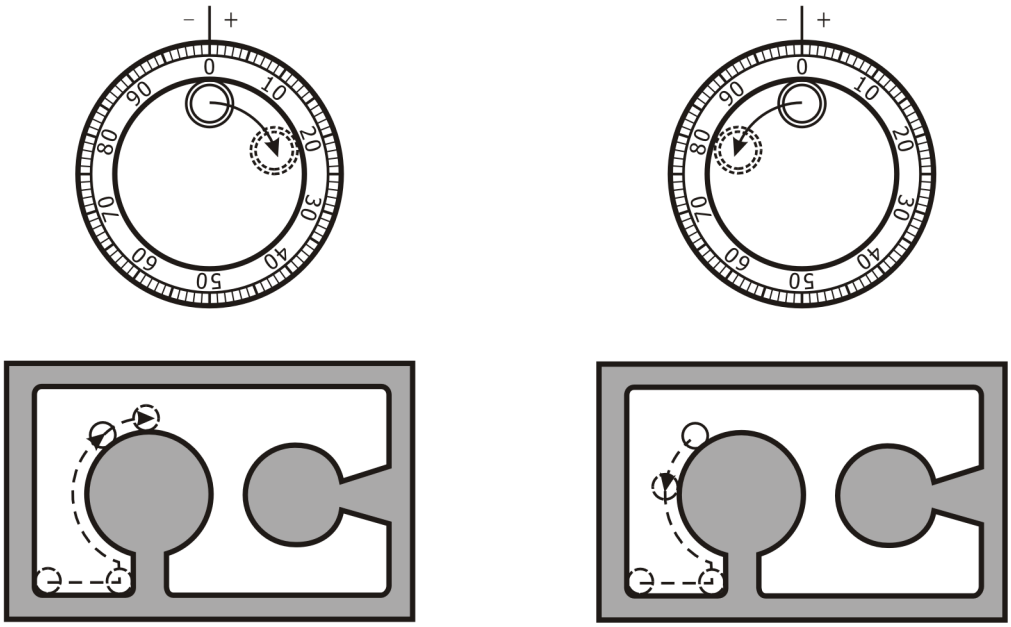
2.6 Manual feed forward or backward

When a CNC program is running, we can press **F8** key or **Feed-hold** button to pause the CNC program. At this situation, users can enable the manual feed function.

Under **WHL** mode, clockwise manual feed operation will keep the interpolation and CNC program running forward with manual feed speed; counter-clockwise manual feed operation will keep the interpolation and CNC program running backward with manual feed speed . In

Under **Inc** or **Jog** mode, the manual feed forward or backward still enables. Therefore we can use these functions to check the tool path by manual feed operation.

(A) CW→forward program path



(B) CCW→backward program path

Figure 2-14 Manual feed forward or backward

Doing manual feed backward, it will reverse the path programmed. Meeting the non-motion command, such as M-code or status-change G-code, the manual feed backward will stop working and don't back anymore .

Three ways enable the manual feed function:

1. Press the specified button to switch the mode to **Manual** and then press 〝**Cycle Start**〞 bottom. The program will be held in to pause state and ready to do manual feed operation.
2. Press **F8** key or 〝**Feed-hold**〞 button during program running, and enable **Manual mode** by GMPU〔2nd operation panel〕. Then it is ready to do manual feed operation.
3. Insert 〝**M100/M101**〞 code in the program. When these two codes being executed, the program will pause like **M00/M01**. Then enable manual feed operation.

Note:

1. **M00/M01** stops running a program and then allow to do the normal **Manual** operation instead of manual feed operation.
2. The value of 〝**Machine parameter→Motion and Speed→Feed acceleration**〞 is

negative denotes the manual feed operation doesn't works, otherwise the manual feed operation dose. The Feed acceleration is positive or negative depending on machine features. Generally, the Feed acceleration has been set by your machine maker.

2.7 File Transmission

INCON-ME3 is a FTP server, we explain how to upload and download files between a controller and a personal computer in this section.

You have to get a FTP software, which is free software from Internet, bought form software company ,or provided by INTEK.

Run the FTP software designed by INTEK, the following page displays on PC screen. The procedure of FTP is list as below:

1. Input the IP address.
2. Input the user name and password. The built-in user name is〝**INTEK**〞and password can be none or other characters.
3. Press **Connect** button, and then log in the FTP controller.

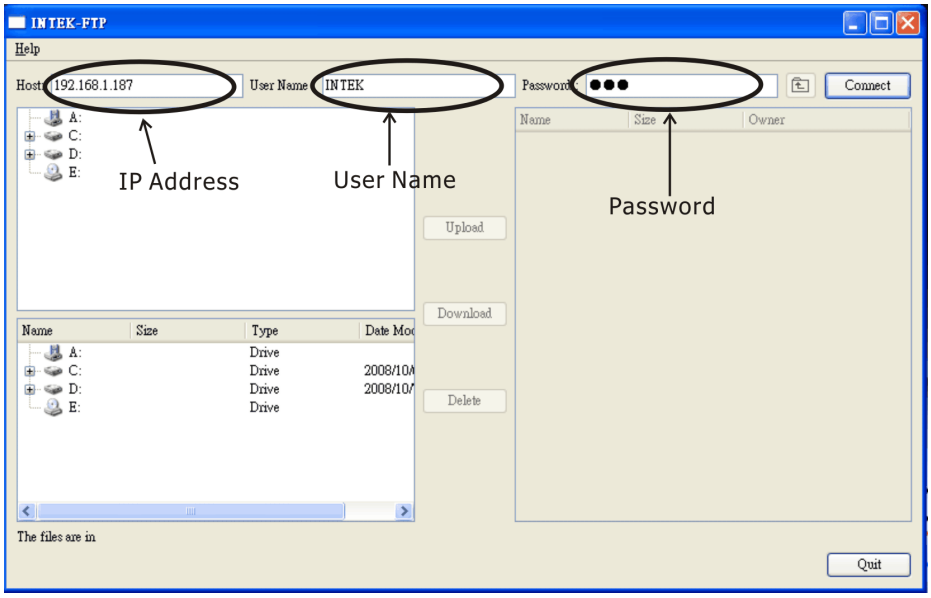


Figure 2-15 the main screen of FTP

At the time of connection success, we find out files of ME3 on the right side of the main screen. We can upload and download files. The operation is like using FTP software generally.

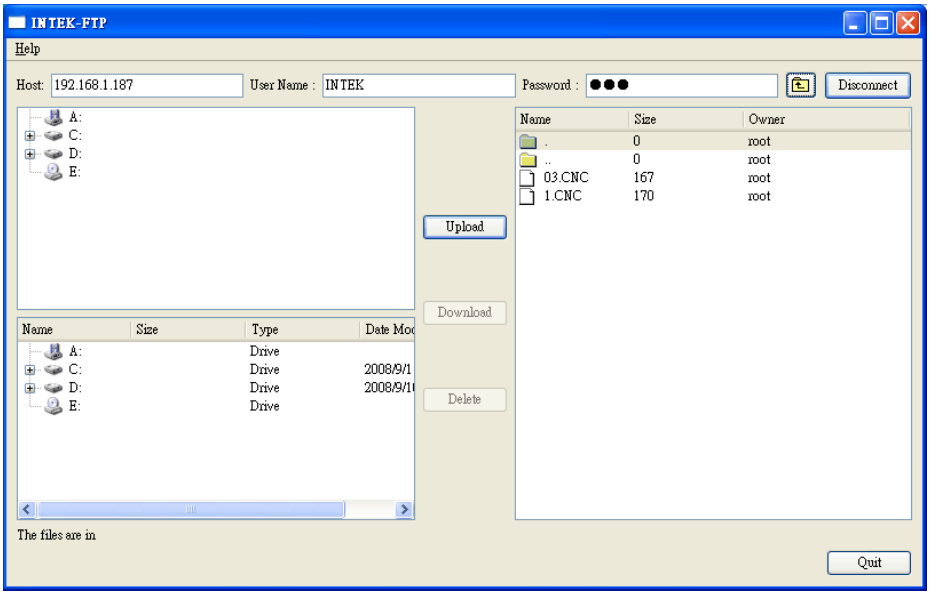


Figure 2-16 the main screen at the time of connection success

2.8 Alarm listing and trouble shooting

When errors happening, the screen pops up an error message dialog box . These error messages are listing in groups as follows:

2.8.1 Fatal system error

No.	Alarm／error message	Trouble shooting
	System check sum error	Power off and turn on again after 10 seconds later. If it still doesn't work, call for service.
501	PLC Alarm	PLC detected I⁄O process error, follow the instruction of the message. If it still doesn't work, please call for service.
514 515 516 517	X position control loop fail Y position control loop fail Z position control loop fail A position control loop fail	Servo driver or encoder error. Power off and turn on again after 10 seconds later. After restarting the system, it still doesn't work, please call for service.
518	PLC Alarm: EMG input is on	PLC detected the EMG button being pushed. Release EMG button and use manual operation to move the tool to safe area.
519 520 521 522	X axis not in-position time out Y axis not in-position time out Z axis not in-position time out A axis not in-position time out	X/Y/Z/A Servo lagged too large to follow the commanded speed. There are many reasons. Refer the above-mention methods to solve the problem.
531	Bad machine parameter data, please reload it	Due to abnormal shutdown. (Call for service, and reload the machine parameter)
546	Spindle close loop control fail as doing rigid tapping	Check the circuit of spindle, motor or encoder. And then check whether spindle rotates smoothly. If it still doesn't work, please call for service.

2.8.2 Motion control alarm

No.	Alarm／error message	Trouble shooting
506	Soft limit on X positive side tripped	Software limit tripped.
507	Soft limit on X negative side tripped	Check the limit travel data in related machine data.
508	Soft limit on Y positive side tripped	Or check the travel distance in the CNC program.
509	Soft limit on Y negative side tripped	
510	Soft limit on Z positive side tripped	
511	Soft limit on Z negative side tripped	
512	Soft limit on A positive side tripped	
513	Soft limit on A negative side tripped	
523	Limit on X positive side tripped	Limit switch tripped.
524	Limit on X negative side tripped	Check the travel distance in the CNC program or
525	Limit on Y positive side tripped	check the limit switch.
526	Limit on Y negative side tripped	
527	Limit on Z positive side tripped	
528	Limit on Z negative side tripped	
529	Limit on A positive side tripped	
530	Limit on A negative side tripped	
542	X axis over travel limit, position is lost, please do home after recover	PLC detected over travel fatal error, and disable the servo driver of the relative axis. Therefore the position is lost and must do home operation after recovery the error.
543	Y axis over travel limit, position is lost, please do home after recover	
544	Z axis over travel limit, position is lost, please do home after recover	
545	A axis over travel limit, position is lost, please do home after recover	

2.8.3 Other fatal errors

No.	Alarm／error message	Trouble shooting
505	Internal error	Illegal G code at N
532	Do homing first	Never do home return operation, therefore program cannot fine the zero point.
533 534	Probe detected outside of detect zone Probe not detect	Check the detect zone again and modify the G37 command
538 539 540	Please home Z axis before this home action Please home Y axis before this home action Please home X axis before this home action	Home Z before this Home Y before this Home X before this
	Divided by zero error Exception error	System process fatal error. If possible, please offer the program for INTEK technology to simulate the error condition and correct the error. Thanks for offering.

2.8.4 General errors

No.	Alarm／error message	Trouble shooting
001	Incorrect command line	Refer to G code list
002 003 004	The number of nested M95 exceeds 8 The number of nested M97 exceeds 8 The number of nested M98 exceeds 8	The layer number of subroutine nest call can up to 8 layers maximum.
005	Incorrect M95 command format	M codes error
006	M97 needs a subprogram number to execute	M97 has no subprogram number
007	M98 needs a subprogram file name to execute	M98 has no a subprogram file name
008	Can not execute M99, since no subprogram has been called	No main program and execute subprogram directly
009	Can not change working plane here	Can not change working plane here in tool radius compensation
010	No homing has been done	Do home operation first
011	No compensation is allowed	No compensation is allowed in MDI dialog box MDI

No.	Alarm／error message	Trouble shooting
012	Can not execute G29, since no G28 or G30 has been done	There must be a middle point specified by G28 or G30 before G29
014	Radius compensation is on already	Don’t given G41/42 again.
015	No G51 is allowed, since G68 is on	G51 can not execute before cancel G68
016	No G50 is allowed, since G69 is not done yet	When G68 is in G51，must use G69 cancel G68 command first and then use G50 cancel G51 command
017	Incorrect G52, at least one of X, Y, Z, OR A must presents	Illegal G code
018	Must be G01 to do chamfering or corner rounding	
019 020	Incorrect mirror axis Working plane has to be specified	Mirror axis is relative to working plane specified
021	Another radius compensation is on	Don’t give G41/42 again under radius compensation
022	Length compensation is on already	Don't give G43 again under length compensation
023 024	The command code to approach radius compensation must be G00 or G01 The command code to depart from radius compensation must be G00 or G01	
027 028	Third axis motion is not allowed when radius compensation is on or in chamfering/corner rounding session Fourth axis motion is not allowed when radius compensation is on or in chamfering/corner rounding session	Radius compensation and chamfering/corner rounding are only performed on dedicated plane
029	No full circle is allowed following G02/G03 command during radius compensation	
030	Interpolation command is not allowed following a full circle when radius compensation is on	
031	Illegal MDI code	Only some legal code can be accepted in MDI
032	Illegal H Code	H range: 1~128
033	Illegal T Code	T range: 1~128
034	Illegal M Code	M range: 0~99
035	Illegal M Code	Refer to G code list
036	No D code allowed here, since tool radius compensation	Don't change tool radius compensation under G41/G42

No.	Alarm／error message	Trouble shooting
	is on	
038	Incorrect can cycle command, need a reference position	
039	Cancycle cancellation is not allowed since cancycle is not done yet	Press F10 to stop Cancycle
040	Radius compensation is on, no motion code is allowed	
041	G33 is not allowed in G94 mode	G33 must be performed in G95 mode.
042	Drilling depth is missing	
043	Reference distance is missing for G87	
044	G87 has to be in G98 mode	
045	No radius compensation is allowed for can cycles	
046	Can cycle or pocket is on, no motion command in MDI is allowed	
047	Scaling is on already	Don’t repeat G51 before cancel it.
048	Rotation is on already	Don’t repeat G68 before cancel it.
049	Out of memory	Exception error (If possible, please offer the program for INTEK technology to simulate the error condition and correct the error. Thanks for offering)
050	G53 has to be run in absolute mode	
051	Tool length measurement is not allowed during fast run stage of run from middle	Fast run doesn’t simulate G37 command. G37 should be under really run the tool length measurement.
052	Tool number has to be assigned before carrying out G37	Must specify the D value before G37
053	Circular pocket radius has to be specified	
054	Rectangular pocket length and width have to be specified	
055	In feed increment is needed for pocket command	
056	Pocket center is not defined	
057	Pocket cutting orientation is not specified	
058	G88 is not allowed during fast run stage of run from middle	Fast run doesn’t simulate the manual operation in G88.
059	Pocket total depth is smaller than its fine feed depth	The fine feed depth must be reasonable.
060	Pocket total dimension is smaller than its fine feed width	The fine feed width must be reasonable.
061	G37 is not allowed in case of machine lock or spindle	Unlock the machine and spindle before execution G37

No.	Alarm／error message	Trouble shooting
	lock	command.
062	No G92 when either tool compensation is on	G92 is not permitted in G42/43/44 mode.
063	Rigid tapping command without S code	Refer to chapter 4
064	Rigid tapping command with too large an S Code	The S value should be reasonable.
065	Rigid tapping is disabled since max rigid tapping spindle speed is 0	The S value should be reasonable.
066	Illegal D Code	D range: 1~128
067	Spindle axis feed rate is missing" for pocket command	Specify the I_ item in G70/71 command.
068	Tool radius can not be zero" for pocket command	Set radius in tool table and specify the tool table number previously.
069	Corner rounding radius is too big	The corner rounding radius should be reasonable
078	Number is too large	The given value larger than 9 digits
081	Tool radius compensation for next two commands will over cut	Tool radius compensation don’t suit for too many continual short lines cutting, short than tool radius
082	Lack of next section for tool compensation	See the note of tool compensation
083	Incorrect sign for some command items	Check the format of the command
084	Illegal can cycle code	Check the format of can cycle
085	Incorrect working plane with respect to spindle aligning axis	Refer to pocket or dowel command
086	OSS shift is necessary for G76 or G87	Refer to the command of G76 or G87
087	Not a close contour	Refer to the command of G70.1x
088	Contour compensation type is necessary	Refer to the command of G70.1x
089	Contour approaching radius is necessary	Refer to the command of G70.1x
090	Contour subroutine is missing	Refer to the command of G70.1x
091	Illegal contour command code	Refer to the command of G70.1x
092	3rd axis motion is not allowed for contour	Refer to the command of G70.1x
093	Too many commands for contour	Refer to the command of G70.1x
094	Incorrect compensation(G41/G42) for contour pocket	Refer to the command of G70.1x
095	4th axis motion is not allowed when direction tracking is on	Refer to the command of G70.1x
096	4th axis has to be on for direction tracking	Refer to the command of G70.1x
097	Incorrect object pattern command line	Refer to the command of G70.1x
098	Object count has to be greater than zero	Refer to the command of G70.1x

No.	Alarm／error message	Trouble shooting
099	Center position is needed for arc object pattern	Refer to the command of G70.1x
100	Sections in contour pocket intersects	Refer to the command of G70.1x
101	while system is running and not held by M00/M01, only M,S,T codes are allowed for MDI	Refer to the chapter of panel operation → MDI section
102	A subroutine/subprogram has to end with an M99	Follow the message
103	Hard Disk Error	Check hard disk
104	Incorrect command line for general data setting	
105	Data setting mode is on	

2.9 Maintenance notice

1. The quality of local electrical power isn't steady, it has to add a power regulator to keep the electrical variation under 10%.
2. To keep the controller in better work condition, it had better to keep the room temperature under 40℃.
3. Clean the filter of the electric cabin weekly or monthly to keep the controller with good ventilation.
4. Use the alcohol or cleaning naphtha to clean the key pad and the acrylic board. Don’t use the strong solvent!
5. Try your best to let all sorts of fluids and ashes not to touch on printed circuit boards of controller .To avoid weak electronic components or short circuit.

Chapter 3 Pull Down Menu

INCON-ME3 provides a convenient window based screen for you to access all functions of this system. There are 5 submenus listed on the upper edge of screen. They are **File**, **Edit**, **Run**, **Parameters**, and **Help**.

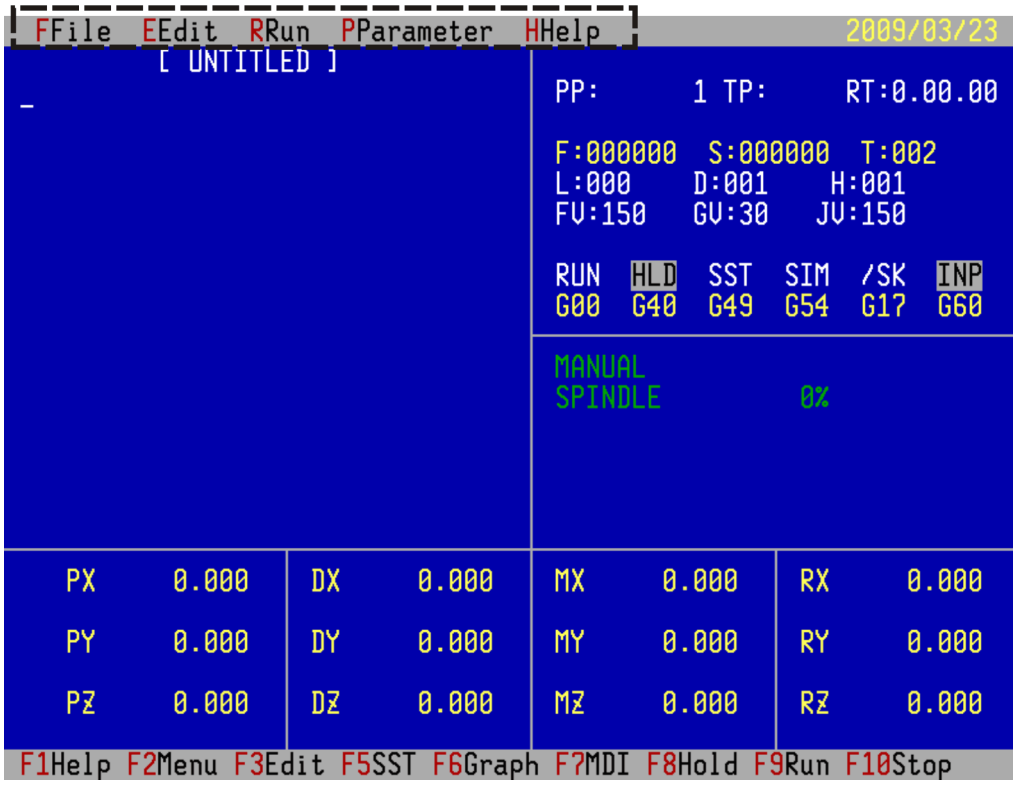


Figure 3-1 Pull Down Menu on the upper edge of screen

Users can pull down each one of them by pressing **F2** and then the highlighted character of that submenu or the arrow keys.

For Example: pressing **F2→E** pulls down the **Edit** submenu.

Then you can use arrow keys as follow:

Up〔↑〕/Down〔↓〕 arrow key : highlight one of the menu item.
Left〔←〕/Right〔→〕 arrow key : switch to another submenu.

The selected menu item will be highlighted. Pressing **Enter** key can be entered to activate the menu item. At this time, there are usually 2 buttons on a dialog box popped up, **OK** or **Cancel**. Pressing **Enter** key means enter and save the setting, but Pressing **Esc** key means cancel the setting and leave the dialog box.

The dialog box of **Parameter** setting uses the **Enter** key to save the setting and dialog box keeps the screen for users to check their settings until press **Esc** key to leave it.

You can also use Hot keys shown in the status line on the lower edge of the screen to activate the desired functions immediately. For example: Press **F6** for graphic display.

3.1 File〔F2→F〕

The **File** submenu provides user to manage the CNC programs.

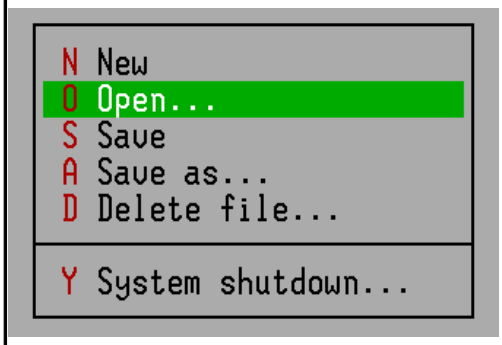
<table>
<tr><td> 

Figure 3-2 The File submenu</td><td>Description</td></tr>
<tr><td rowspan="2"></td><td>N：Initiate a new empty CNC program
O：Select an existed CNC program
S：Save current CNC program
A：Save current CNC program with a new name
D：Delete CNC program</td></tr>
<tr><td>Y：Shutdown the system in normal sequence automatically.</td></tr>
</table>

Note: It is strongly recommended to use the **System Shutdown** menu item before turning off the power. The system shutdown menu item lets system has a chance to save the important

parameters of the system.

3.1.1 Menu Item : File→New

Select this menu item to initiate a new empty CNC program.

User can start to program a new CNC program, and then select the menu item 〝**File→Save as ...**〞 or 〝**File→Save as...**〞 to save the new CNC program.

3.1.2 Menu Item : File→Open...

ME3 system provides memory capacity to save CNC program, and how many bytes memory capacity has depends on the model of machines. Users also can load or save CNC programs from USB device〔This is an optional function.〕

Select this menu item to open an 〝**Open a File**〞 dialog box to select an existed CNC program. The dialog box only lists the files named with extension name 〝**.CNC**〞. Users can open a file by direction key to select desired file or inputing file name.

Note: Dialog box listing does not show extension name.

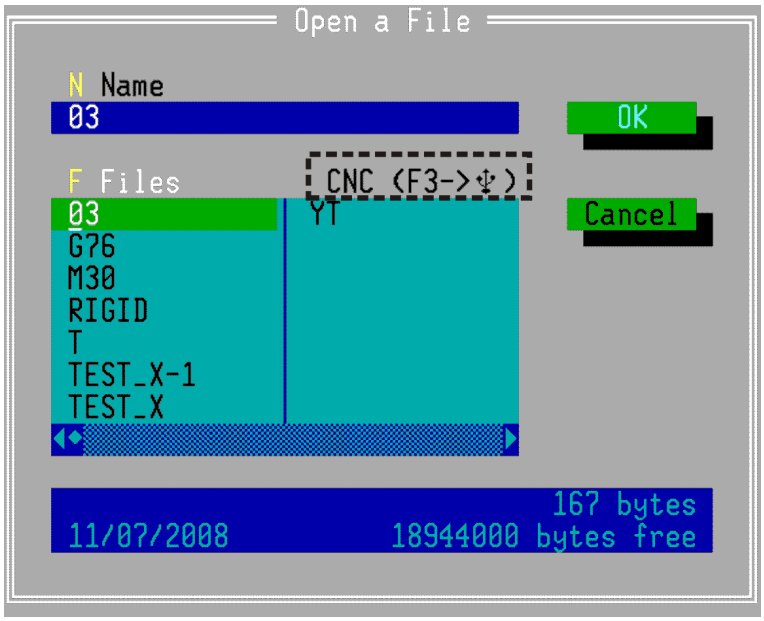


Figure 3-3 An〝Open a File〞dialog box to select an existed CNC program in the ME3 controller.

Press F3 to switch the memory device to USB on the 〝**Open a File**〞 dialog box. When your desired file is in ME3 controller, press F3 and then return to controller to open a file. The memory ／ USB switch hint is on the right side of the 〝**Open a File**〞 dialog box.

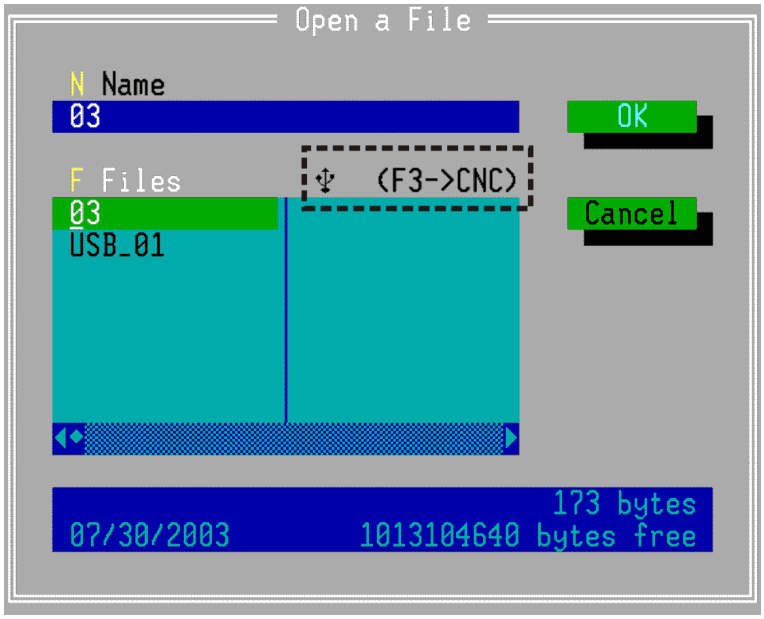


Figure 3-4 This is a screen of 〝**Open a file**〞 dialog box from USB, and user can open a file from USB.

3.1.3 Menu Item : File→Save...

Select this menu item to save current program under the name shown on program title in controller or USB.

3.1.4 Menu Item : File→Save as...

This file dialog box is to enter file name and save file.

3.1.5 Menu Item : File→Delete File...

Select this menu item to open a〝**Delete a File**〞dialog box to delete file selected. Press **Enter** to finish the deletion.

All operations are the same, no more mentions in the later chapter.

3.1.6 Menu Item : File→System Shutdown...

Shut down the system in normal sequence automatically.

Note: It is strongly recommended to use the **System Shutdown** menu item under File submenu before turning off the power. The system saves its important parameters in the system shutdown procedure.

3.2 Edit〔F2→E〕

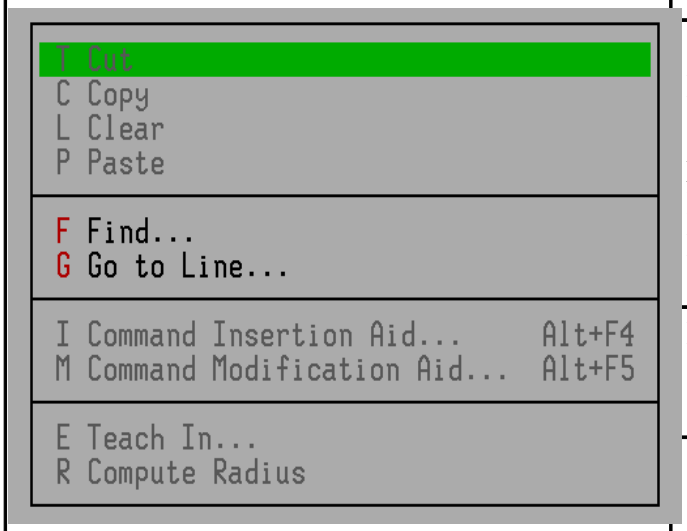


Figure 3-5 The pull down menu: Edit

Description
T：Remove selected text from a CNC Program and put it on the clipboard C：Copy selected text from the CNC program and put it on the clipboard L：Clear the selected text P：Insert a copy of the clipboard contents at the insertion point
F ：Search for specified character string and make it reverse white . G：Go to a specific line
I ：Help user on programming G code commands. M：Show the detail information about the command pointed to by editor cursor.
E：Input the position data to the CNC program automatically R：Auto compute radius by 3 teach in point

The **Edit** submenu provides well known functions, such as 〝**Cut, Copy, Paste, Find, Goto…**〞. ME3 also provides more powerful functions, such as 〝**Command Insertion Aid, Command modification aid**〞, 〝**Teach In**〞, 〝**Compute Radius**〞, for the users who does not know the G or M codes well.

3.2.1 Editor

Upon opening an existed program, it can be executed at once. When you want to modify an existed CNC program, press **F3** key and switch from current mode to edit CNC program.

Operation Key	Function	Operation Key	Function
→	Shift cursor right	**PgUp**	Page up
←	Shift cursor left	**PgDn**	Page down
↑	Shift cursor up	**BS**	Delete a character forward
↓	Shift cursor down	**Delete**	Delete a character
Ctrl + →	Shift right a word	**Ctrl + T**	Delete a word
Ctrl + ←	Shift left a word	**Ctrl + Y**	Delete a line
Shift +↑↓→←	Block an area	**Ctrl + Delete**	Delete a block
Home	Shift cursor to line start	**Ins**	Insert Mode
End	Shift cursor to line end	**Ent**	New Line
Ctrl + Home	Shift cursor to page top		
Ctrl + End	Shift cursor to page bottom		

3.2.1.1 Menu Item : Edit→Find...

Use this **Find** command to search for specified character string.

A **Find** dialog box will pop up for users to type in the string for searching. When finish typing string and then press **Enter** key, the command executes and shows specified character string highlighted.

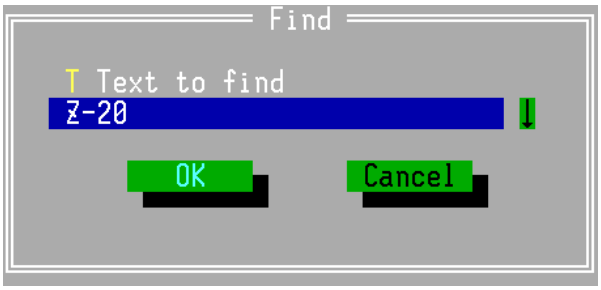


Figure 3-6 A Dialog box : Find

3.2.1.2 Menu Item : Edit→Go to Line...

Executing this item can move a cursor to a specified line.

A **Goto / Jump** dialog box will pop up for users to input line number desired.

Input the sign〝+〞or〝-〞in front of the line number, cursor moves up or down 〝**nnn**〞lines from the current line, else cursor moves to〝**Line number**〞lines from first line.

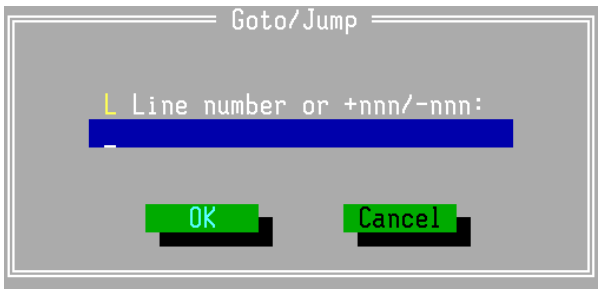


Figure 3-7 A Dialog box: Goto / Jump.

3.2.2 Clipboard

Clipboard is a buffer for cut or copied text.

The clipboard doesn't retain the information until you cut or copy another piece of context into the clipboard. The default clipboard is empty when power on.

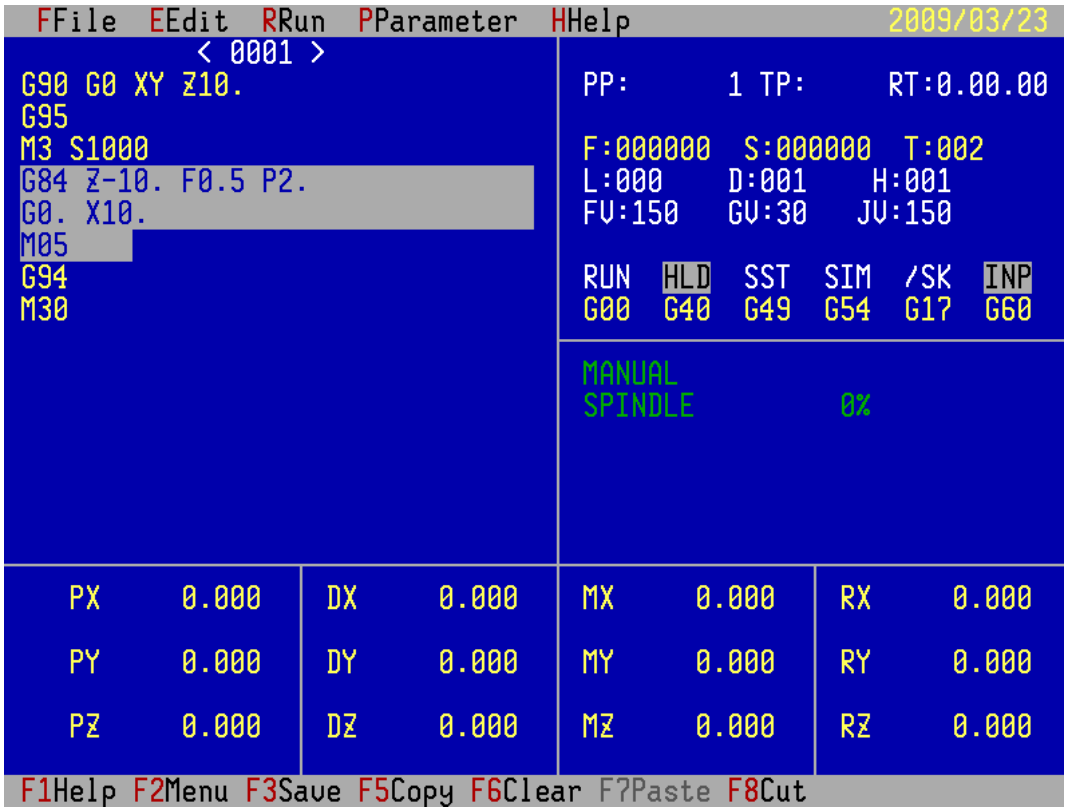


Figure 3-8 The selected text will be highlighted ,and the lower edge of the screen appears statusline under the〝Edit〞mode.

How to Select a Piece of Text

Shift the cursor in the front of the text desired to cut or copy, and then use the **SHIFT + direction key** to select a piece of text.

The selected text becomes highlighted at once and users can cut or copy a piece of text to the clipboard by〝**Edit→Cut**〞or〝**Edit→Copy**〞. Then users can paste it by〝**Edit→Paste**〞.

There is one more easier way to do above-mentioned motion: Under **Edit mode**, press **F5** to copy, press **F8** to cut, press **F7** to paste and press **F6** to clear the Piece of Text.

3.2.3 Command Aid

3.2.3.1 Menu Item : Edit→Command Insertion Aid

Select this menu item to pop up a submenu of command items which can be selected to help users on programming G code commands.

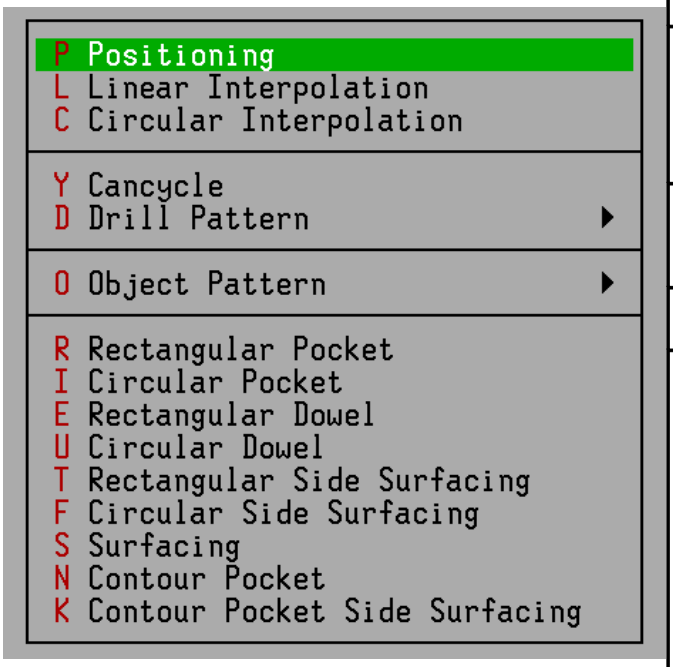


Figure 3-9 a submenu of command Insertion Aid

Description
P：G00 L：G01 C：G02、G03
Y：G73、G74、G76 and G81～G89 D：G70.07~G70.09
O：G70.21~G70.23
R：G70.00 I ：G70.01 E：G70.02 U：G70.03 T：G70.04 F：G70.05 S：G70.06 N：G70.10 K：G70.11 **Note:** These functions can be used under the **Edit** mode

Under **Edit** mode, **Command Insertion Aid** or **Command Modification Aid** function enables by pressing **Alt+F4** or **Alt+F5** all the time and its dialog box pop up for users to input data.

Note: Instructions related to coordinate system are effective to **Command Insertion Aid.** For example, **G17/G18/G19** plane selection, **G20/G21** metric/ inch system, **G90/G91** absolute/ incremental dimension, **G15/G16** Cartesian/ polar coordinate, **G54~G59** work

coordinate selection, **G52** child coordinate, the zero point of **G92** setting and so on.

There are some signs in front of input items in the **Command Insertion Aid** dialog boxes. The followings are meanings of signs:

(1) 〝*〞：the item can be specified or not.
(2) 〝#〞：the item can be combined with other items which have the same sign, **#**, to become varied formats of commands.

I. **Instructions related to route**〔Positioning, **Linear and Circular interpolation**〕

They are usually used in varied route cutting.

1. Positioning〔G00〕

Format：**G00 X_Y_Z_A_;**

When starting or finishing a route cutting, it is usually used the positioning instruction to go into or exit the route. The dialog box of **G00** is as follows.

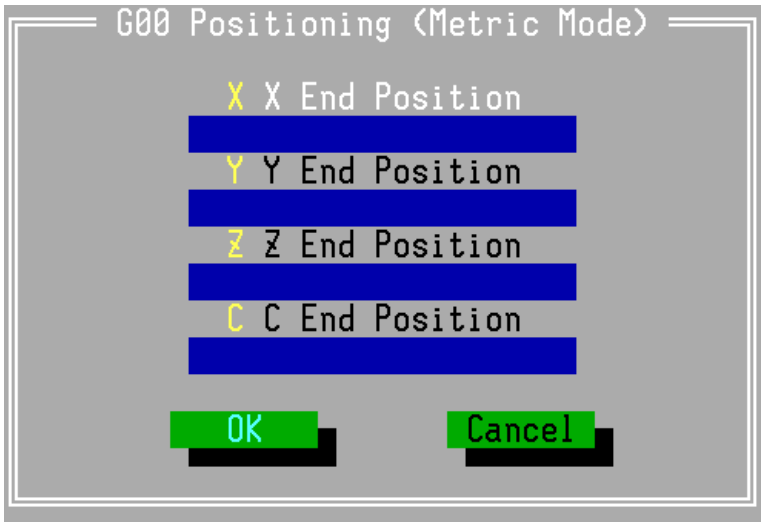


Figure 3-10 A command Insertion Aid dialog box: G00 positioning

Note: Before input the data in the item of dialog box, users must know the settings related to coordinate system of the current program line to avoid errors. The settings are such as

metric/inch system, absolute / increment dimension or Cartesian / polar coordinate and so on.

When **G00** follows **G40/G41/G42**, **G00** also executes transiting into / out the compensated route.

2. Linear interpolation〔G01〕

Format： **G01 X_Y_Z_A_(R_/L_)F_;**

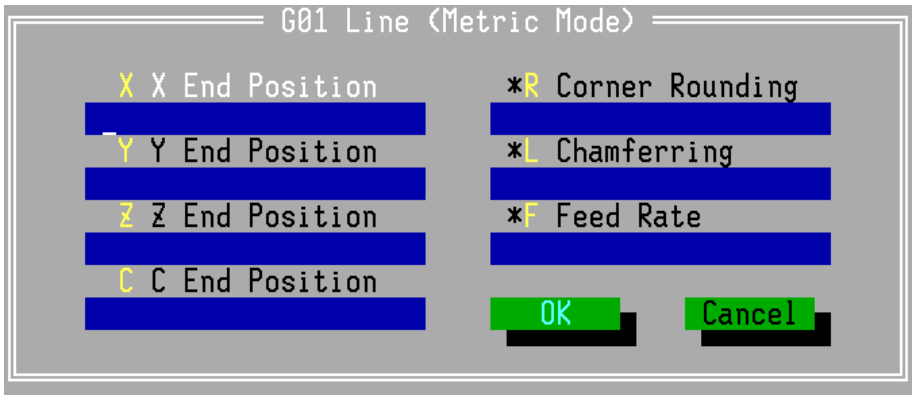


Figure 3-11　A command Insertion Aid dialog box: G01 Linear interpolation

Note: Before input the data in the item of dialog box, users must know the settings related to coordinate system of the current program line to avoid errors. The settings are such as metric/inch system, absolute / increment dimension or Cartesian / polar coordinate and so on.

〝**Corner Rounding**〞 、 〝**Chamfering**〞 specify the R / L value in the following figures, and can be just used in between the two interpolation instructions.

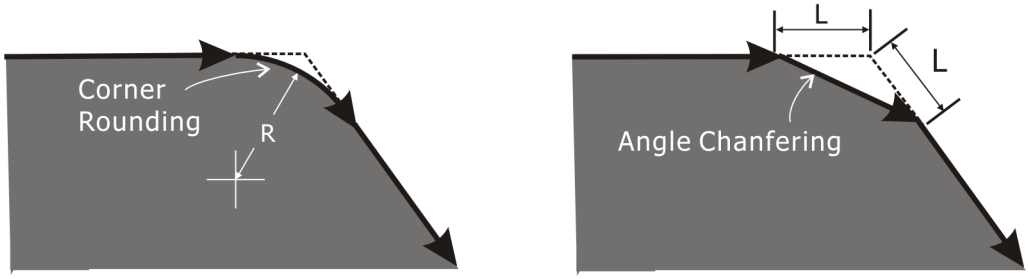


Figure 3-12　The R value of Corner Rounding and the L value of chamfering

The item 〝**Feed Rate**〞 having been set and users don't need to change its value, users can not input value anymore and the setting keeps the last value automatically.

3. Circular Interpolation〔G02/G03〕

Format：

```
G02/G03 X_Y_(Z_)R_F_                    ;Radius Mode
G02/G03 X_Y_(Z_)I_J_(I_K_ / J_K_)F_     ;Center Mode
G02/G03 I_J_(I_K_ / J_K_)A_F_           ;Angle Mode
```

The dialog box will help to insert **G02/G03** into a CNC program. The end position of circular or arc can be edited by absolute / increment dimension or Cartesian / polar coordinate.

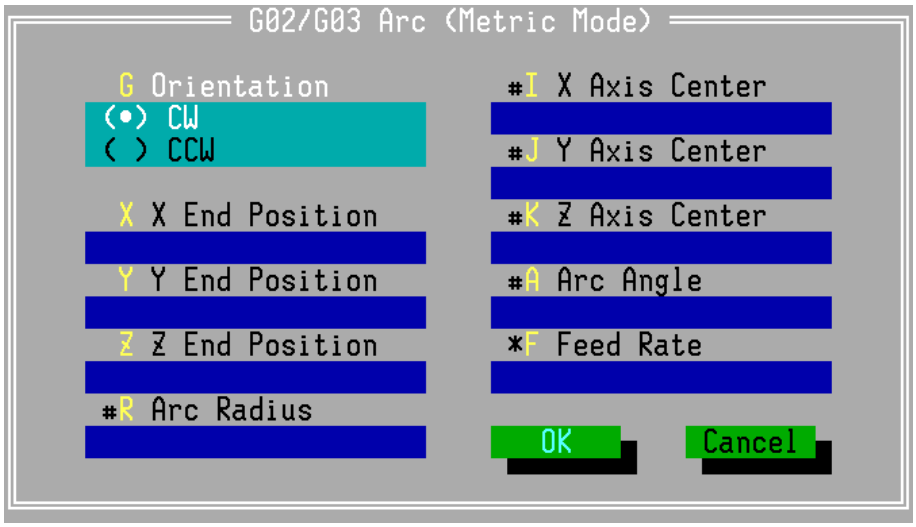


Figure 3-13 A command Insertion Aid dialog box: G02/G03 arc interpolation

The items with the 〝**#**〞 sign in front of item can combine with each other to generate 3 kinds of varied formats of **G02/G03**, the following examples is in **G17** mode:

(1) **Radius Mode**：Input 〝**X X End Position**〞, 〝**Y Y End Position**〞 and 〝**R Arc Radius**〞. When the R value is negative, the arc will be more than 180 degrees.
(2) **Center Mode**：Input 〝**X X End Position**〞, 〝**Y Y End Position**〞 and 〝**I X Axis Center**〞、〝**J Y Axis Center**〞. 〝**I X Axis Center**〞 and 〝**J Y Axis**

Center〞specify the distance from center to start point.

(3) **Angle Mode**：Input 〝**I X Axis Center**〞, 〝**J Y Axis Center**〞 and 〝**A Arc Angle**〞.

The details of **G02/G03** refer to Chapter 4.

II. Cancycle

Format：**G73/G76/G83/G87 X_Y_Z_(R_Q_ K_F_)**
Format：**G74/G81/G84/G85/G86/G88/G89 X_Y_Z_(R_ K_F_)**
Format：**G82 X_Y_Z_(R_P_K_F_)**

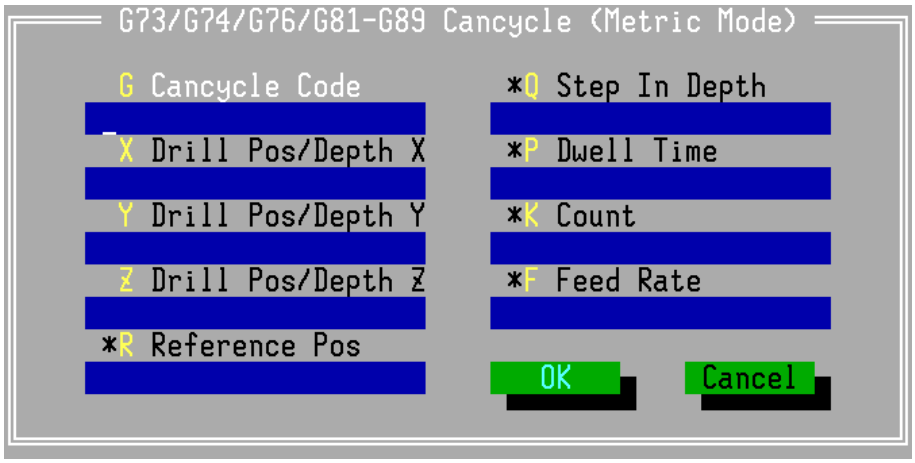


Figure 3-14 A command Insertion Aid dialog box: Cancycle

All meanings in these input items of this dialog box are the same as ones of Chapter 4 Cancycles. Users input data in desired items according to varied cancycles.

The functions of **G90/G91** and **G98/G99** works under this mode.

Drilling cycle instruction codes：**G81/G82、G73/G83**
Boring cycle instruction codes：**G81/G82、G76/G85~89**
Tapping cycle instruction codes：**G74/G84**

The item 〝**K Count**〞 in the dialog box specifies the repeat counts of the drilling can cycle. When 〝**X Drill Pos / Depth X**〞 and 〝Y **Drill Pos / Depth Y**〞 are specified under incremental dimension 〔**G91** mode〕, this motion of drilling can cycle is the same as one of

〝**D Drill Pattern→L Line Drill Pattern**〞.

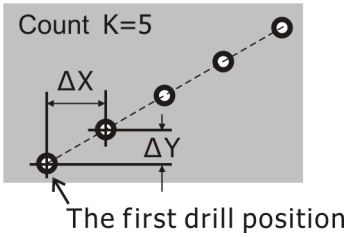


Figure 3-14 drilling can cycle

III. Drill Pattern

There are three drill patterns: line, Grid and Arc drill pattern.

Every can cycle instruction, **G73~G89**, are acceptable in this drill pattern, therefore the **Drill Pattern** also can do boring group or tapping.

The items such as 〝**1st Axis...**〞, 〝**2nd Axis...**〞 in every dialog box of Drill Pattern means: 〝**1st Axis...**〞 denotes X axis and 〝**2nd Axis...**〞 denotes Y axis under **G17** mode. Other modes list as follows:

Plane Selection	〝1st Axis...〞	〝2nd Axis...〞
G17	X	Y
G18	Z	X
G19	Y	Z

1. Line Drill Pattern

Format：**G70.07 G_X_Y_Z_R_I_J_L_Q_P_F**

The holes of cancycle of **Line Drill Pattern** are drilled in a line disposition.

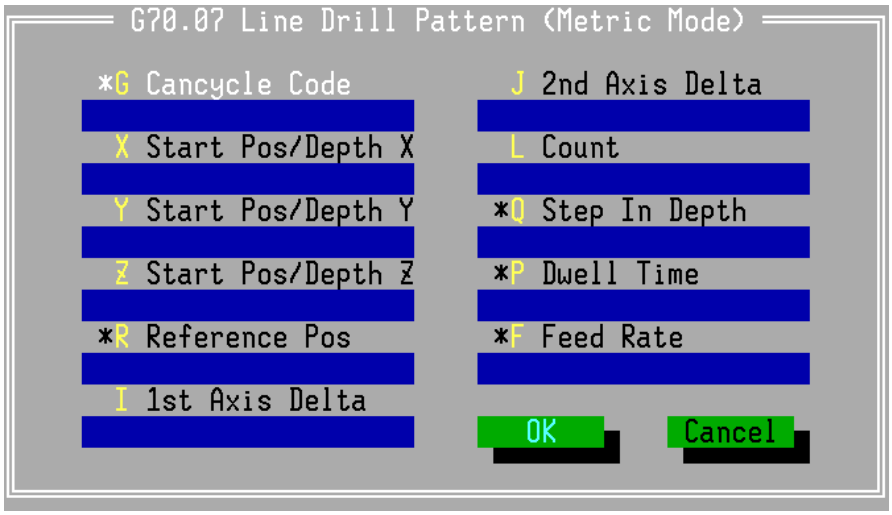


Figure 3-16 A dialog box: Line Drill Pattern

(1) 〝**X〔Y, Z〕 Start Pos / Depth X〔Y, Z〕**〞: Drill pattern start position and depth. The input value can select **G90/G91** mode and **G98/G99** mode.

(2) 〝**I 1st Axis Delta**〞: Delta distance of 1st axis, which is the distance between two consecutive drilling position in 1st axis.

〝**J 2nd Axis Delta**〞: Delta distance of 2nd axis, which is the distance between two consecutive drilling position in 2nd axis

(3) 〝**L Count**〞: Drill pattern counts. This item must be a non-negative number.

(4) After finishing the line drilling, the tool will stop at the initial level of the final position. 〔Note: **the initial level** means the 3rd axis position on the current plane 〔**G17**, **G18** or **G19**〕.

(5) 〝**Q Step In Depth**〞: Cut in depth. The depth **U_**, **V_** or **W_** depends on plane selection〔**G17**, **G18** or **G19**〕.

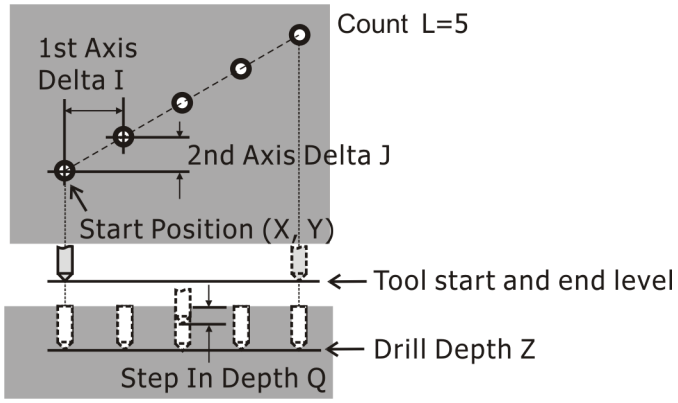


Figure 3-17 Line Drill Pattern

2. Grid Drill pattern

Format：**G70.08 C_G_X_Y_Z_R_I_J_A_B_U_V_W_D_Q_P_F_**

The holes of cancycle of **Grid Drill Pattern** are drilled in a checker disposition.

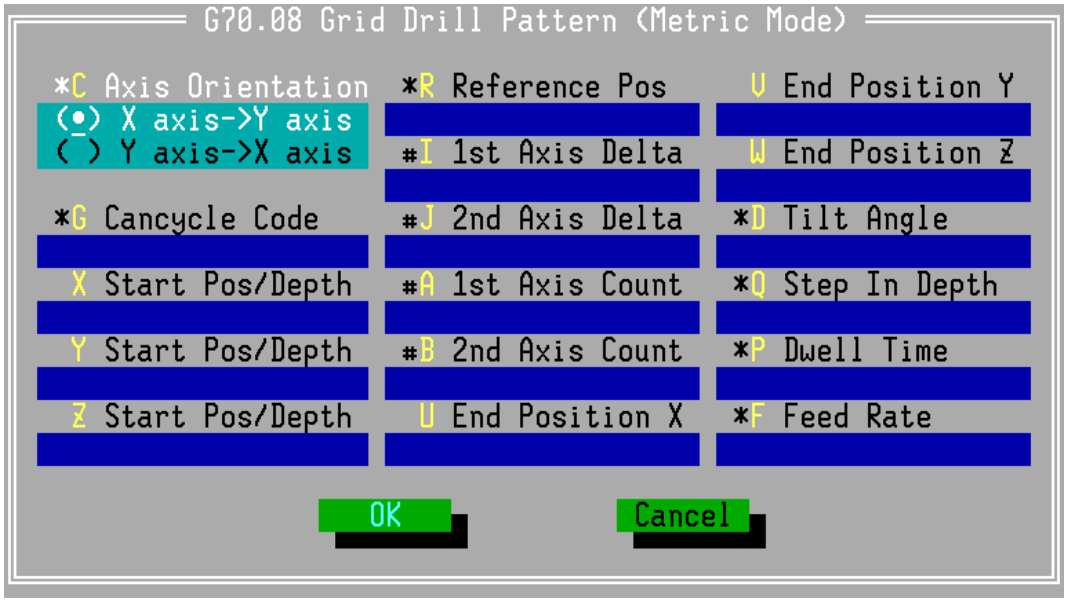


Figure 3-18 A dialog box: Grid Drill Pattern

〝**X〔Y, Z〕 Start Pos / Depth X〔Y, Z〕**〞 programs in **G90/G91** mode. The items with 〝**#**〞 combine to each other to generate three following kinds of **Grid Drill Pattern .**

(1) Input the distance between two consecutive drilling position in 1st / 2nd axis, 〝**I 1st Axis Delta**〞 and 〝**J 2nd Axis Delta**〞 .And then input drill pattern counts of 1st / 2nd axis 〝**A 1st Axis Count**〞、〝**B 2nd Axis Count**〞.

(2) Input drill pattern end position 〝**U End position X**〞, 〝**V End position Y**〞 or 〝**W End position Z**〞, which are defined respectively by two axes among U, V and W depends on which plane is specified. And then input drill pattern counts of 1st / 2nd axis 〝**A 1st Axis Count**〞、〝**B 2nd Axis Count**〞.

(3) Input drill pattern end position〔U, V or W〕and the distance between two consecutive drilling position in 1st / 2nd axis , 〝**I 1st Axis Delta**〞 and 〝**J 2nd Axis Delta**〞.

When input data including of end position 〔U, V or W〕, the distance between two

consecutive drilling position in 1st / 2nd axis〔I, J〕and counts of 1st / 2nd axis〔A, B〕, The end position will be ignored.

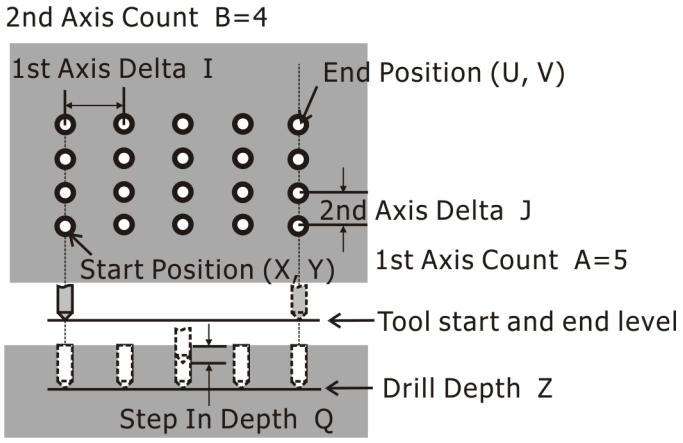


Figure 3-19 Grid Drill Pattern

Note: After finishing the line drilling, the tool will stop at the initial level of the final position.

〝**D Tilt Angle**〞: The tilt angle specifies the rotation angle around center **(X,Y,Z)**. The item with 〝*****〞 means it is optional.

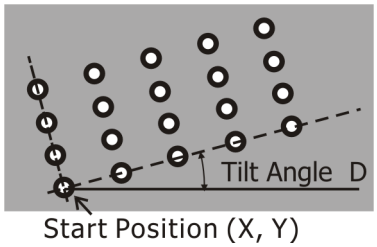


Figure 3-20 Grid Drill Pattern with tilt angle

3. Arc Drill Pattern

Format：**G70.09 G_X_Y_Z_R_C_A_I_L_B_Q_P_F_**

The holes of cancycle of **Arc Drill Pattern** are drilled in an arc disposition.

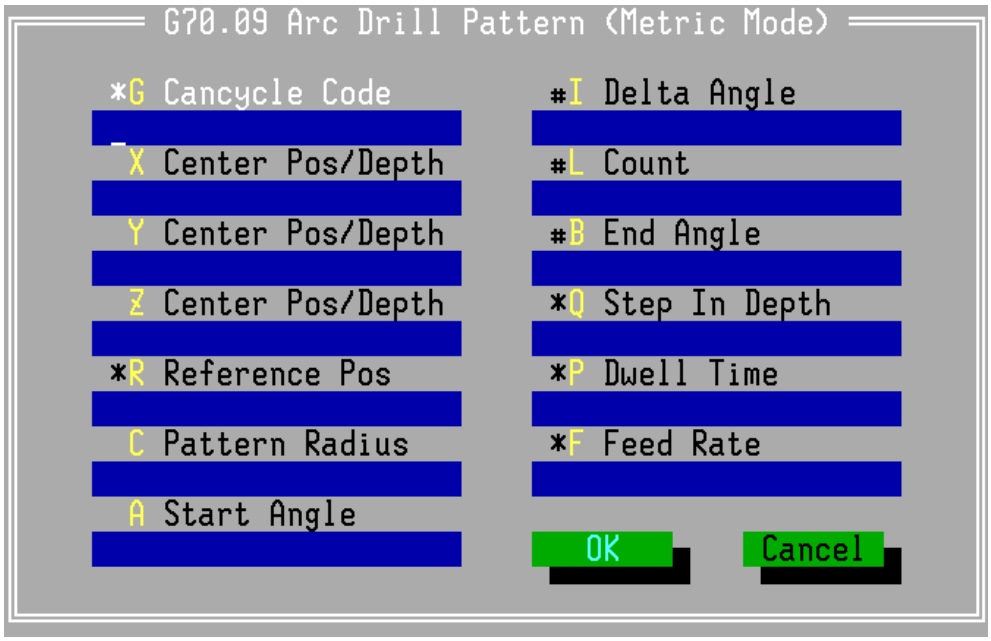


Figure 3-21 A dialog box: Arc Drill pattern

Center position programs in absolute or incremental coordinate system.

The items with 〝**#**〞 combine to each other to generate three following kinds of **Arc Drill Pattern .**

(1) Input 〝**I Delta Angle**〞 and 〝**L Count**〞
(2) Input 〝**B End Angle**〞 and 〝**L Count**〞
(3) Input 〝**B End Angle**〞 and 〝**I Delta Angle**〞

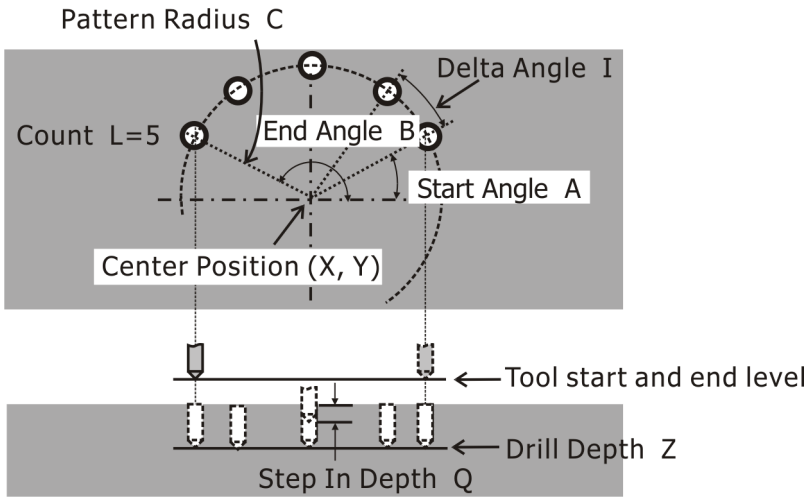


Figure 3-22 Arc Drill Pattern

〝**I Delta Angle**〞: the angle between two consecutive drilling position.

〝**L Count**〞: Drill pattern counts.

〝**B End Angle**〞: must be larger than 〝**A Start Angle**〞. The following figure is an example, End Angle must input 390°〔it is equal to 360°+30°〕 instead of 30°.

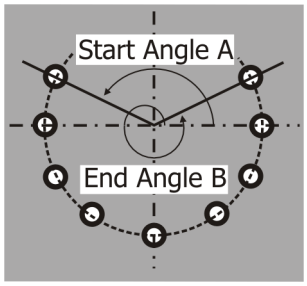


Figure 3-23 Start Angle and End Angle

When input data including of 〝**I Delta Angle**〞, 〝**L Count**〞 and 〝**B End Angle**〞, 〝**B End Angle**〞 will be ignored. After finishing the arc drilling, the tool will stop at the initial level of the final position.

IV. Object Pattern

The pocket, dowel, side surfacing and contour pocketing can be all combined with the instructions of rotation, mirror image and scaling to do each single object arrangement. This controller also provides three kinds of regular object patterns to arrange array objects: linear, grid and arc patterns. @G70.10 or @G70.11 specified the arranged object must follow with these pattern arrangement instructions.

1. Line Object Pattern

Format：**G70.21 I_J_L_**

@ **G70.10/G70.11 P_X_Y_**.....

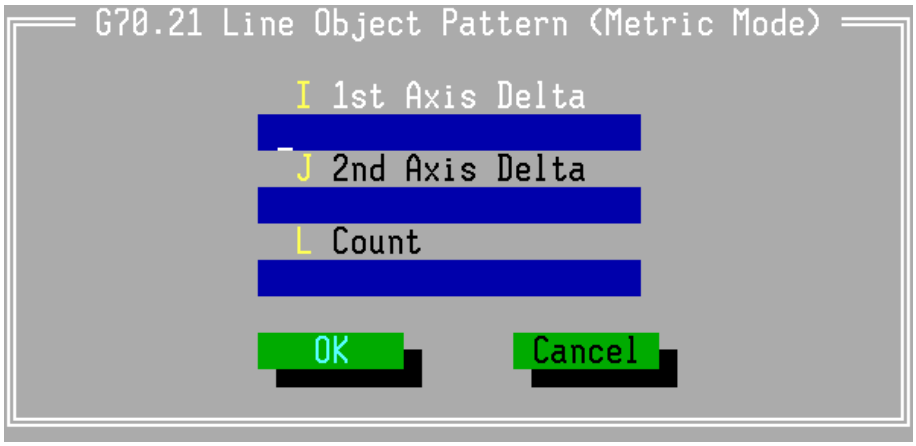


Figure 3-24 A dialog box: Line Object Pattern

〝**I 1st Axis Delta**〞 ：The distance between two consecutive object in 1st axis.
〝**J 2nd Axis Delta**〞 ：The distance between two consecutive object in 2nd axis.
〝**L Count**〞 ：Object counts. This has to be a non-negative number.

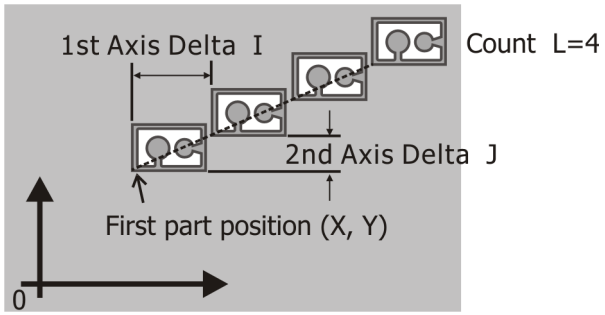


Figure 3-25 Line Object Pattern

The start position **(X,Y)** of first object is specified in the object instruction , **@ G70.10/G70.11**. After finishing **Line Object Pattern**, the tool will stop at the initial level of the final position.

2. Grid Object Pattern

Format：**G70.22 C_I_ J_A_B_(D_)**
@ G70.10/G70.11 P_X_Y_.....

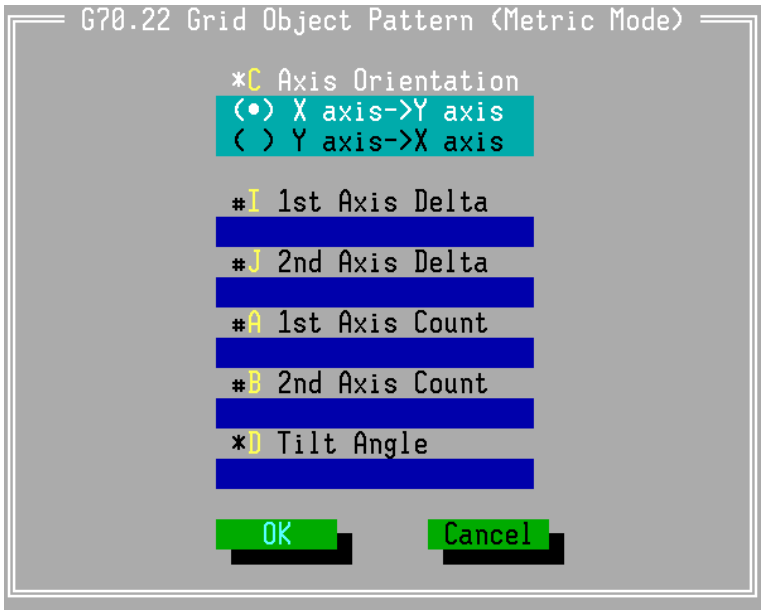


Figure 3-26 A dialog box: Grid Object Pattern

〝**C Axis Orientation**〞：Object cut orientation. C=0, cut object along 1st axis, transit along 2nd axis, otherwise, cut object along 2nd axis, transit along 1st axis.

〝**I 1st Axis Delta**〞：The distance between two consecutive object in 1st axis.

〝**J 2nd Axis Delta**〞：The distance between two consecutive object in 2nd axis.

〝**A 1st Axis Count**〞：Object counts of 1st axis. This has to be a non-negative number.

〝**B 2nd Axis Count**〞：Object counts of 2nd axis. This has to be a non-negative number.

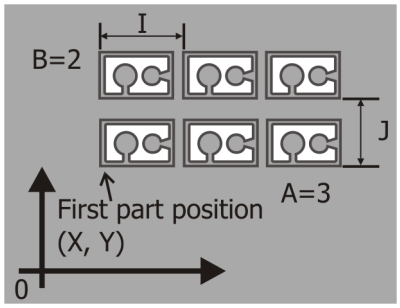


Figure 3-27 Grid Object Pattern

〝**D Tilt Angle**〞: The tilt angle specifies the rotation angle to start point of first object. The item with 〝*****〞 means it is optional.

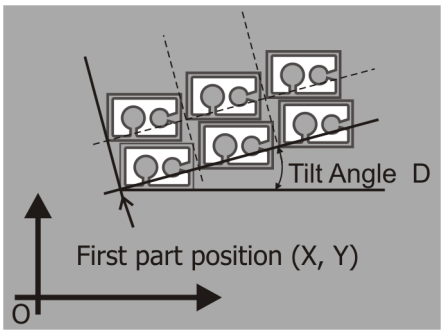


Figure 3-28 Grid Object Pattern with tilt angle

The start position **(X,Y)** of first object is specified in the object instruction , @ **G70.10/G70.11**. After finishing **Line Object Pattern**, the tool will stop at the initial level of the final position.

3. Arc Object Pattern

Format：**G70.23 (X_Y_/R_ A_)I_L_**
@ **G70.10/G70.11 P_X_Y_**.....

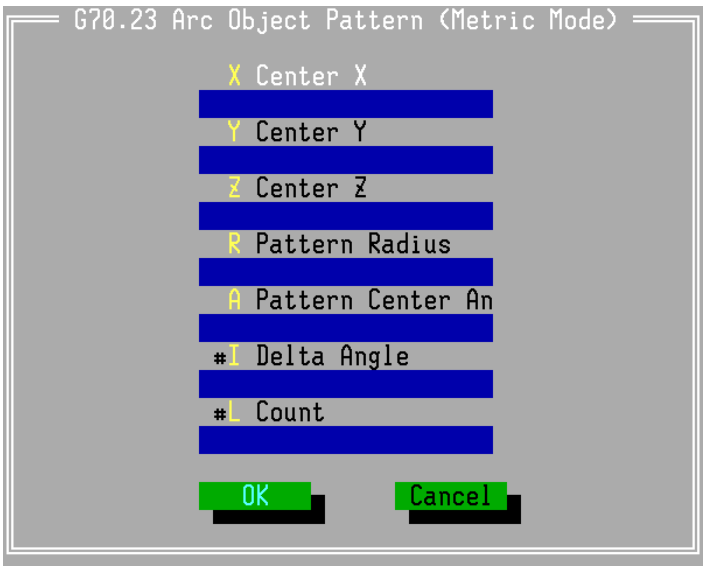


Figure 3-29 a dialog box : Arc Object Pattern

〝**X/Y/Z Center X/Y/Z**〞：Center of object pattern, which are defined by two of the axes among X, Y and Z depends on the plane specified. These are the distances from the first object to the center.

〝**R Pattern Radius**〞：Radius of object pattern. This has to be a non-negative number.

〝**A Pattern Center Angle**〞：This is the angle from the 1st axis to the vector from first object to the center of the object pattern.

〝**I Delta Angle**〞：The angle between two consecutive object. This has to be a non-negative number.

〝**L Count**〞：Object counts. This has to be a non-negative number.

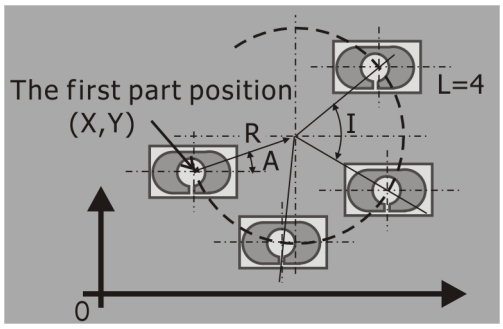


Figure 3-30 Arc Object Pattern

The start position **(X,Y)** of first object is specified in the object instruction , @ **G70.10/G70.11**. After finishing **Line Object Pattern**, the tool will stop at the initial level of the final position.

V. Common Characteristics in Side surfacing/ Pocketing/ Dowel

There are some common characteristics in the side surfacing, pocketing and dowel commands. These characteristics are defined as follows:

1. Corner Rounding

This function adds arcs of radius R to the four corners of rectangular. It is applied to the contour of rectangular only in side surfacing or pocketing or dowel commands.

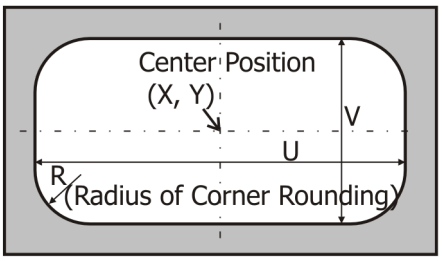


Figure 3-31 Corner Rounding

2. Chamfering

This function adds chamfers to the four corners of rectangular. It is applied to the contour of rectangular only in side surfacing or pocketing or dowel commands.

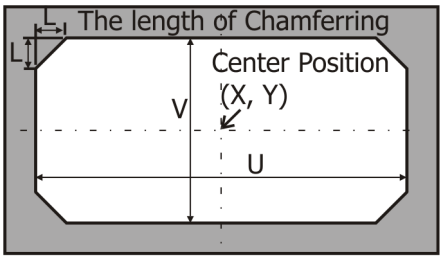


Figure 3-32 Chamfering

3. Tilt

This function tilt the contour of rectangular and track field with a tilt angle. It is applied to both rectangular and track field in side surfacing or pocketing or dowel commands.

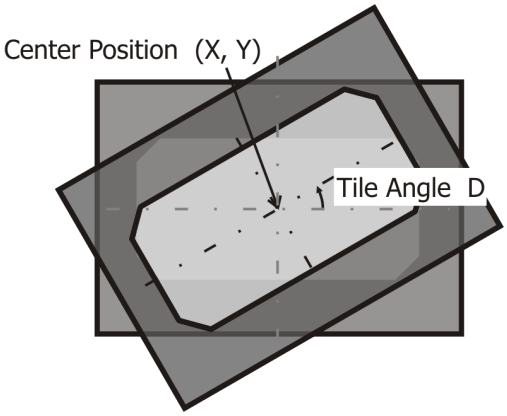


Figure 3-33 Tilt

VI. **Pocketing**

There are two pocketing instructions: rectangular and circular pocket .On using these pocket instructions, some common rules must be noticed:

(1) 〝**G Orientation** 〞〔**CW/CCW**〕will create **G02/G03** in the command line, and the five selection items in 〝**P Step In Axis**〞will create **P0~P4** in the command line.
(2) Before the pocket cutting, the tool table number〔**D_**〕and the radius of the tool must be set previously so that the system can get the correct radius data for pocket cutting.
(3) Pocket commands are classified to **one shot command** means each block of pocketing must be specified by **G70.xx**. For continuous blocks of pocketing, when programming directly by **Editor** instead of Command Insertion Aid, the **G70.00/70.01** and the new central position **(X,Y,Z,)** of the pocket must be specified. The other variables will use the previous setting.
(4) The d value〔d means cut in retract length〕in the figures is the same as one specified in the 〝**parameter→cancycle parameters→Cut In Retract Length**〞, so don't specify it again.
(5) When fine feed width〔**J_**〕and fine feed depth〔**K_**〕haven't been specified, the last fine cutting isn't performed. Under **G70.00** mode, when **L_/R_** hasn't been specified, corner chamfering /rounding isn't performed.
(6) When **E_** hasn't been specified，fine feed rate is performed with half of **F_** instead of **E_**.
(7) After finishing the pocket cutting, the tool will retract to the pocket center and the initial level.

The motions of the instruction are as below:

(1) The tool moves rapidly to the pocket center **(X,Y)** and the safe level **(Z+C)**.
(2) Cut in depth 〝**Q Step In Depth**〞 for each drilling step in speed 〝**I Spindle Feed Rate**〞.
(3) Do pocket cutting in speed 〝**F Feed Rate**〞according to selected orientation.
(4) When finishing pocketing the current layer, the tool retract at the initial level and positioning to the pocket center.
(5) Cut in depth〔**Q_**〕again and do pocketing the next layer in speed 〝**I Spindle Feed Rate**〞.

(6) Repeat the steps (1)~(5) until reach the specified depth〔**W_**〕.
(7) Do fine cutting in speed 〝**E Fine Feed Rate**〞 finally.
(8) When finishing pocketing, the tool retract at the initial level and stop at the pocket center.

Pocket cutting each layer is broadened from the pocket center circle by circle. The broadened width is specified by 〝**Parameter→Cancycle Parameters→Step In Width**〞, which provides 4 selections 150%, 125%, 100% and 75%.

1. Rectangular Pocket

Format：

```
G70.00 (G02/G03) P_X_Y_Z_U_V_W_C_J_K_Q_(R_/L_)D_I_E_F_
```

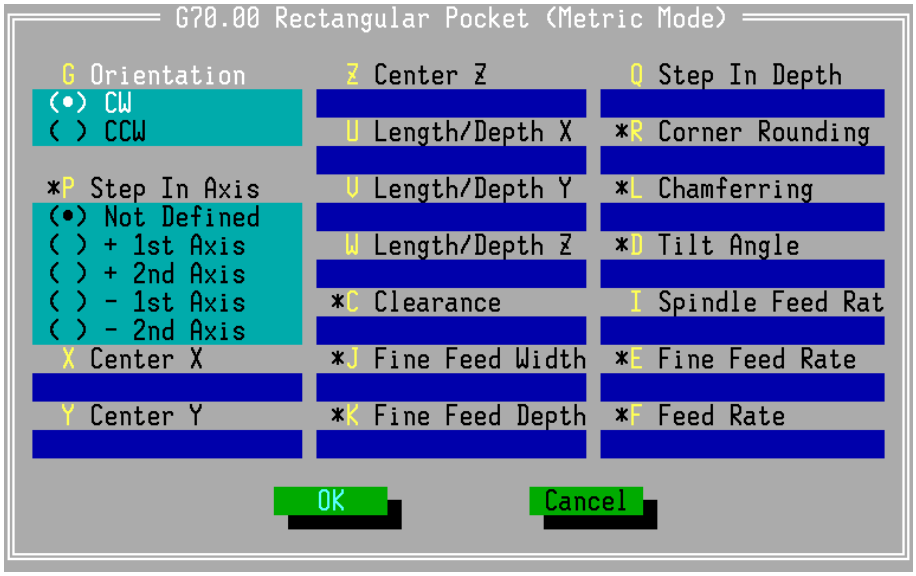


Figure 3-34 A dialog box: Rectangular Pocket

(1) The pocket center **(X,Y)** and initial level **Z_** can program in absolute or incremental dimension
(2) **U_/V_/W_** : Pocket length, width, and depth, which are defined respectively by **U_**, **V_** and **W_** depends on which plane is specified. Length and width have to be non-negative number. The sign of the depth determines the direction of drilling.
(3) The radius of the tool number in 〝**Parameter→Tool Table**〞 must be set previously, or rectangular pocket doesn't run.

(4) Corner rounding and chamfering are optional items. When both are specified, chamfering will be ignored.

(5) When **E_**〔Fine Feed Rate〕 hasn't been specified，fine feed rate is performed with half of **F_**〔Feed Rate〕instead of **E_**.

(6) 〝**D Tilt Angle**〞: Pocket tilt degree. This is the angle rotates from the rectangular pocket center with CCW orientation.

Rectangular Pocket with corner rounding:

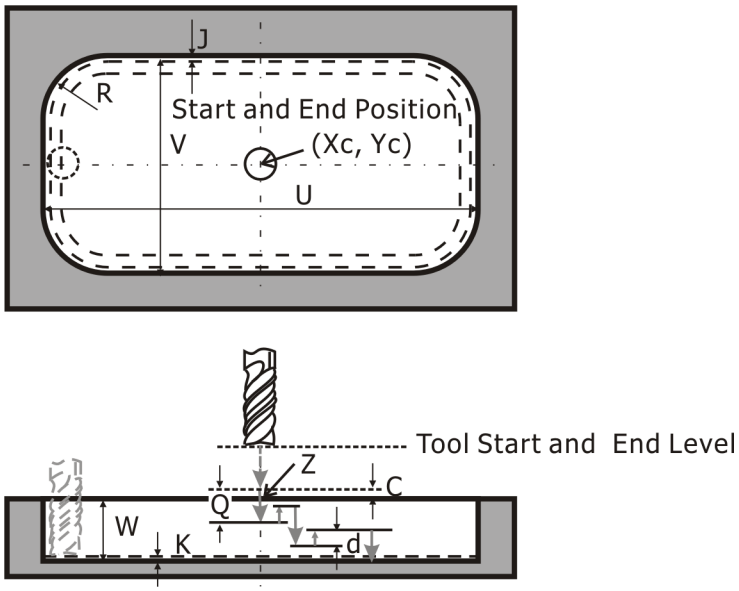


Figure 3-35 Rectangular Pocket with corner rounding

Rectangular Pocket with chamfering：

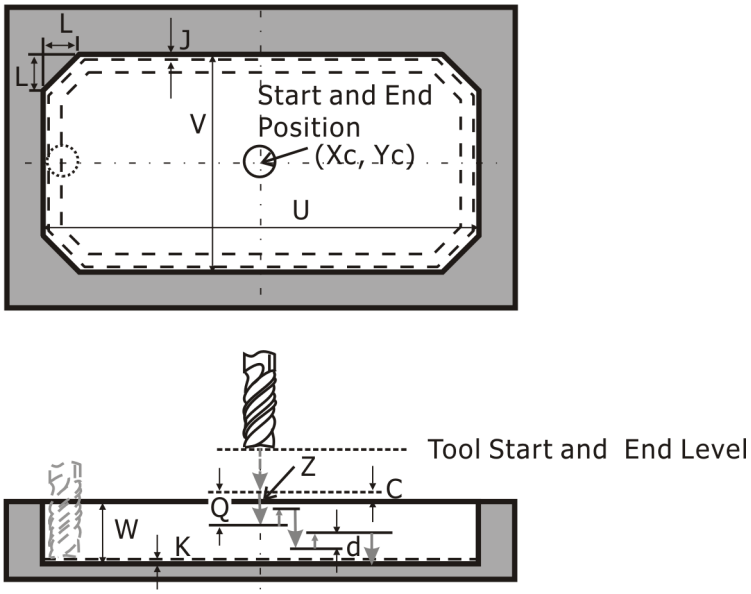


Figure 3-36 Rectangular Pocket with chamfering

Rectangular Pocket with chamfering and tile angle:

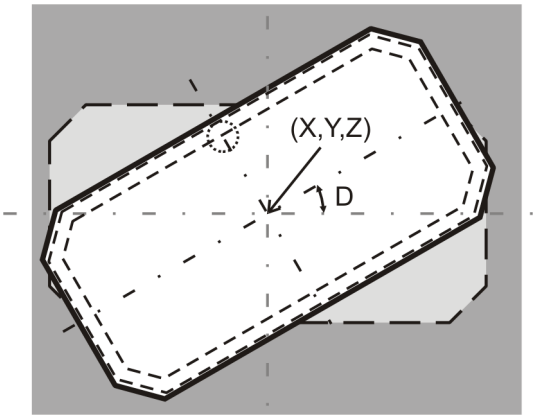


Figure 3-37 Rectangular Pocket with chamfering and tile angle

2. Circular Pocket

Format： **G70.01 (G02/G03)(P_) X_Y_Z_R_U_(C_ J_ K_)Q_I_(E_ F_)**

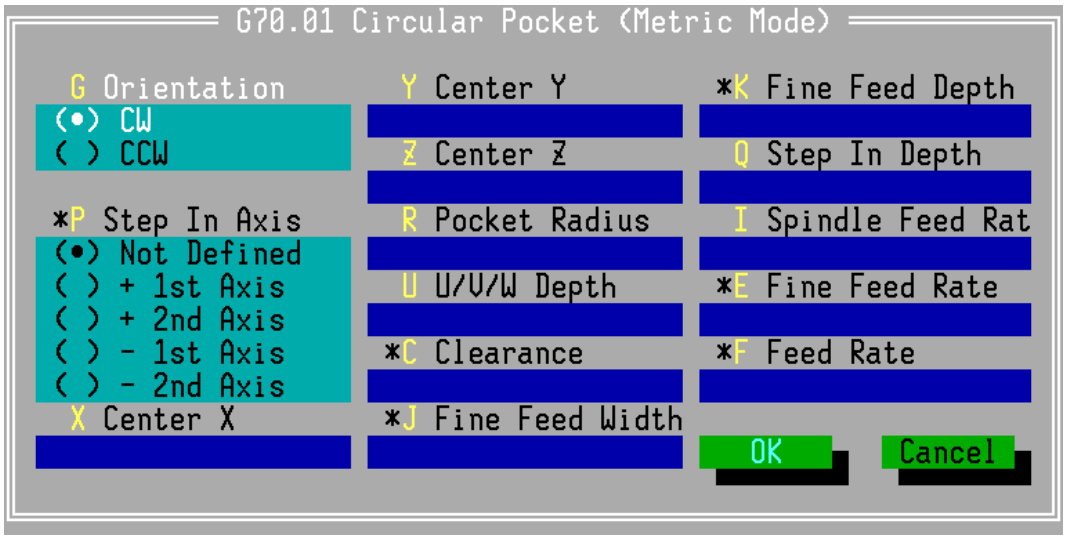


Figure 3-38 A dialog box: Circular Pocket

(1) The pocket center **(X,Y)** and initial level **Z_** can program in absolute or incremental dimension

(2) 〝**R Pocket Radius**〞: Pocket corner rounding radius.

(3) The radius of the tool number in 〝**Parameter→Tool Table**〞 must be set previously, or rectangular pocket doesn't run.

(4) When **E_**〔Fine Feed Rate〕 hasn't been specified，fine feed rate is performed with half of **F_**〔Feed Rate〕instead of **E_**.

(5) 〝**G Orientation**〞:Cutting orientation. **G02** for CW, **G03** for CCW.

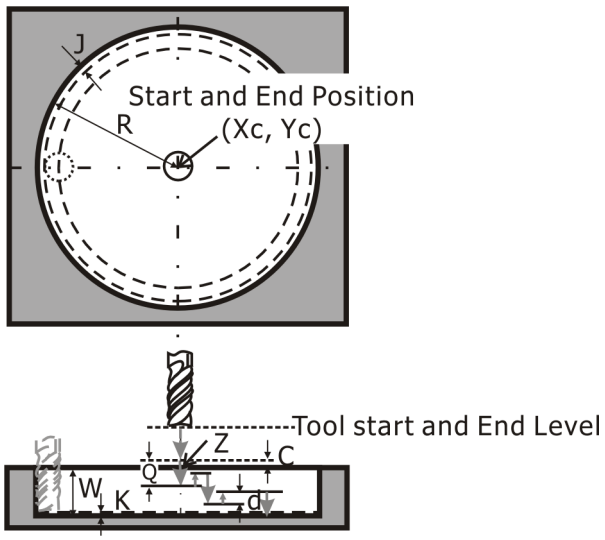


Figure 3-39 Circular Pocket

VII. Dowel

There are two Dowel cutting: rectangular and circular Dowel.

The motions of these instructions are as below:

(1) The tool moves rapidly to the start position **(X$_S$,Y$_S$)**, which is calculated automatically by the input value **(X,Y,U,V)**, and safe level **(Z+C)**.
(2) Cut in depth 〝**Q Step In Depth**〞 for each drilling step in speed 〝**I Spindle Feed Rate**〞.
(3) Do Dowel cutting in speed 〝**F Feed Rate**〞 according to selected orientation.
(4) When finishing Dowel cutting the current layer, the tool retract at the initial level and positioning to the dowel center.
(5) Cut in depth〔**Q_**〕again and do doweling the next layer in speed 〝**I Spindle Feed Rate**〞.
(6) Repeat the steps (1)~(5) until reach the specified depth〔**W_**〕.
(7) Do fine cutting in speed 〝**E Fine Feed Rate**〞 finally.
(8) When finishing doweling, the tool retracts at the initial level and stops at the dowel center **(X,Y)**.

Dowel cutting each layer is broadened from the pocket center circle by circle. The broadened width is specified by 〝**Parameter→Cancycle Parameters→Step In Width**〞, which provides 4 selections 150%, 125%, 100% and 75%

1. Rectangular Dowel

Format：**G70.02(G02/G03)(P_)X_Y_Z_U_V_W_A_B_(C_ J_K_)Q_(L_/R_ D_) I_(E_F_)**

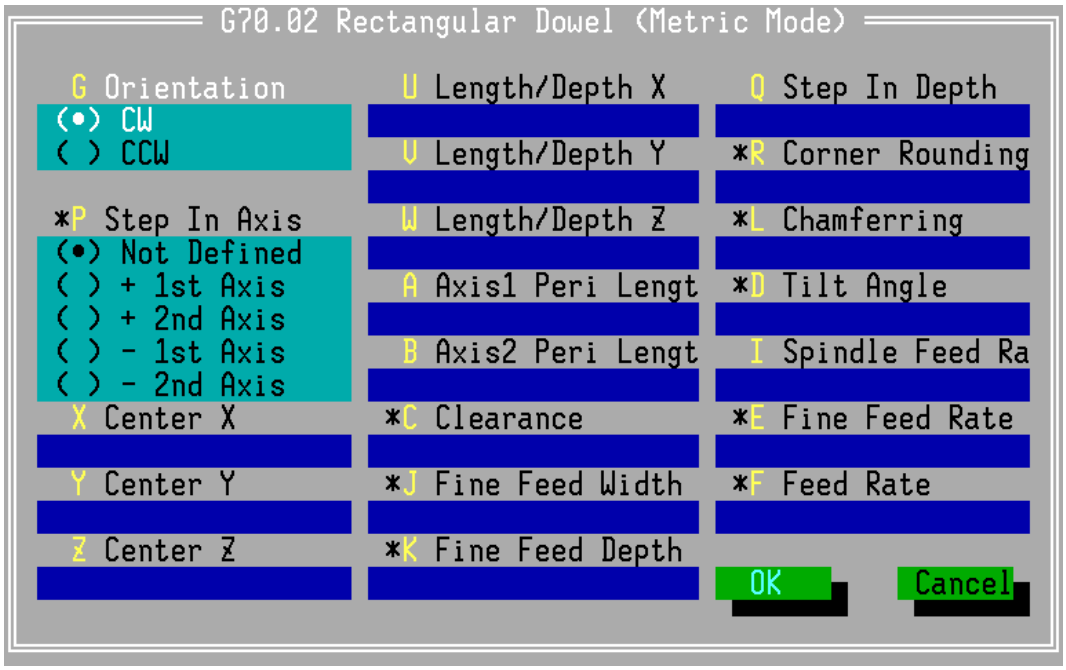


Figure 3-40　A dialog box: Rectangular Dowel

(1) The dowel center **(X,Y)** and initial level **Z_** can program in absolute or incremental dimension

(2) **U/V**: dowel length, width.

(3) **A/B**: Perimeter length and width.
Keep in mind: The value **A/B** must be larger than **U/V**, or the program error occurs and doesn't work.

(4) 〝**D Tilt Angle**〞: Dowel tilt degree. This is the angle rotates from the rectangular dowel center with CCW orientation.

(5) When **E_**〔Fine Feed Rate〕 hasn't been specified，fine feed rate is performed with half of **F_**〔Feed Rate〕instead of **E_**.

(6) Corner rounding and chamfering are optional items. When both are specified, chamfering will be ignored.

(7) When **E_**〔Fine Feed Rate〕 hasn't been specified，fine feed rate is performed with half of **F_**〔Feed Rate〕instead of **E_**.

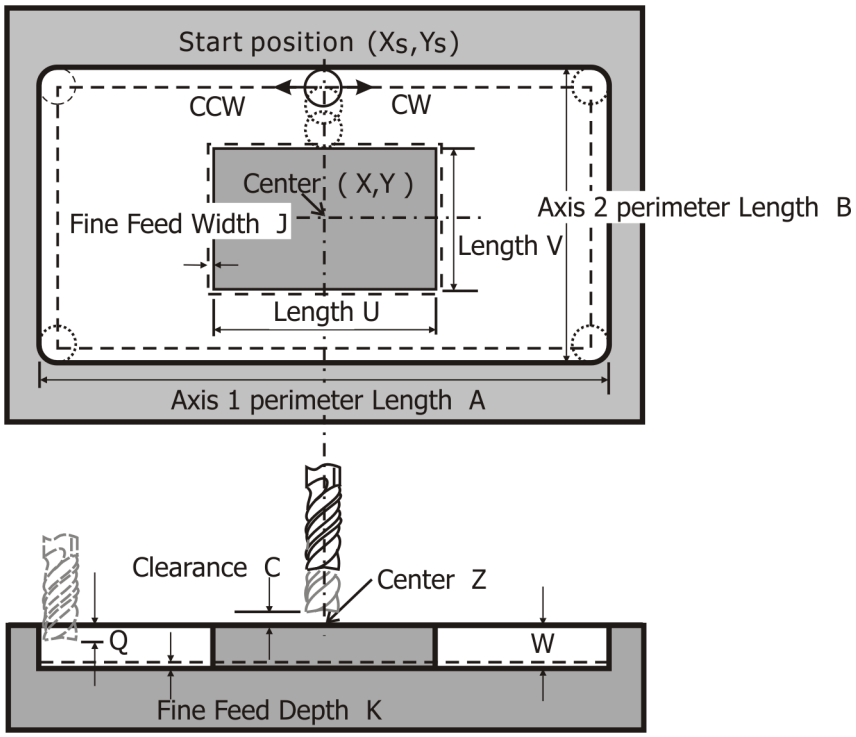


Figure 3-41 Rectangular Dowel

2. Circular Dowel

Format： **G70.03 (G02/G03) (P_)X_Y_Z_R_U_ A_(C_J_K_)Q_ I_(E_F_)**

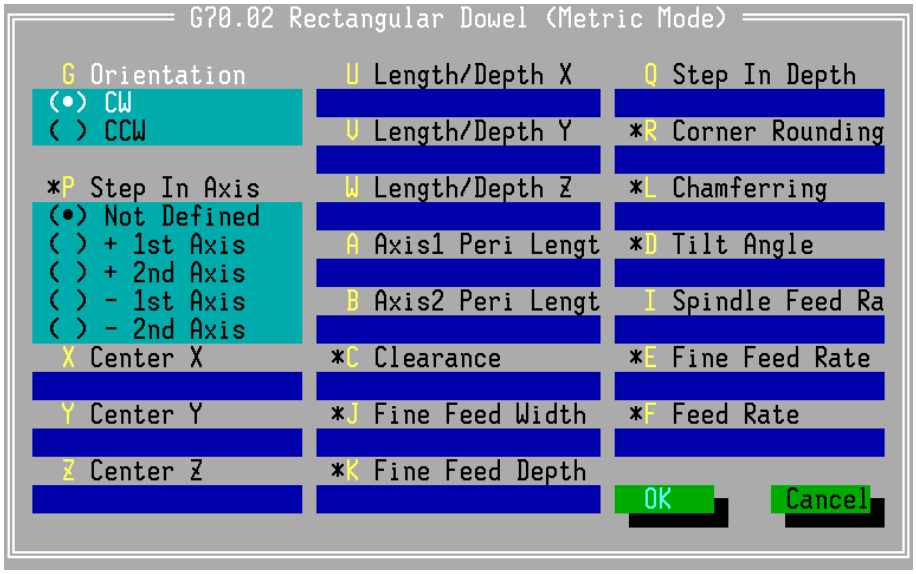


Figure 3-42 A dialog box: Circular Dowel

(1) The dowel center **(X,Y)** and initial level **Z_** can program in absolute or incremental dimension

(2) **R_** is dowel corner rounding radius. **A_** is Perimeter radius.

(3) Keep in mind: 〝**A Perimeter Radius**〞 must be larger than 〝**R Dowel Radius**〞 ,or the program error occurs and doesn't work.

(4) The radius of the tool number in 〝**Parameter→Tool Table**〞 must be set previously, or rectangular pocket doesn't run.

(5) When **E_**〔Fine Feed Rate〕 hasn't been specified，fine feed rate is performed with half of **F_** 〔Feed Rate〕instead of **E_**.

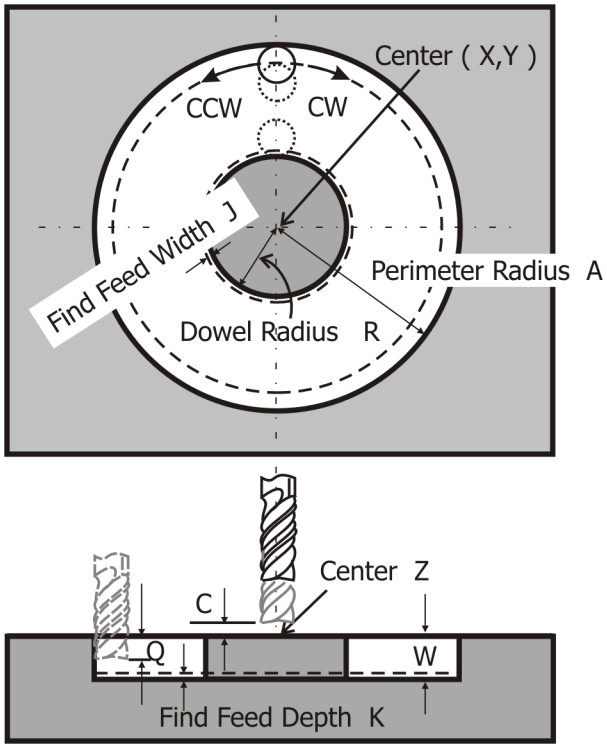


Figure 3-43 Circular Dowel

VIII. Side Surfacing

There are two side surfacing: outside surfacing and inside surfacing.

Every side surfacing provides two types of contour: rectangular and circular.

The motions of these instructions are as below:

(1) The tool moves rapidly to the start position **(Xs,Ys)**, which is calculated automatically by the input value **(X,Y,U,V)**, and safe level **(Z+C)**.

(2) Cut in depth 〝**W Length / Depth Z**〞, and do side surfacing with selected orientation in speed 〝**I Spindle Feed Rate**〞.

(3) When finishing side surfacing, the tool retracts at the initial level and stops at the Side surfacing center **(X,Y)**.

Note:

(1) The start position is located at the distance of the radius twice from the contour of side surfacing. Keep in mind: don't make a mistake to cut some part of the work piece at the distance.

(2) When starting side surfacing, the tool cuts into the surface of side surfacing with a semi-circle path. When finishing side surfacing, the tool leaves out the the surface of side surfacing with a semi-circle path.

1. Rectangular Outside / Inside Surfacing

Format： **G70.04 (G02/G03)I_(P_)X_Y_Z_U_V_W_ (L_/R_ D_ F_)**

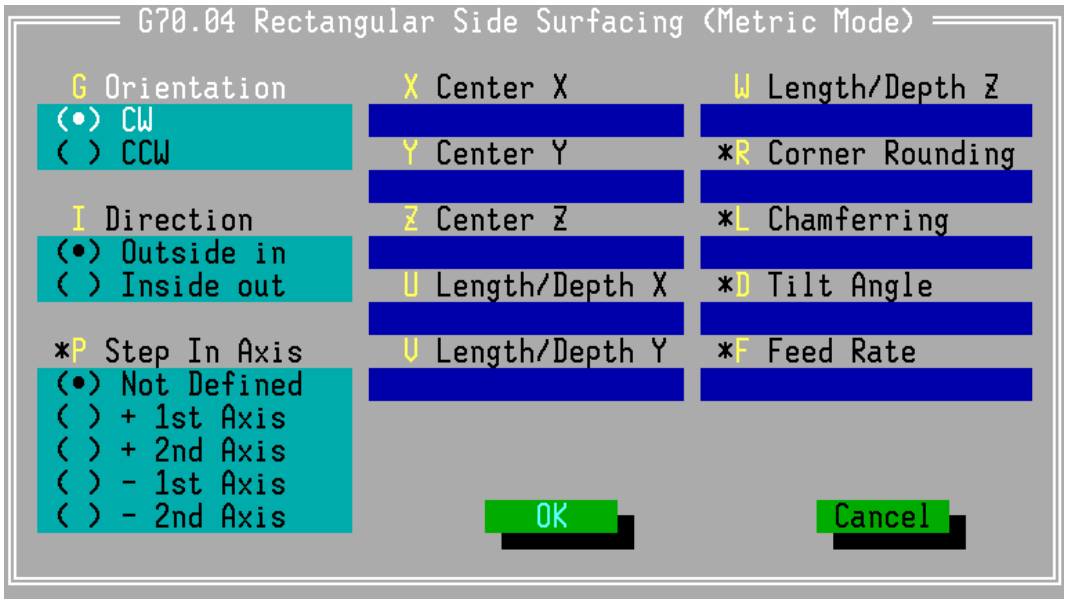


Figure 3-44 A dialog box: Rectangular side surfacing

Rectangular Outside Surfacing and **Rectangular Inside Surfacing** are both set by the dialog box.

〝**I Direction**〞: Rectangular Side Surfacing direction. **I0** for outside surfacing〔from outside toward inner〕, otherwise, **I1** for inside surfacing〔from inside toward outer〕.

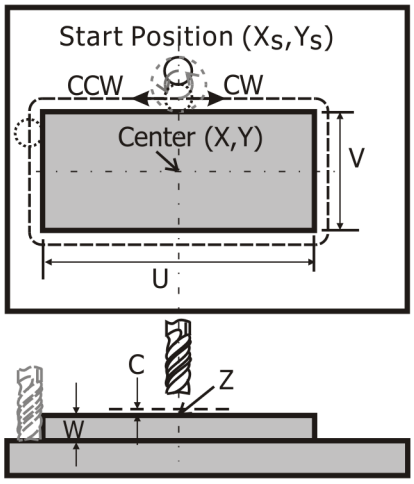


Figure 3-45 Rectangular Outside Surfacing

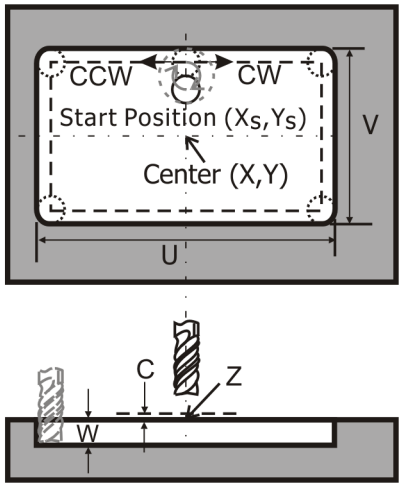


Figure 3-46 Rectangular Inside Surfacing

(1) The rectangular side surfacing center **(X,Y)** and initial level **Z_** can program in absolute or incremental dimension
(2) **U/V/W** : Side surfacing length, width, and depth.
(3) 〝**D Tilt Angle**〞: Side surfacing tilt degree. This is the angle rotates from the rectangular side surfacing center with CCW orientation.
(4) The radius of the tool number in 〝**Parameter→Tool Table**〞 must be set previously, or rectangular side surfacing doesn't run.
(5) Corner rounding and chamfering are optional items. When both are specified, chamfering will be ignored.

2. **Circular Outside / Inside Surfacing**

Format：**G70.05 (G02/G03) I_(P_)X_Y_Z_ R_U_(F_)**

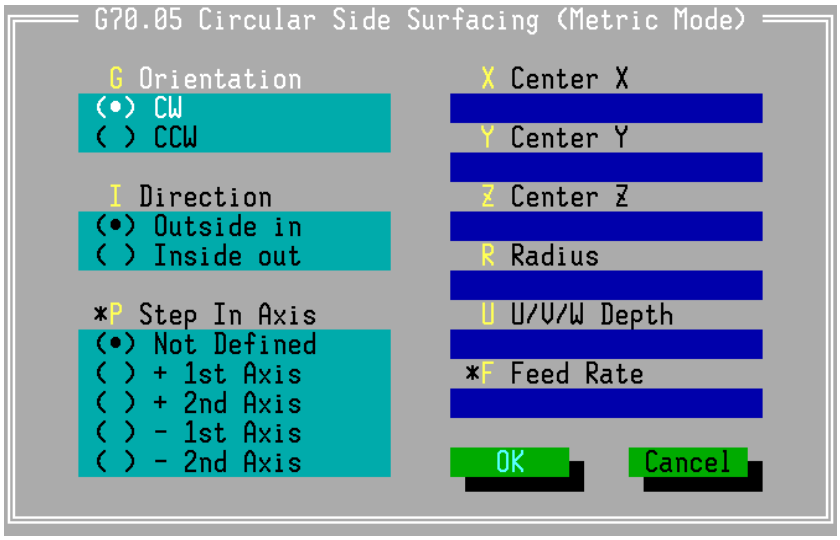


Figure 3-47 A dialog box: Circular Side Surfacing

〝**I Direction**〞: Circular Side Surfacing direction. **I0** for outside surfacing〔from outside toward inner〕, otherwise, **I1** for inside surfacing〔from inside toward outer〕.

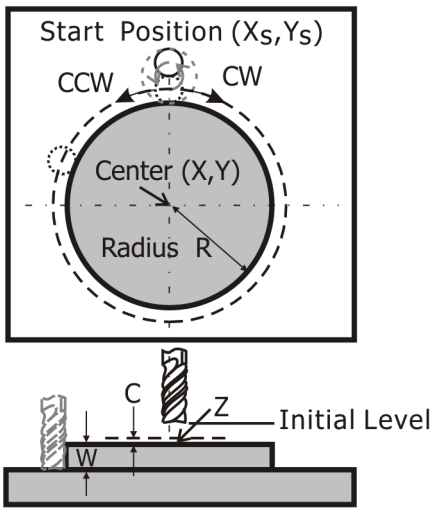


Figure 3-48 Circular Outside Surfacing

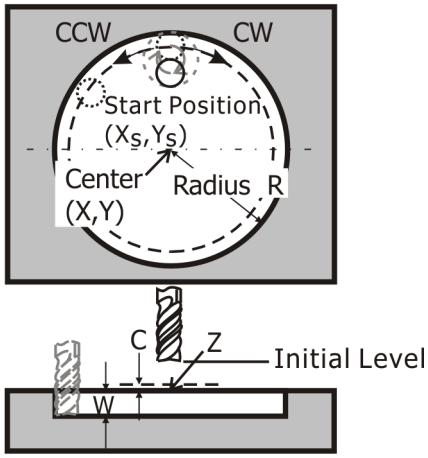


Figure 3-49 Circular Inside Surfacing

(1) The rectangular side surfacing center **(X,Y)** and initial level **Z_** can program in absolute or incremental dimension

(2) **U/V/W**: Side surfacing length, width, and depth.

(3) The radius of the tool number in 〝**Parameter→Tool Table**〞 must be set previously, or rectangular side surfacing doesn't run.

IX. Surfacing

Format： **G70.06 D_X_Y_Z_U_V_W_C_I_F_**

There are four surfacing mode:

D=0： cut along X axis, vertical transition
D=1： cut along Y axis, vertical transition
D=2： cut along X axis, tilt transition
D=3： cut along Y axis, tilt transition

The motions of the instruction are as below:

(1) The tool moves rapidly to the start position **(X_s,Y_s)**, which is calculated automatically by the input value **(X,Y,U,V)**, and safe level **(Z+C)**.
(2) Cut in depth 〝**W Length / Depth Z**〞, and do surfacing with selected orientation in speed 〝**I Spindle Feed Rate**〞.
(3) When finishing Surfacing, the tool retracts at the initial level and stops at the Surfacing center **(X,Y)**.

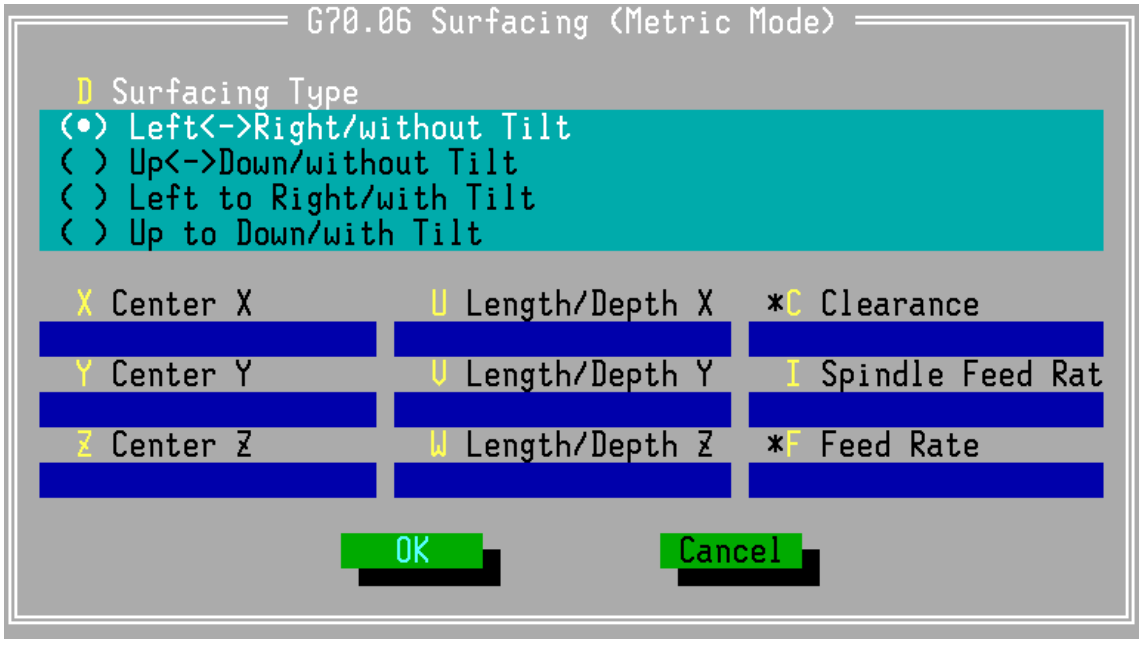


Figure 3-50 A dialog box: Surfacing

(1) The surfacing center **(X,Y,Z)** can program in absolute or incremental dimension
(2) **U_/V_** is defined the length of X / Y axis of surfacing rectangle.
(3) During Surfacing, ，The width per step in is set by 〝**Parameter→Cancycle Parameter→Step In Width**〞, which provides four selections ,150%, 125%, 100% and 75%.

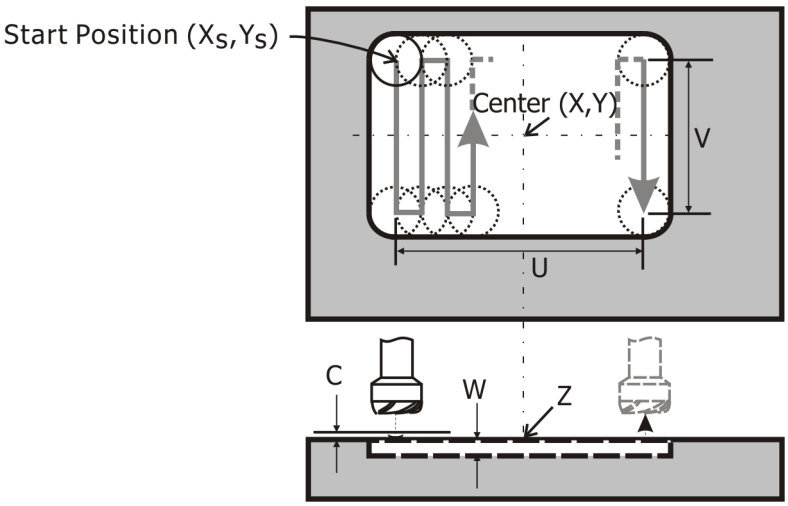


Figure 3-51 Surfacing〔D1 mode〕

1. Surfacing , Left↔Right / without Tilt〔D0〕

Upon the tool does surfacing and shifts the distance **U_**along X axis, the tool turns to Y axis and shifts the distance 〝**Step In Width**〞. Then the tool dose surfacing and shifts back the distance **U_**along X axis, the tool turns to Y axis and shifts the distance 〝**Step In Width**〞again. Then the tool does along X axis again..... Do the above-mentioned motions again and again until finish all surfacing process.

2. Surfacing , Up↔Down / without Tilt〔D1〕

Upon the tool does surfacing and shifts the distance **V_**along Y axis, the tool turns to X axis and shifts the distance 〝**Step In Width**〞. Then the tool dose surfacing and shifts back the distance **V_**along Y axis, the tool turns to X axis and shifts the distance 〝**Step In Width**〞again. Then the tool does along Y axis again..... Do the above-mentioned motions again and again until finish all surfacing process.

3. Surfacing , Left↔Right / with Tilt〔D2〕

Upon the tool does surfacing and shifts the distance **U_**along X axis, the tool goes back with a hypotenuse transition 〔the hypotenuse has two legs of right triangle, **U_** and the distance〝**Step In Width**〞along Y axis〕. Do the above-mentioned motions again and again until finish all surfacing process.

4. Surfacing , Up↔Down / with Tilt〔D3〕

Upon the tool does surfacing and shifts the distance **V_**along Y axis, the tool goes back with a a hypotenuse transition 〔the hypotenuse has two legs of right triangle, **V_** and the distance〝**Step In Width**〞along X axis〕. Do the above-mentioned motions again and again until finish all surfacing process.

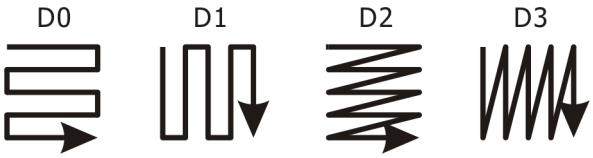


Figure 3-52 Four modes of surfacing

X. Contour Pocket / Contour Pocket Side Surfacing

Any closed loop contour can be pocketed layer by layer or do the side surfacing with the tool radius compensation.

1. Contour Pocket

Format：**G70.10 (G41/G42) P_X_Y_Z_(U_V_W_)C_J_K_Q_R_I_E_F_**

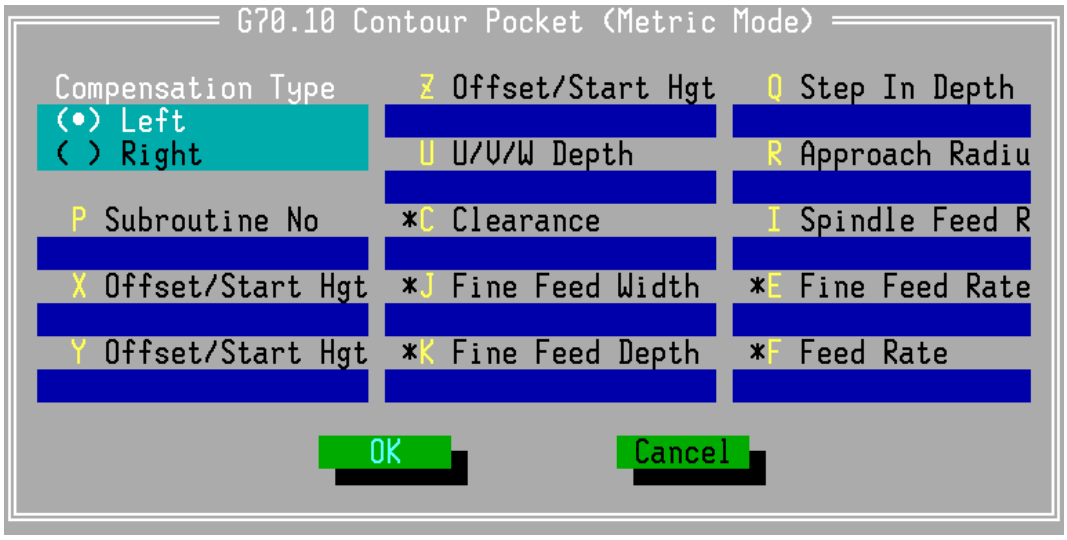


Figure 3-53 A dialog box: Contour Pocket

P : Subroutine number of the contour. The subroutine follows main program and it is named by **Oxxxx** .

X,Y,Z : Contour offset and start position, which are defined respectively by X, Y and Z depends on the plane specified. The offset is the distance of the contour pocket position from the standard contour pocket defined in subroutine **P**.

C : Safety clearance, which is the distance to the pocket start position.

U/V/W : Contour depth, which is defined by **U_**, **V_** or **W_** depends on which plane is specified. The sign of the depth determines the direction of drilling.

J : Fine feed width. Motion in this stage is with fine feed rate. This has to be a non-negative number.

K : Fine feed depth. Motion in this stage is with fine feed rate. This has to be a non-negative number.

Q : Cut in depth, which is the infeed amount for each drilling step.

R : Approach radius, which is the radius to approach for fine finish.

I : Spindle axis feed rate.

E : Fine feed rate. This is a modal state. This has to be a non-negative number.

F : Feed rate. This is a modal state. This has to be a non-negative number.

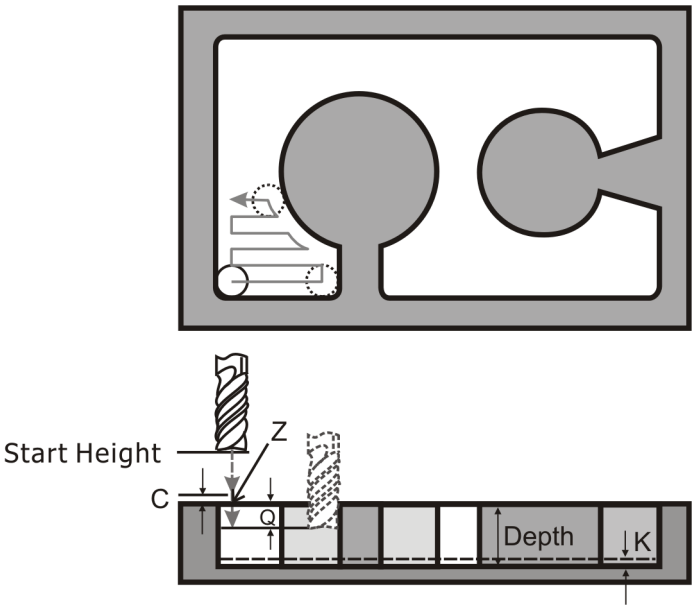


Figure 3-54 Contour Pocket

2. Contour Pocket Side Surfacing

Format：**G70.11 P_X_Y_Z_(U/V/W_)(C_)R_I_(F_)**

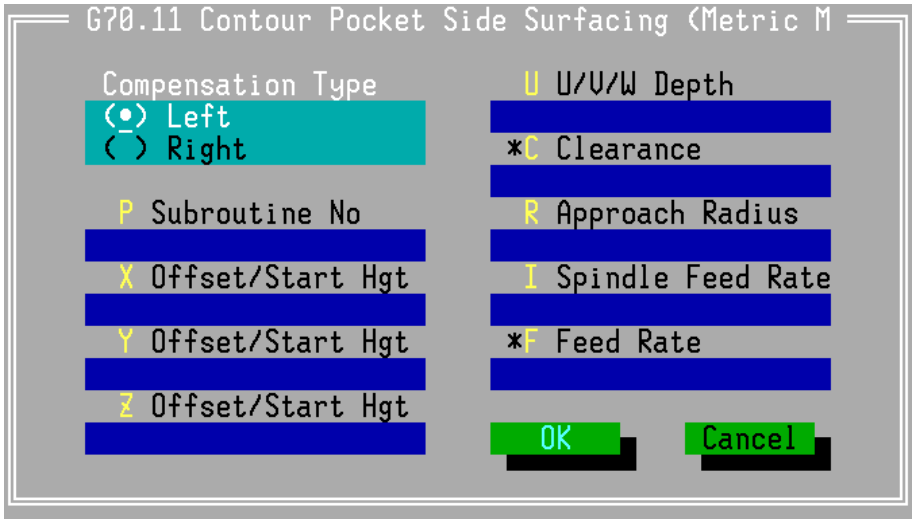


Figure 3-55 A dialog box: Contour Pocket Side Surfacing

P ：Subroutine number of the contour. The subroutine follows main program and it is named by **Oxxxx** .

X,Y,Z：Contour offset and start position, which are defined respectively by X, Y and Z

depends on the plane specified. The offset is the distance of the contour pocket position from the standard contour pocket defined in subroutine **P**.

C ：Safety clearance, which is the distance to the pocket start position.

U/V/W： Contour depth, which is defined by **U_**, **V_** or **W_** depends on which plane is specified. The sign of the depth determines the direction of drilling.

R ：Approach radius, which is the radius to approach for fine finish.

I : Spindle axis feed rate. This has to be a non-negative number.

F : Feed rate. This has to be a non-negative number.

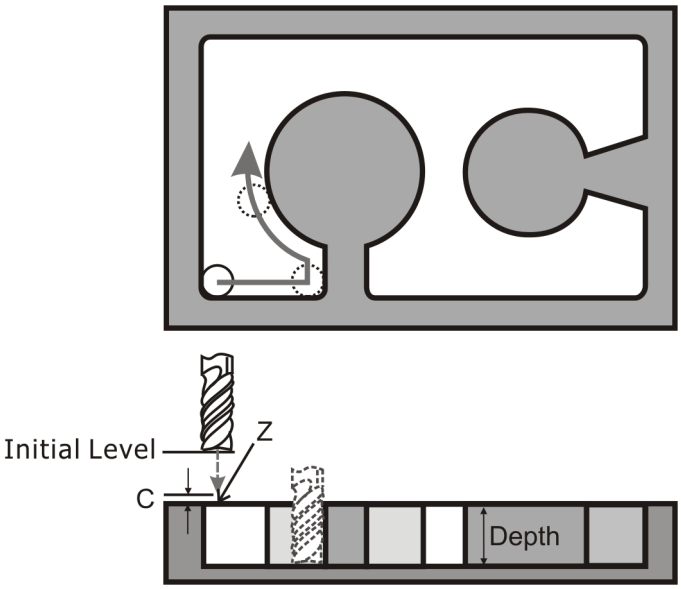


Figure 3-56 Contour Pocket Side Surfacing

3.2.3.2 Menu Item : Edit→Command Modification Aid

Select this menu item to pop up a dialog box which shows the detailed information about the instruction pointed to by editor cursor. This item can help users to modify motion instructions. The descriptions of the hot keys are as below:

F3 Mod & Nxt **:** Modify the current line and the dialog box pops the next line.

F4 No & Nxt **:** Don't modify the current line and the dialog box pops the next line.

F5 Mod & Prv **:** Modify the current line and the dialog box pops the previous line.

F6 No & Prv **:** Don't modify the current line and the dialog box pops the previous line.

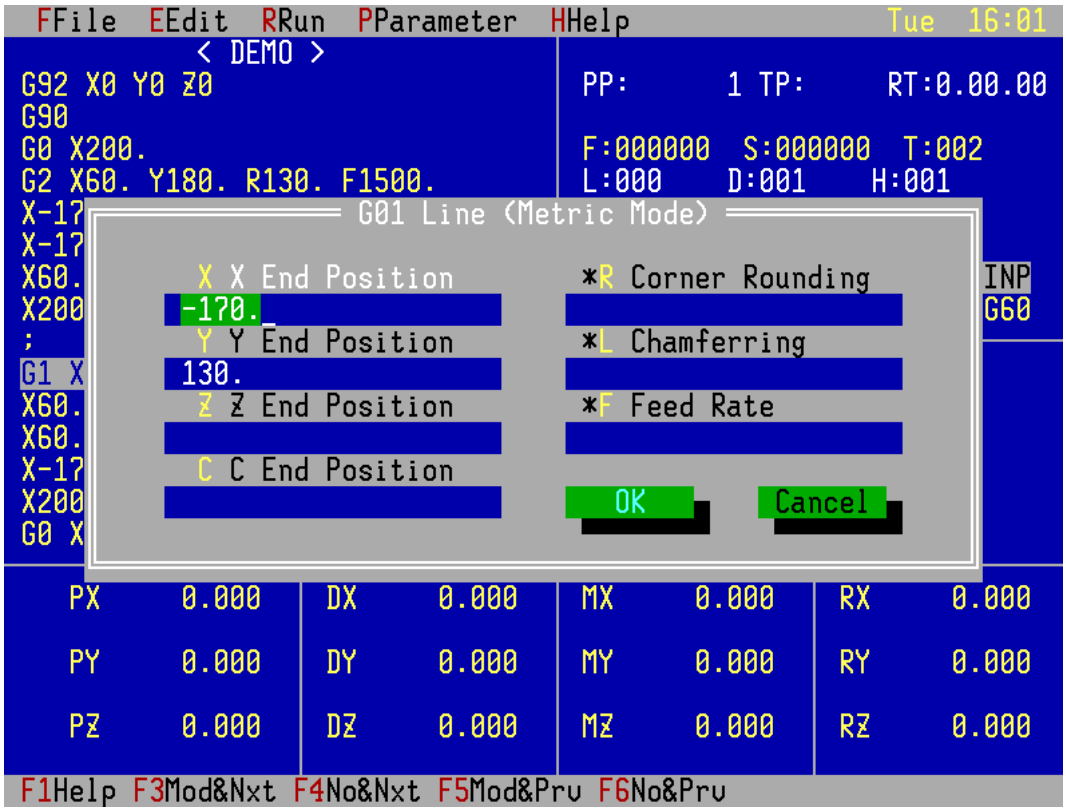


Figure 3-57 A dialog box: Command Modification Aid

Note: The function can be used under the **Edit** mode.

Press **Esc** to leave the **Edit** mode after finishing editing. And then press 〝**F2→ F→S**〞 to save the current program. You can also do it by the hot key **F3**.

3.2.4 Menu Item : Edit→Teach In

You can do manual operation when you edit a CNC program. Move the tool to touch the surface of part model and press **Alt+T** to teach in program position on the specified axis.

Select this menu item to create a command line with the program position of each specified axis, then insert this command line into the editor at the cursor position. The teach in axis is specified in set teach in axis parameter.

For instance: The current program position is **(123.000, 100.120)**

The inserted command line will be **X123.000 Z100.120**, when all axis are specified.

The command line will be **X123.000**, when only X axis is specified.

3.2.5 Compute Radius

We know a geometric rule : A circle is defined by three points not in a line .

INCON-ME3 provides a function which computes radius of this arc defined by three points not in a line . Locate the cursor at the next line of the command lines of three points, select this menu item or press **Alt+R** , so the radius will be calculated and the 2nd,3rd line will be replaced by computed **G02** or **G03** code automatically.

This function works under absolute dimension mode. Both **X_** and **Y_** in the command lines of three points must exist.

System alarms when three point specified locates in a line.

Here is example:

Before computing radius :

```
G00 X+000.000 Y+000.000
G01 X+003.000 Y+004.000
G01 X+000.000 Y+004.000
```

Locate cursor right after the third line and press **Alt+R**. The program will be :

```
G00 X+000.000 Y+000.000
G02 X+003.000 Y+004.000 R+002.500
G02 X+000.000 Y+004.000 R+002.500
```

3.3 Run Program〔F2→R〕

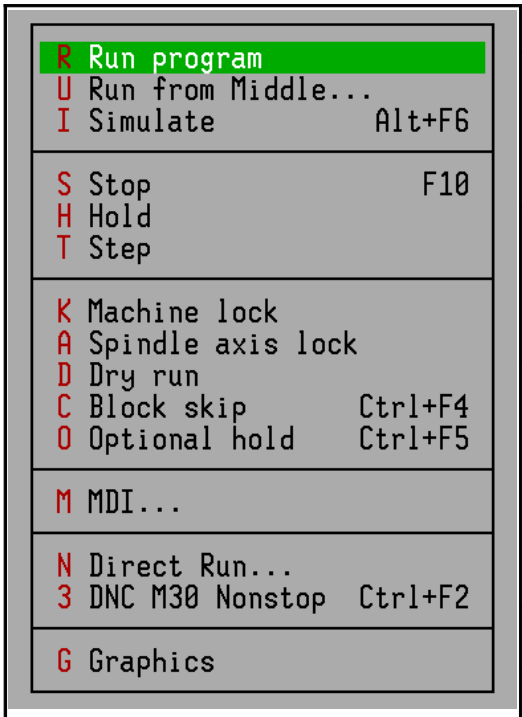


Figure 3-58 The Run submenu

Description
R：Program execution U：Execute program from the cursor pointed line I ：Simulate program execution
S：Stop running program H：Pause running program T：Toggle switch，run **Single Step** or continuously
K：Toggle switch，switch machine lock or unlock A：Toggle switch, spindle axis lock/dislock motion D：Toggle switch，switch **Dry run** or not C：Toggle switch，switch **Block skip** or not O：Toggle switch， switch **M01** running or not
M：Manual Direct Input
N：Direct run .DNC program from hard disk 3：Ignore M30 in direct running under DNC mode
G：Switch to graphic mode or not

〝**Toggle switch**〞 is the same as state switch, each being selected will change to one mode or the other mode, such as **Off→On** and **On→Off**; **Disable→Enable** and **Enable→Disable**.

3.3.1 Menu Item: Run→Run program

Select this menu item to execute a program according to the run state defined in 〝**Parameter→Default Status→Set Run Mode**〞.

Run program starts from the current cursor position. Commands before the current cursor position, such as **Status Change G-code** and **Motion G-code** are all ignored because of no executing program from the beginning.

Warning: Watch out for the safety on machine limits before start to run.

3.3.2 Menu Item: Run→Run From Middle

Select this menu item to emulate a fast run of program from the beginning to the line and task number of selected file according to the run state defined in the run state item of user parameter. This way, INCON-ME3 system can obtain the status for program to run. After the fast emulation run is over, a dialog box will pop up for user to confirm program execution or abort.

Warning: Watch out for the safety on machine limits!

When in program run state, 〝**Run from middle**〞 will restore the conditions, except M,S, and T codes, which are set by the program of previous to the cursor located. On selecting 〝**Run from middle**〞, it will pop out a dialog box as follows:

This dialog box is for user to input the desired CNC program name, line number, task number and hit count from which program will start running.

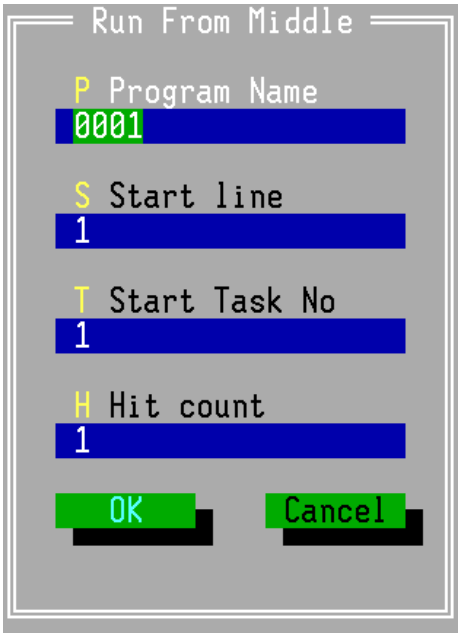


Figure 3-59 The dialog box: Run from middle

After finishing the above-mentioned dialog box, a **Confirm dialog box** will pop up to notice whether users restore M, S and T code original conditions before executing **Run From Middle**. If not, set M, S and T code conditions well by MDI. When everything is ready, the function **Run From Middle** will execute from middle line. The function is used to resume program from the interrupted command.

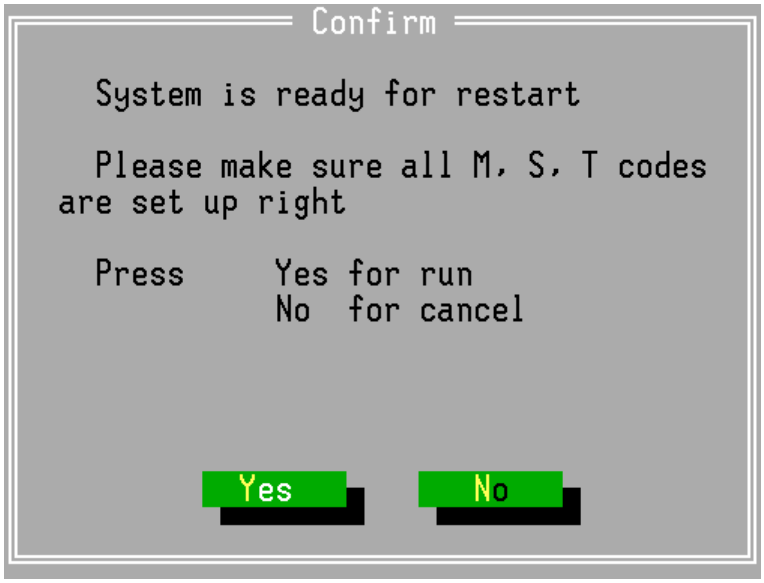


Figure 3-60 The Confirm dialog box: Run from middle

3.3.3 Menu Item : Run→Simulate

Select this menu item to simulate program run according to the run state defined in the run state item of user parameter. The simulate mode will run the system in simulate speed without actually moving the machine table. User can use this item to verify the correctness of the program.

3.3.4 Menu Item : Run→Stop F10, Hold and Step

Stop F10

Select this menu item〔or press F10〕 to abort program execution immediately. This is the same as pushing the Emergency Stop button to stop all the motion. This button will always exist on another control panel provided by machine maker.

Hold

Select this menu item to enter the system in hold state and hold the motion temporarily. You can press **F9** to leave the hold state and resume program execution.

This menu item acts as a toggle switch to set／clear the single step status.

Step

If the single step status was set, the **SST** status in system state window will be reversed white, and the program execution will run only one line. Each **F9** is pressed and the program pointer will stay at the current line.

If the single step status was cleared, pressing **F9** will run the program without pausing.

3.3.5 Run→Machine Lock, Spindle Axis Lock, Dry Run, Block Skip, Optional Hold

Machine Lock／**Spindle Axis Lock**／**Dry Run**／**Block Skip**／**Optional hold:** these five functions will toggle these related state to turn on or turn off only. When press **F9** to start running CNC program, the system checks these four states automatically and follows states to execute the CNC program:

Machine Lock:

This menu item acts as a toggle switch to lock／unlock the machine.

This action will be effective both during program execution and manual control. If switched to machine lock, the **MLK** status in system state window will be white reversed, and the machine will not move at all.

Spindle Axis Lock：

This menu item acts as a toggle switch to lock/unlock the spindle axis.

This action will be effective both during program execution and manual control. If switched to spindle axis lock, the SLK status in system state window will be white reversed, and the spindle axis will not move at all.

Dry Run〔Ctrl+F3〕:

This menu item〔or press **Ctrl+F3**〕 acts as a toggle switch to enable／disable dry run state.

If the **Dry Run** state is enabled, the **Dry** status in system state window will be highlighted and all feed commands〔`G01, G02, G03, G32`〕 will be executed in dry run speed.
〔Ignore the F code in the program〕

Dry Run speed is set by〝**Machine parameter→Dry run speed**〞. When **Dry Run** speed need to be transformed, please call your machine maker for service.

Block Skip〔Ctrl+F4〕:

This menu item〔or press **Ctrl+F4**〕acts as a toggle switch to enable／disable block skip.

If it is enabled, the **/ SK** status on system state window will be highlighted, and command lines with a leading〝/〞will be ignored.

Optional Hold〔Ctrl+F5〕:

This menu item〔or press **Ctrl+F5**〕acts as a toggle switch to enable／disable **M01**.

If it is enabled, the **M01** status on system state window will be highlighted, and **M01** in the program will act exactly like **M00** to pause the program until press **F9** to resume. If it isn't enabled, **M01** will be ignored. Then program continues to run.

Note: During program running, the state of 〝**Machine lock**〞 can not be changed, but the other four states can be changed at any time.

3.3.6 Menu Item: Run→MDI...

MDI〔Manual Direct Input〕can be used to input and execute one single line of CNC commands exclusively. Select this menu item will open an MDI dialog box for you to enter one line of commands and execute it at once.

You can also use down arrow〔↓〕key to recall the commands that was entered previously.

After making sure that the command line is correct, press **ENTER** to execute, **Esc** to abort **MDI** motion, **F10** to stop the whole system〔including the program that is running〕.

Warning: Press **ENTER** key will execute the command in the input line immediately. Notice the safety on machine limits before pressing the **ENTER** key!
MDI〔Manual Direct Input〕can be used to input and execute single line of CNC commands exclusively. Select this menu item will open a **MDI** dialog box for users to enter single line of commands and execute it at once.

You can also use down arrow 〔↓〕key to recall the commands entered previously.

After making sure that the command line is correct, press **ENTER** to execute, **Esc** to abort **MDI** motion, **F10** to stop the whole system〔including the running program〕.

Warning: Press **ENTER** key will execute the command in the input line immediately. Notice the safety on machine limits before pressing the **ENTER** key!

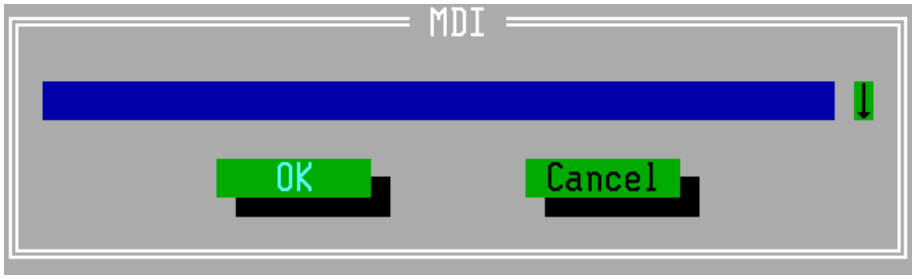


Figure 3-61　A dialog box: MDI

When the program execution has been paused by **M00/M01**, **MDI** can be used to insert single line program execution. When the program has been paused by **M00/M01** during tool radius compensation, this system will not allow **MDI** to input any **Motion G-code** instructions but will still allow to input **Status Change G-code** instructions.

Press the hot key **F7** to pop up a **MDI** dialog box.

If the machine maker opens multi-line MDI commands on PLC setting, executing MDI will not pop signal line mode〔as above figure 3-61〕 but a **MDI Edit Window** on the left top of the main screen, and the program will be executed from the line cursor pointed to end line.

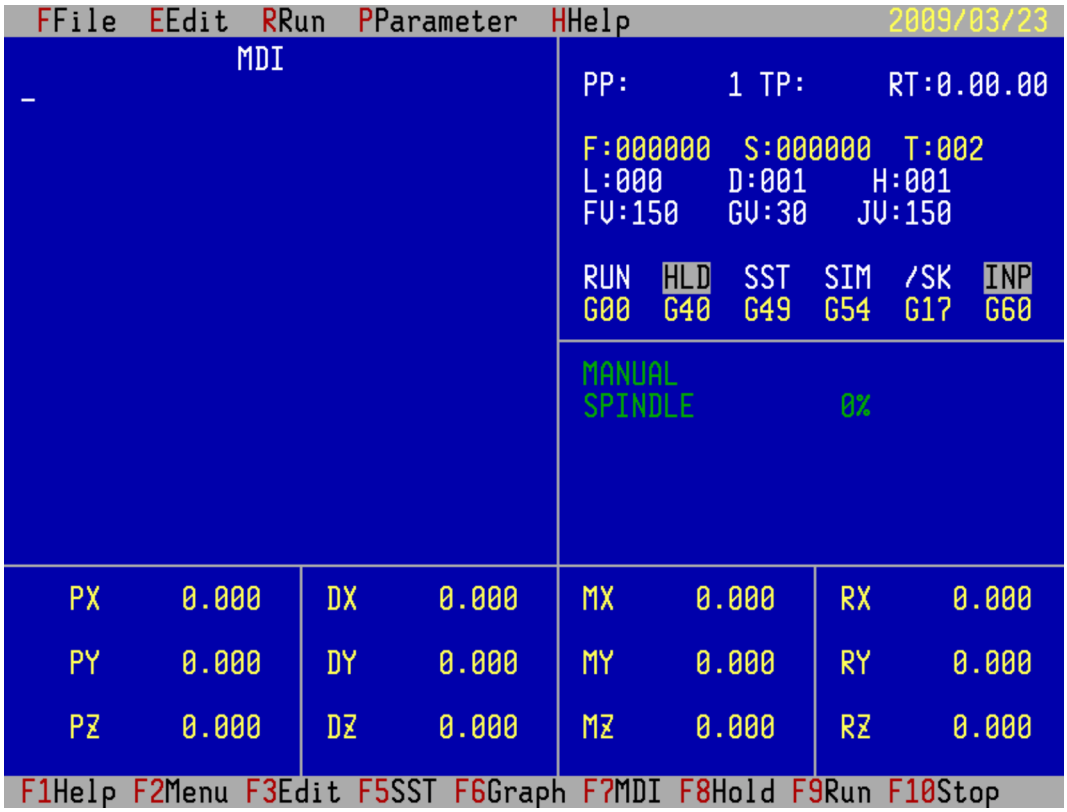


Figure 3-62 The multi-line MDI Edit Window

3.3.7 Menu Item : Run→Direct Run...

Select this menu item to open an **Open Direct Run File** dialog box to select an existed DNC program to run under the direct run mode.

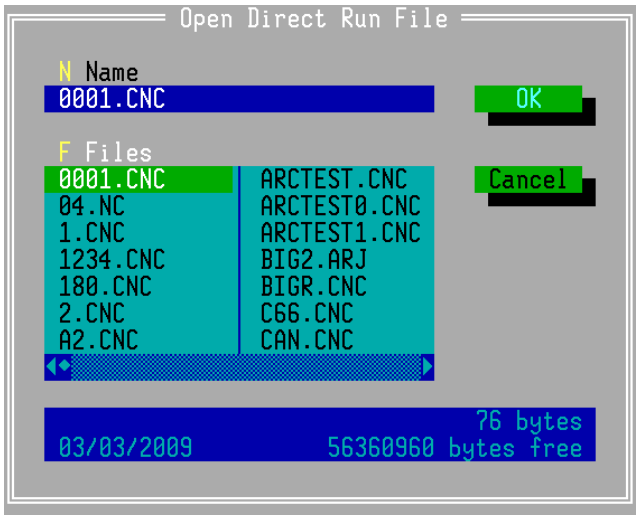


Figure 3-63 an Open Direct Run File dialog box

3.3.8 DNC M30 Nonstop

This menu item〔or press **Ctrl+F2**〕acts as a toggle switch to enable/disable **DNC M30 Nonstop** state. Normally, the **M30** status in system state window is highlighted. In this situation, **M30** will stop running program. If the **DNC M30 Nonstop** state is enabled, the **M30** status in system state window will not be highlighted so that **M30** command from DNC will not stop running program. In such case, users can run series of files by〝**Run→Direct Run→Run File List**〞.

This **Run File List** dialog box is for user to make a series of run file list for machine to run consecutively. To run multiple files, user has to switch the system to **DNC M30 Nonstop** mode.

The following are keys to make the list:

- **Ins** : to select files from a file dialog box for running.
- **Del** : to delete the selected file from the list.

3.3.9 Menu Item : Run→Graphics

Select this menu item to switch to graphic mode.

There are some keys can be used to configure the graphic environment:

Ctrl+Del : Clear the graphic view
Ctrl+Arrow keys : Move the graphics up / down / left / right
Ctrl+I / Ctrl+O : Zoom in / out the X and Y axis of graphics
Ctrl+Home : Center the graphic cursor
ESC : Exit the graphic mode.

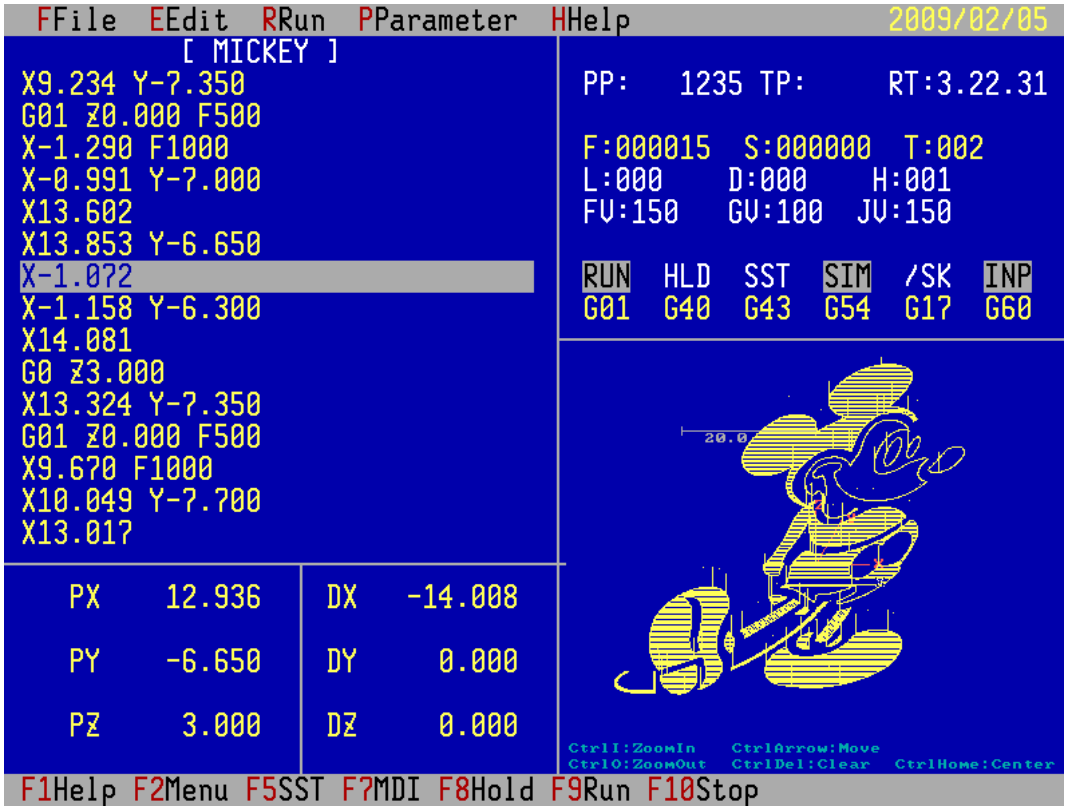


Figure 3-63 The trace window is on the right bottom of the screen.

Also some keys to control the execution of program:

F5 : single step
F8 : hold

F9 : run

F10 : emergence stop

3.4 Parameter〔F2→P〕

Parameter is used for users to confirm or set situations or values, their description is as below:

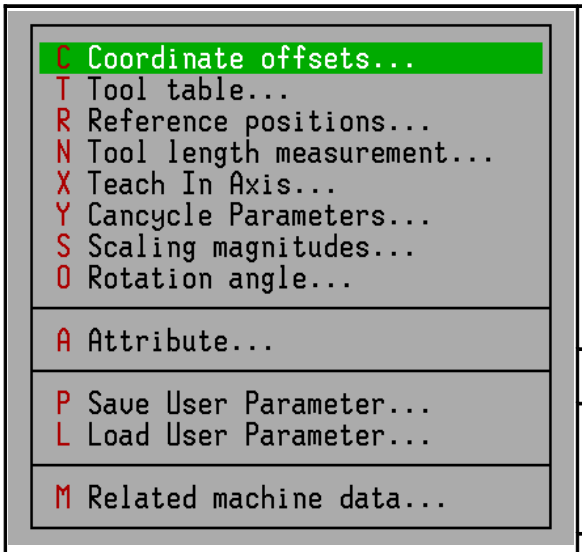


Figure 3-64 The Parameter sub menu

C：The offset setting of six work coordinates T：Tool radius and length offset setting R：4 reference points offset setting N： Tool length measurement setting X：Select a axis for teaching in Y：Can cycle setting S：Default scaling magnitudes setting O：Default rotation angle setting
A：Set the default modal attribute of system
P：Save User Parameter in MECNC document L：Load User Parameter in MECNC document
M：For viewing of related machine data

In dialog boxes under **Parameter** submenu, press **Enter** to save and **Esc** to leave.

3.4.1 Menu Item : Parameter→Coordinate offsets...

This dialog box is used to set the coordinate offsets of **G54~G59**, and also shows the current offsets of **G92** and **G52**.

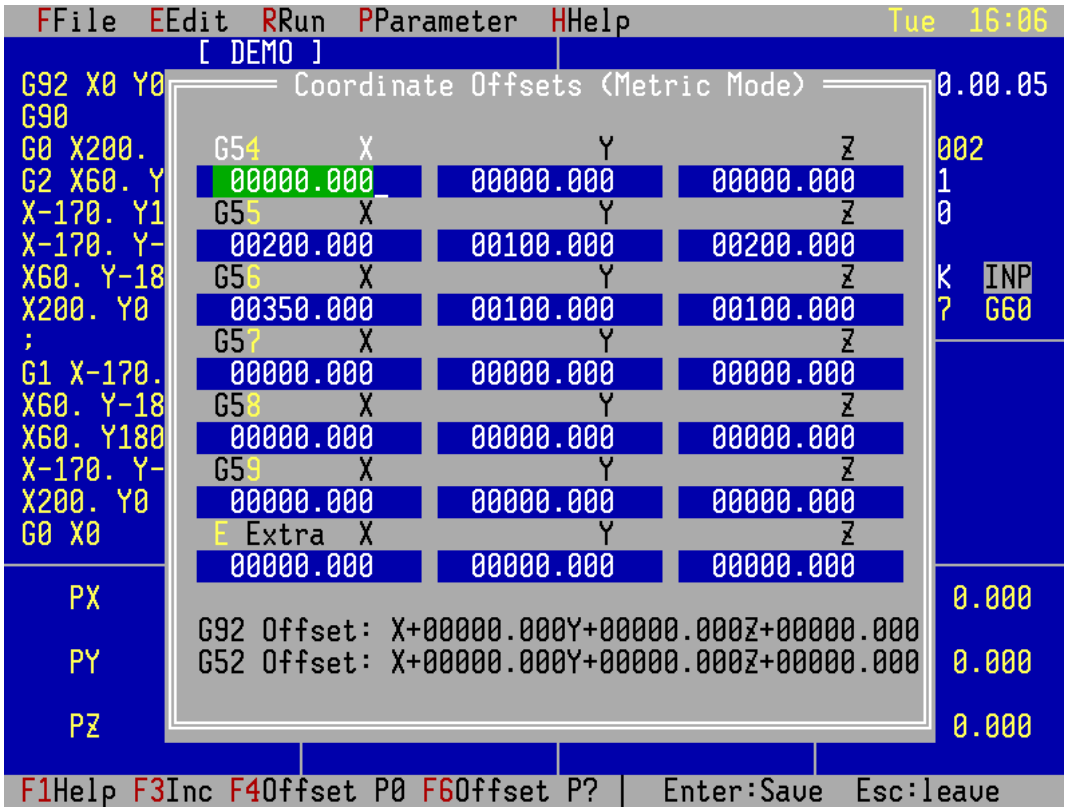


Figure 3-65 A dialog box: Coordinate Offset

There is also a function key **F4** shown in the statusline, when it is pressed, the current system offset of the selected axis of the selected work coordinate system will automatically be calculated and put into the input line. After that, if **Enter** key is pressed, the calculated offset value will replace the original value; else, if **Esc** key is pressed, the original value will not be replaced.

3.4.2 Menu Item : Parameter→Tool Table

This is a table for tool parameter setting. The parameters are tool radius and tool length.

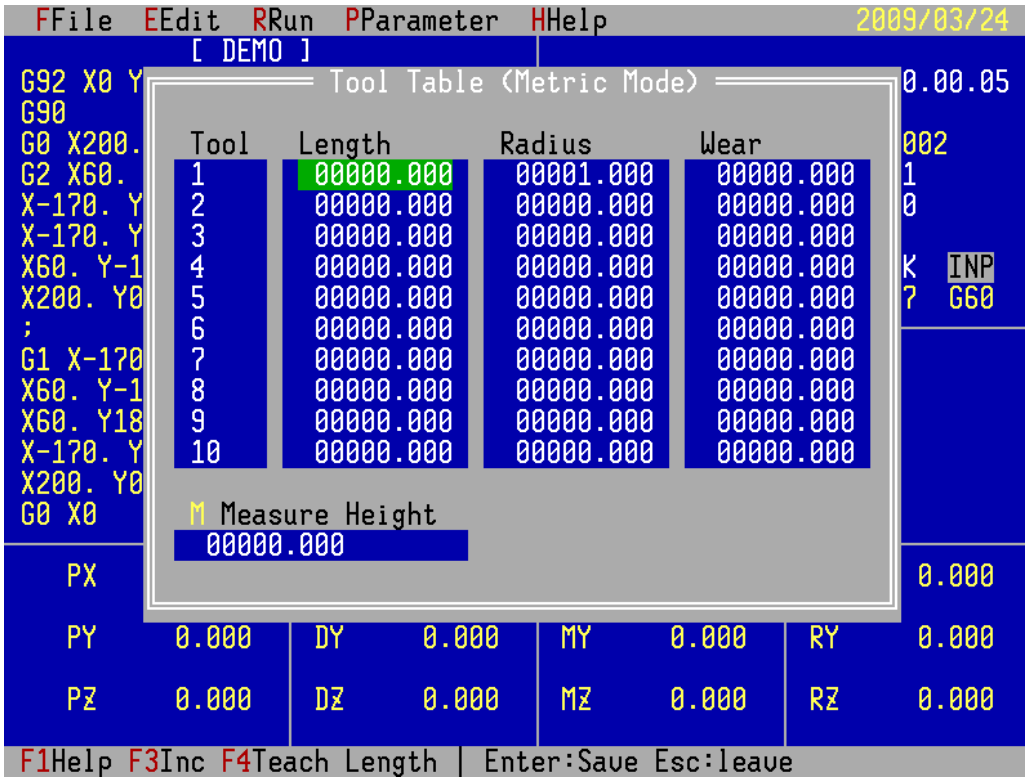


Figure 3-66 A dialog box: Tool table

Use **D_** code to select tool radius compensation data set numbers of this tool table. Use **H_** code to select tool length compensation data set numbers of this tool table.

Tool radius compensation is activated by **G41** or **G42**. Tool length compensation is activated by **G43**.

Note：

1. The tool table provides 128 tool settings.〔Every 10 tools per one page in the dialog box. Use **PgUp** and **PgDn** to switch page.〕
2. The tool length value is a tool adjustment compared to a normal tool, therefore it can be positive or negative value.
3. The tool radius value must be positive only, and do left or right compensation by **G41/G42**.
4. 〝**Wear**〞 means a tiny wear after using a tool for a period. For the more correct cutting size, the wear value must fill in the table.
5. **Length = machine position – Measure Height**

3.4.3 Menu Item : Parameter→Reference Position

Select this menu item to open a **Reference Position** dialog box to set four reference positions relative to the origin of machine coordinate.

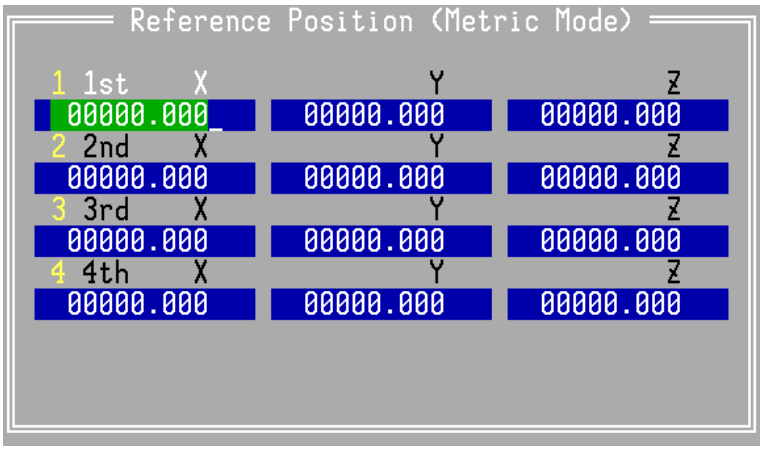


Figure 3-67 A dialog box : Reference Position

3.4.4 Menu Item : Parameter→Tool Length Measurement Parameter...

Select this menu item to open a **Tool Length Measurement Parameter** dialog box to set the parameters for tool length measurement.

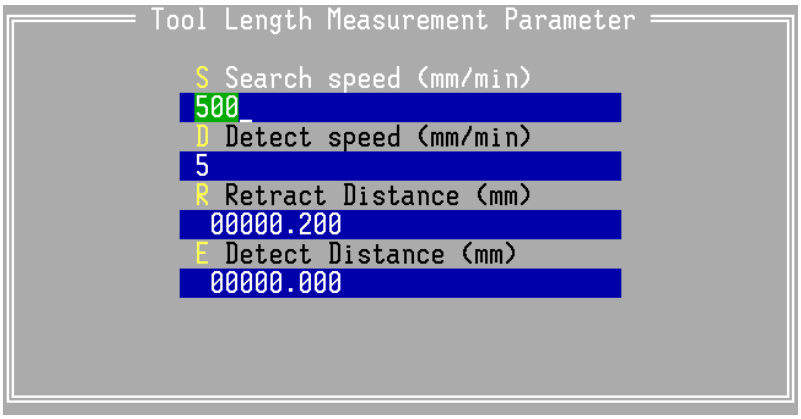


Figure 3-68 A dialog box: Tool Length Measurement Parameter

Search speed : Don't set this value too fast to crash the probe or tool tip.
Detect speed : It is usually a very slow value to get high precision.
Retract distance : Don't set this value too long to waste time.
Detect distance : Don't set this value too long to protect the tool or probe.

3.4.5 Menu Item : Parameter→Set Teach In Axis

This dialog box is for user to set teach in axis.
On editing a program, we can use the manual operation to move the tool to trace a model or an object point by point. And then press **Alt + T** keys to insert the position value of selected axes in this parameter to the program, and generate the program of manual tracing automatically.

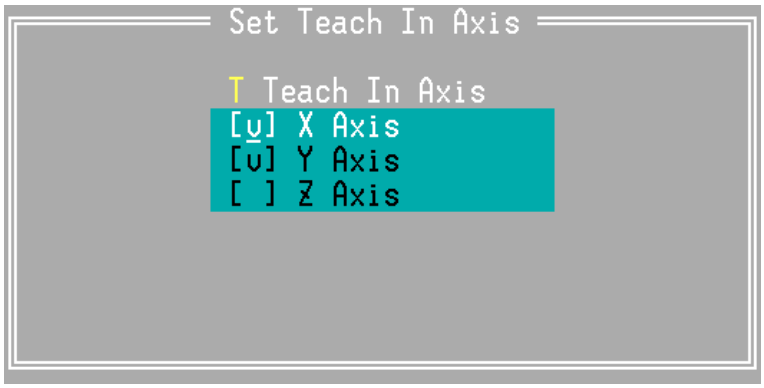


Figure 3-69 A dialog box: Set Teach In Axis

3.4.6 Menu Item : Parameter→Cancycle Parameter

In Cancycles, both **G73/G83** pecking drilling do the action of retraction to dispose the chips. This retract distance〔the **d** value in **G73/G83** figures〕is set by 〝**Parameter→Cancycle Parameter→Cut In Retract Length**〞.〔unit :mm/ inch〕.

In **G70.00/70.01** pocket instruction, do the action of retraction after each layer

machining. The retract distance is also set by 〝**Parameter→Cancycle Parameter→Cut In Retract Length**〞.

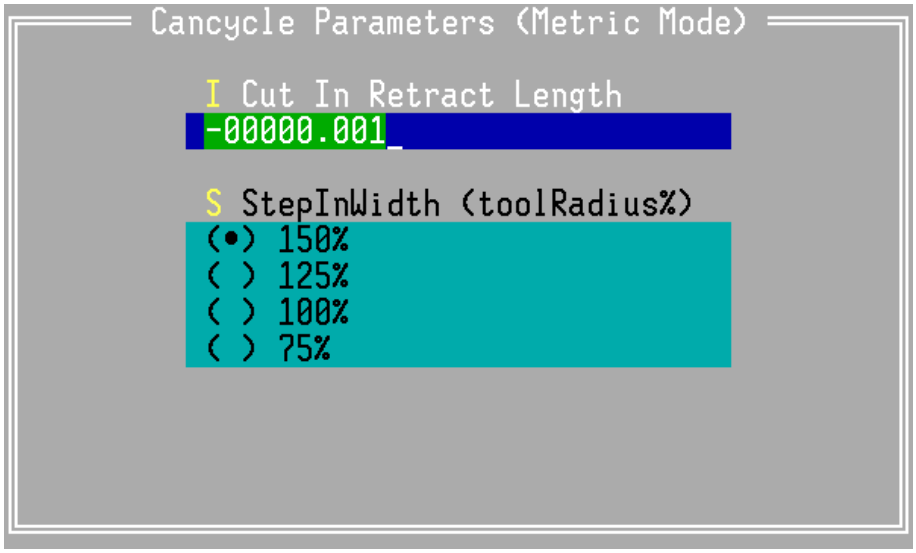


Figure 3-70 A dialog box: Cancycle Parameter

〝**Step In Width**〞:the distance which tool cuts from one cycle to another broadened cycle each layer when executing pocket. The 〝**Step In Width**〞 provides four radius rate settings: 75%, 100%, 125% and 150% .

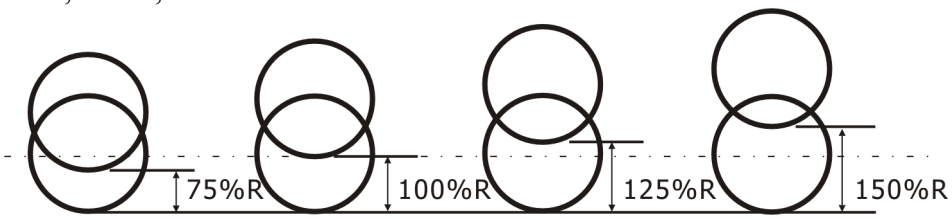


Figure 3-71 There are four radius rate of Step In Width：75%、100%、125% and 150%.

3.4.7 Menu Item : Parameter→Scaling Magnitudes

Select this menu item to open a **Default Scaling Parameter** dialog box to set the default scaling magnitudes of each axis and scaling least increment for the format of magnitudes.

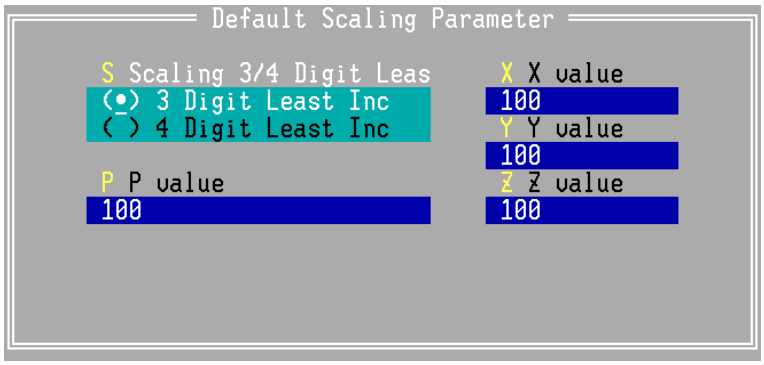


Figure 3-72 a dialog box: Default Scaling Parameter

The formats of **I_,J_,K_** and **P_**of **G51** are different from ones of other instructions. Decimal is not allowed in **G51** instruction. The value inputed here means magnification, for example, **I20000** means the magnification is 20 when select 〝**Scaling ¾ Digit least→(●) 3 Digit Least Inc**〞, but it means the magnification is 2 when select 〝**Scaling ¾ Digit least→(●) 4 Digit Least Inc**〞.

When **I_,J_,K_** and **P_**of **G51** are not specified, 〝**Scaling Magnitudes→P Value**〞 is viewed as the magnification for all axes. When one of **I_,J_** and **K_** is not specified, its magnification will be replaced by **P Value**. The default **P Value** is just positive number.

3.4.8 Menu Item : Parameter→Rotation Angle...

Select this menu item to open a **Default Rotation Parameter** dialog box to set the default rotation angle and absolute or incremental angle mode for rotation.

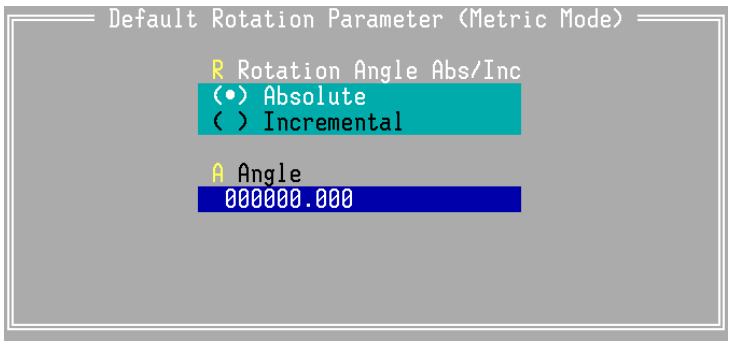


Figure 3-73 A dialog box: Default Rotation Parameter

Rotation angle Incremental/absolute : specify the **R_** code value in **G68** is in incremental or absolute sense.
Angle: when **A_** code is not specified in **G68** instruction, **A_** will be replaced by the **Angle** value of this dialog box.

Note:

The radius programs by absolute or incremental dimension depending on **G90/G91**. The angle does depending on 〝**Parameter→Rotation angle→Rotation Angle Abs / Inc**〞.

3.4.9 Menu Item : Parameter→Attribute

Select this menu item to open a **Default Status** dialog box to set the default modal attribute when system starts up.

The language can be set to English and Chinese.

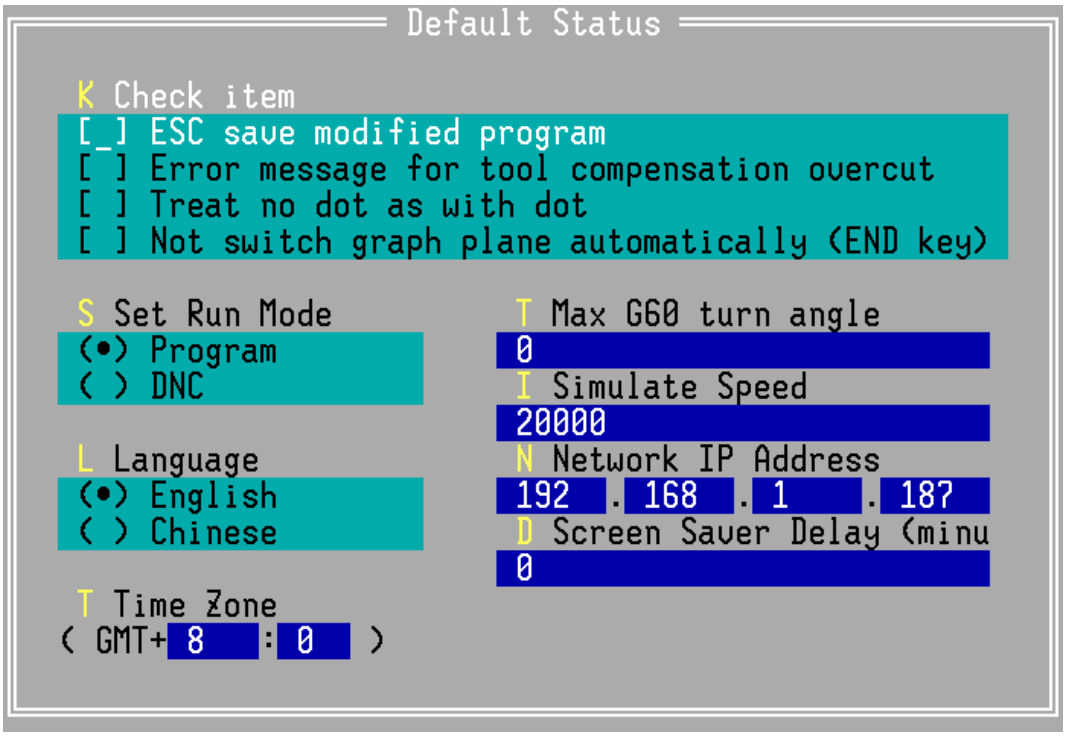


Figure 3-74 A dialog box: Default Status

This dialog box set the default modal attribute of system when system starts up. These default modal attribute includes:

G00/G01,G20/G21,G90/G91 and **G60/G61.**

Check item :

- **ESC save modified program**: If this item is selected, modified program will automatically be saved when Esc is pressed. If not, modified program will pop up a confirm dialog box to ask whether users save the program, press Y to save it and press N not to do.
- **Error message for tool compensation overcu**t: If this item is selected, error message will be signaled when overcut happens in tool radius compensation. Otherwise, a smoother path defined by this application will be taken.
- **Treat no dot as with dot** : If this item is selected, a command value without decimal point will be treated as one with decimal point. It is usually specified a real number in a normal program but users input a integer number ,which is transformed integral multiple of the system least unit. For example: the inputed value, **X1234**, is transformed as **X1.234**. When this item is selected, the inputed value, **X1234**, is transformed as **X1234.**,which is the same as **X1234.000**.
- **Not switch graph plane automatically (END key)** :If this item is selected, the display plane of trace window will not switch because of change of **G17/G18/G19** automatically. It can be switched by **End** key.

Set Run Mode :

The item is used to choose run mode. The Run Mode means system executes a program from controller itself or DNC.

Language :

The language mode of the system can be set to English or Chinese mode. It works after reset.

Time Zone :

To set time of controller upon where machine is. 〝**GMT**〞 means the standard time in Greenwich. For example, if it is in Taiwan, key in 〝**+8:0**〞; if it is in Kathmandu, key in 〝**+5:45**〞 ; if it is in the east of America, key in 〝**-5:0**〞...and so on.

Max G60 turn angle :

It means the max included angle of consecutive block paths.

When included angle of consecutive block paths of the program is larger than the **Max G60 turn angle**, in-position check is always performed whether in **G60** mode. This motion makes work piece of larger degrees no distortion and get exact acute angle, as the following figure (A). When the **Max G60 turn angle** is set to zero, in-position check isn't performed even in **G61** mode.

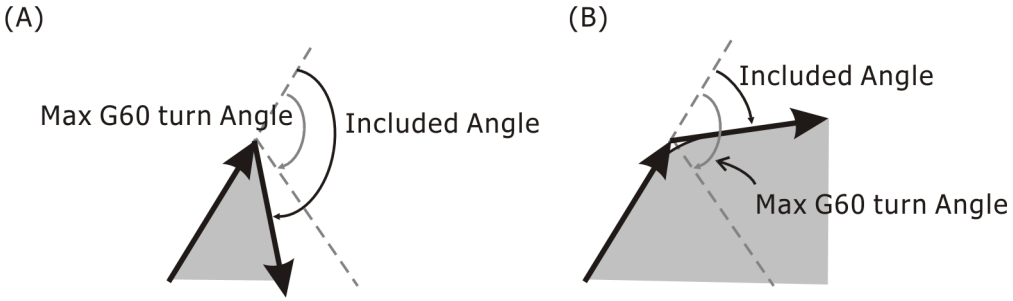


Figure 3-75 (A) Included angle>the Max G60 turn angle, in-position check. (B) Included angle<the Max G60 turn angle, no in-position check.

Simulate Speed :

Set the simulate speed of system. It can be up to 65000 mm / min.

Network IP Address :

Transmitting files through Internet is the specified function of INCON-ME3. A hard disk and network card has been built in INCON-ME3. There is a connect RJ45 on outside of controller to be used to connect with a HUB. ME3 controller is a FTP server upon connecting Internet, and other computers connected with ME3 are clients.

Most of companies network are planned as class C. It means the first three digits are network name, and the last digit is the computer name. The file servers〔ME3 controller〕must be the same network name with other computers in a company, so that the first three digits in one company must be the same. The last digit is given by MIS of the company〔1 to 254〕to avoid unconvertible Internet by IP address conflicting .

Please ask your MIS about setting of other network class levels.

Screen Saver Delay (minute) :

This item is used to set the delay time of screen saver. When over the setting delay time, the screen will be changed into a screen saver pattern with a float string,INCON-ME3. The screen restore the original pattern until press any key.

Note:The setting value, 0, means screen saver isn't applied.

3.4.10 Menu Item : Parameter→Save／Load User Parameter

Select this menu item to open a **Save User Parameter** dialog box to define the file name under which the user parameters is going to be saved in the MECNC document. When you need it one day, you can select this menu item to open a **Load User Parameter** dialog box to select a user parameter file from the MECNC document to load into the system.

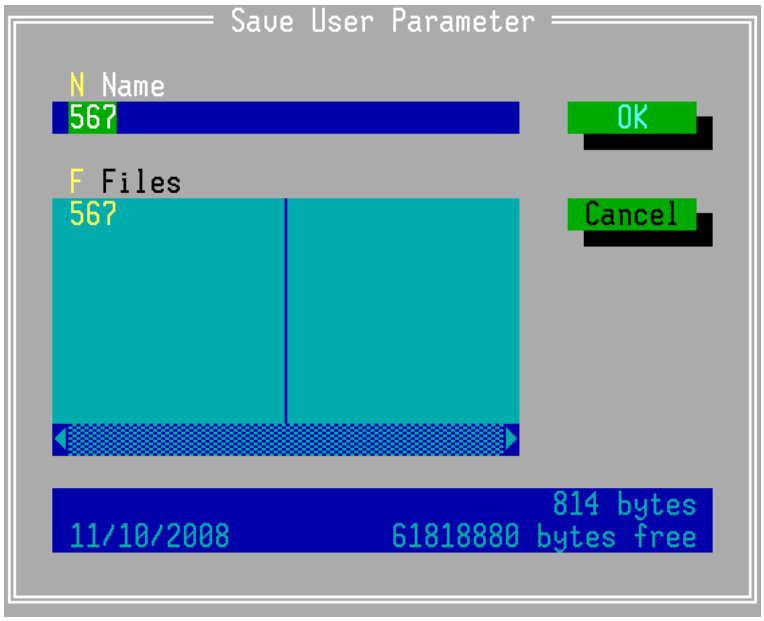


Figure 3-76 A dialog box : Save User Parameter

3.4.11 Menu Item : Parameter→Related Machine Data

Select this item to open a **Related Machine Data** dialog box to view information of some related machine parameters. Users can't modify it.

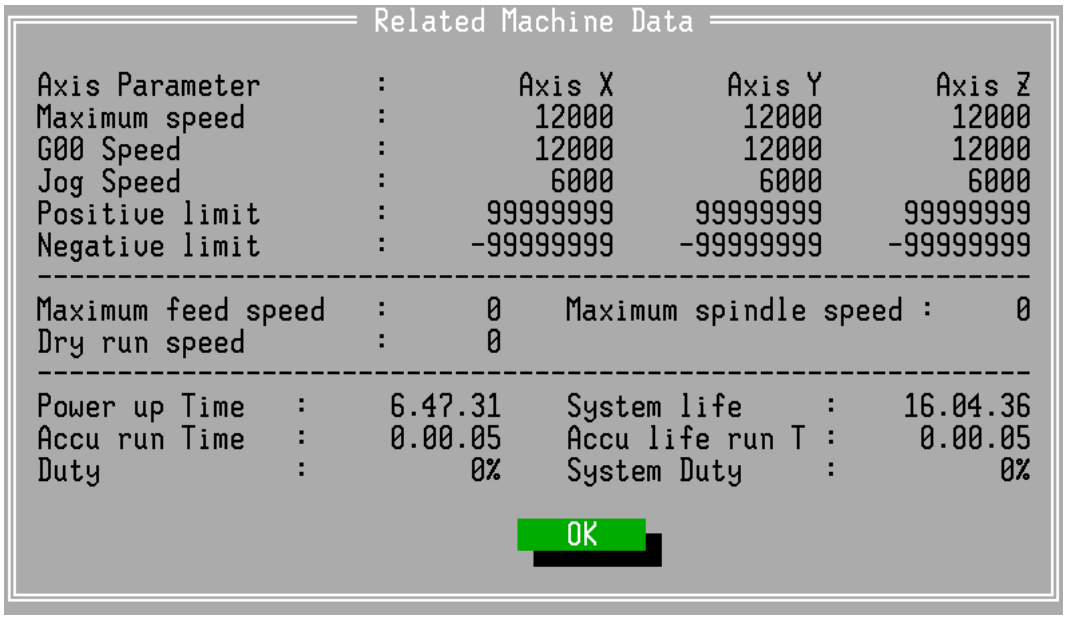


Figure 3-77 A dialog box : Related Machine Data

- **Maximum Speed :** Maximum speed for each axis motion.〔mm / min〕
- **G00 speed :** The normal 100% **G00** speed.〔mm / min〕
- **G00 0 override speed :** The speed of **G00** 0% override is usually not the zero speed. It is usually a slow speed of about 500~1000 mm / min, depends on machine.
- **Jog speed :** The normal 100% Jog speed.〔mm / min〕
- **Positive / Negative limit :** Machine traverse limit for each axis, in minimum unit of the system.〔such as μm〕
- **Maximum feed speed :** Maximum feed speed for each axis. If the given **F_** value applying feed rate overrate over this value, then the real speed will be clamped to this value.〔mm / min〕
- **Dry run speed** : The speed for dry run 〔mm / min〕. It is set by 〝**Machine Parameter→Motion and Speed→Dry Run Speed**〞.
- **Maximum spindle speed :** The maximum speed of spindle rotation.〔rpm〕

Three lines data in the lower of this **Related Machine Data** dialog box shows the used time of this machine. The left half shows the time status from power up of current time, the right half shows the time status accumulated from the time of machine delivery.

- **Power up Time :** Accumulated power-on time of current time from power up.
- **Accumulated run Time:**Accumulated machining time of current time from power up.
- **Duty :** (Accumulated run Time) / (Power up Time)×100%
- **System life :**Accumulating time of power on from machine delivery.

- **Accumulated life run Time:** Accumulated machining time from machine delivery.
- **System Duty :** (Accumulated life run Time) / (System Life)×100%

3.5 Help〔F2→H〕

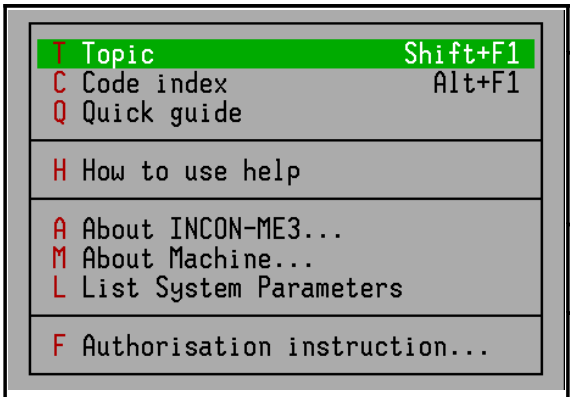


Figure 3-79 the **Help** submenu

Description
T：to show a list of help topics of the system C：to show the code listing of the system Q：to describe a quick guide to edit and run a demonstration motion program
H：to describe how to use INCON-TE2 context sensitive help system
A：to show the copyright information and version number of INCON-ME3 M：to show information of this machine L：list and print system parameters
F：to enter key code to unlock feed lock

At any time, press **F1** to open context sensitive help window, **Esc** to leave.

The following are key strokes for using help :

Up / down / left / right arrow keys: move around the help text
PgUp / PgDn: scroll up / down one text page
Tab key: switch one cross reference topic forward
Shift+Tab: switch one cross reference topic backward
Enter: enter selected cross reference topic
Esc: close help window.

3.5.1 Menu Item : Help→Topic Shift+F1

Select this menu item to open a help window which shows a list of help topics of the system.

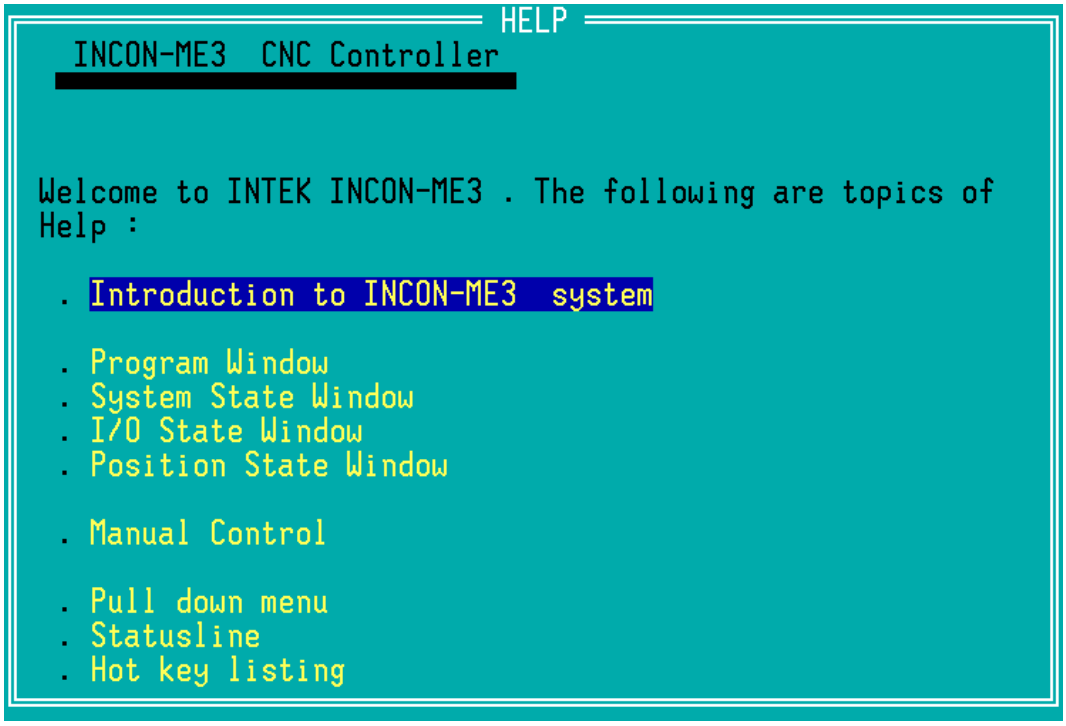


Figure 3-80 a help window which shows a list of help topics of the system

3.5.2 Menu Item : Help→Code Index Alt+F1

Select this menu item to open a help window which shows the code listing of the system.

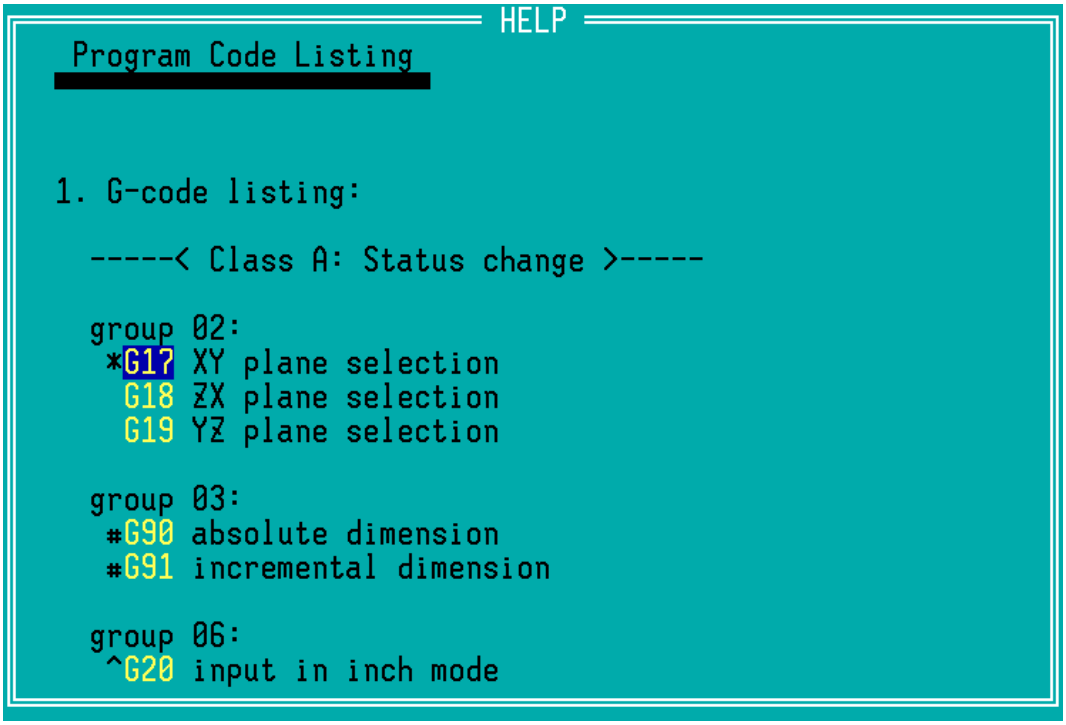


Figure 3-81 a help window which shows the code listing of the system

3.5.3 Menu Item : Help→Quick Guide

Select this menu item to open a help window which describes a quick guide to edit and run a demonstration motion program.

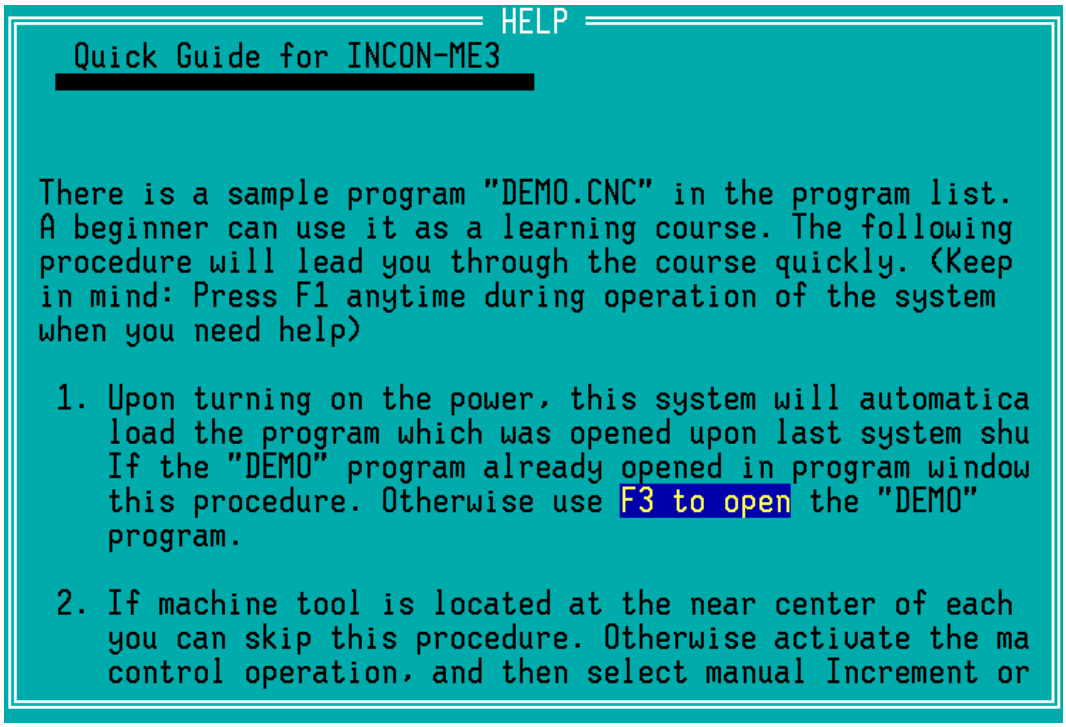


Figure 3-82 a help window which describes a quick guide to edit and run a demonstration motion program

3.5.4 Menu Item : Help→How to use help

Select this menu item to open a help window which describes how to use INCON-TE2 context sensitive help system.

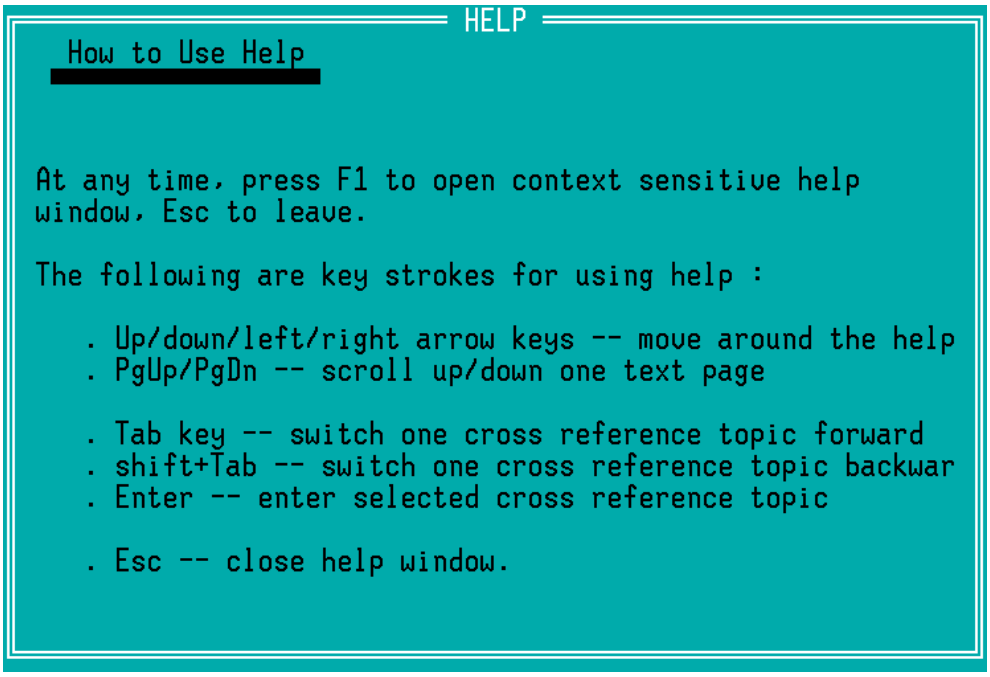


Figure 3-83 a help window which describes how to use INCON-TE2 context sensitive help system

3.5.5 Menu Item : Help→About INCON-ME3...

Select this menu item to open a greeting window which shows the copyright information and version number of INCON-ME3.

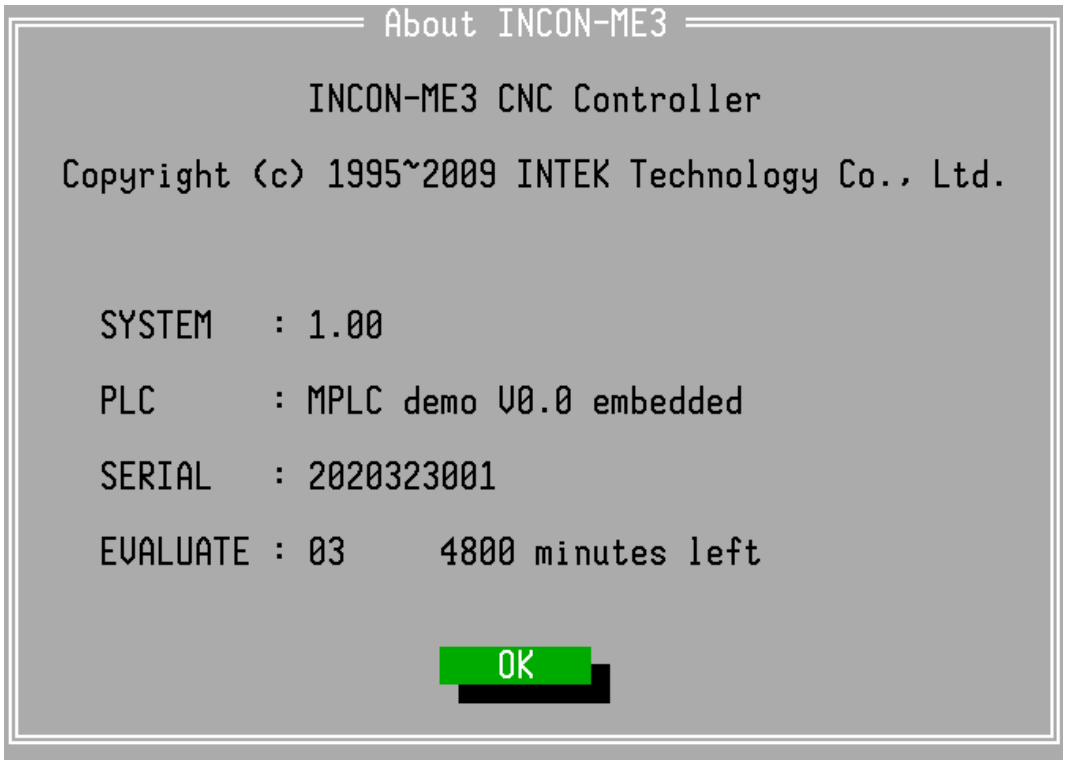


Figure 3-84 A greeting window

Every INCON-ME3 has its own SERIAL number built in ME3 controller. You can look at SERIAL when you select 〝**Help→About INCON-ME3**〞.
For example:

2	2	0323	001
A	B	C	D

A: controller, the digit 2 means the controller is ME3.
B: year, the digits 02 means the controller is made in 2009.
C: date, the digits 0323 means the controller is made on March 23.
D: serial number

3.5.6 Menu Item : Help→About Machine...

Select this menu item to open a greeting box which shows information of this machine.

Figure 3-85 a greeting box which shows information of this machine

3.5.7 List System Parameter

User parameter and machine parameter are saved by a special format. This parameter can not be read on normal computers directly. Select this menu item to translate user parameter and machine parameter into plain text format which can be read on normal computers, and therefore they can be printed or saved on normal computers.

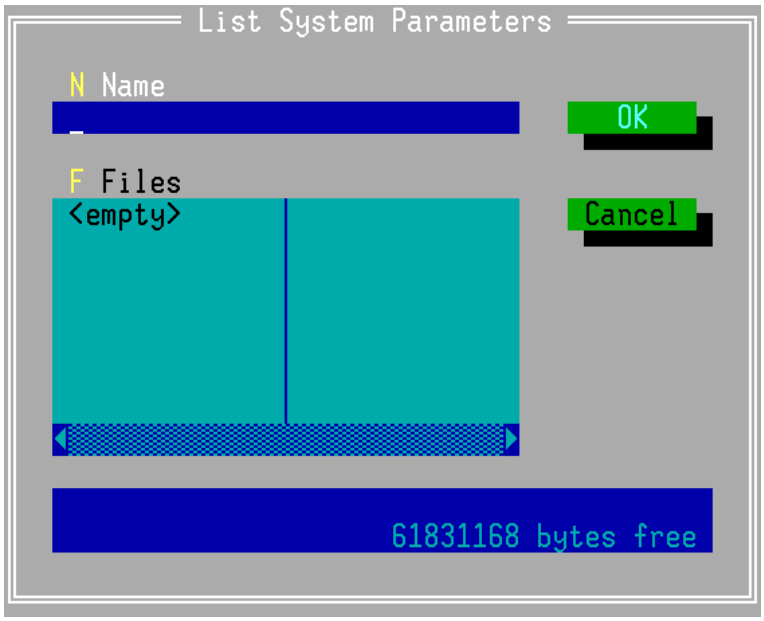


Figure 3-86 A dialog box : List System Parameters

Selecting this menu item will output〝**About INCON-ME3**〞, machine parameter and user parameter into a plain text format file, its extension name is PLS and Chinese words are encoded by ET code. We provide the encoder program to translate ET code into BIG5 code, and the program can download from Http://support.intek.com.tw .

3.5.8 Menu Item: Help→Authorisation instruction

Besides basic functions, INCON-ME3 controller provides six optional functions. The optional function shows in the 〝**About INCON-ME3**〞 greeting window. For example, 〝**FACILITY : 01**〞 means user can save, open and run programs by USB device ; 〝**FACILITY : 02**〞 means user can run macro instructions. Other optional functions shows in chapter 5.

When user needs optional function and finishes business procedure, INTEK will provide a authorisation instruction according to Serial and optional function. User inputs the authorisation codes into this dialog box 〝**Authorisation Instruction**〞 within three days from codes producing. And user will execute the optional function.

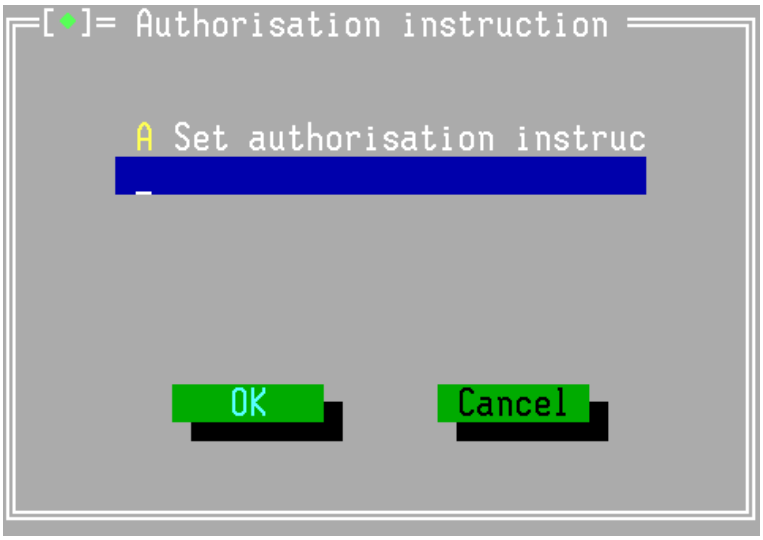


Figure 3-87 the dialog box : Authorisation Instruction

User can get one optional function trial upon inputing trial authorisation codes by the same way. The difference of the trail is period limit in minute. The remainder trial time shows in the dialog box 〝**Authorisation Instruction**〞.

Chapter 4 Codes

4.1 Specification and definition

4.1.1 Specification

I、 Programming

- 3 axes positioning〔The 4th axis option〕
- 3 axes linear interpolation 〔The 4th axis option〕
- 2 axes angle chamfering and corner rounding
- 2 axes circular/3-axes helical cutting
- Absolute/incremental dimension
- Cartesian/polar system
- Metric/inch system
- Scaling, mirror and rotation
- Tool radius/length compensation
- Automatic tool length measurement
- Canned cycles〔drilling, boring and taping〕
- Rigid tapping〔option〕
- Thread、constant surface speed and feed per revolution
- 6 work coordinates with child coordinate
- Inner program nest execution/ call inner subroutine / call external subprogram
- Editor can edit capacity of parts program up to 20 M bytes
- Circular / rectangular pocket and dowel pocket
- Circular / rectangular inner/outer side surfacing

- Surfacing
- Line/ matrix / arc pattern drilling
- Object machining in line / matrix / arc pattern arrangement

II、 Operation

- Single block execution /optional block skip
- Dry run/ Machine lock/ Spindle axis lock
- MDI〔manual data input execution〕
- Manual increment / hand wheel / jogging / home operation, auto setting the program position
- Feed rate override for `G00` positioning speed、cutting speed and Jog speed
- CRT graphic mode to trace tool path
- State Window updates at once
- Enlarged position counter display
- Windows based pull down menu
- Context sensitive help
- Default English version, can load other language file to replace it
- Command Insertion Aid can use table to fill in commands.
- Use dialog box input instructions and shows figures relative its instruction
- Manual aid teach-in program position for auto program generating
- Manual feed program running in forward or backward cutting
- Multi-hand wheel installable to operation the machine in traditional way
- The 4th axis option
- Network connecting, can be a FTP server for up/down load CNC programs

III、 Capacity

- Max 20 M-byte working RAM for each CNC program
- Battery memory backup RAM for all programs
- Execute or edit programs by USB device, and transmit files from CAD work station by Ethernet
- ME3 is a FTP server

4.1.2 Block definition

The program of INCON- ME3 is similar to most of CNC controllers. The normal program consists of one of four kinds of function codes〔**G, M, S, T**〕and address with value following with function codes.

A **block** of CNC program is the same meaning as a line of CNC program. It consists of the items of the table below. Each item includes an alphabet and some figures, which are called the address, number, code or value.

Nxxxx	**Gxx**	**Xxxx.x**	**Yxxx.x**	**Zxxx.x**	**Ixxx.x**	**Jxxx.x**	**Kxxx.x**	**Pxxxx**	**Lxxxx**	**Rxxx.x**
Sequence Number	preparatory function	address & value	address & value	address & value	address & value	address & value	address & value	address & value	address & value	address & value

Fxxx.x	**Mxx**	**Sxxxx**	**Txx**	**Enter**
Feedrate setting	Miscellaneous function	Spindle Speed	tool table select	end of block

The combination of the alphabet and figures will explain in the next section G-code & M-code format later.

The values of the address could be an integer or a real number. If the value must be a real number but user just input an integer, the inputed value will be multiplied by least command increment〔least command increment means 0.001 mm in metric system，0.0001 inch in imperial system〕and input the transformed value into the controller.

For example, in metric system, 〝**X12000**〞 given by user, then the value will be transformed as 〝**12**〞 mm〔12000×0.001 mm＝12 mm〕by controller. When user desire to input 12 mm, must give 〝**X12.**〞 or 〝**X12000**〞. Most users usually use 〝**X12.**〞 because they consider this more convenience and save more capacity of disk. In other words, when users desire 12 mm but input 〝**X12**〞, the actual value transformed by controller is 0.012 mm. The difference multiple between these two values is 1000, so that crash or cutting over occurs then. Must keep in mind.

ME3 controller provides the other input mode, 〝**Parameter→Attribute→Check item→[✔]Treat no dot as with dot**〞, for users who are not used to do by the above-mentioned method. 〝**[✔]Treat no dot as with dot**〞 means whether inputed value is 〝**X1234**〞 or 〝**X1234.**〞, it will be transformed as 〝**X1234.000**〞 by controller.

4.1.3 Program Definition

The CNC program name could be up to eight characters. The eight characters are composed of some of 26 alphabets〔**A~Z**〕and 10 numerals〔**0~9**〕. When the program is saved, it will follow the extension name with 〝**.CNC**〞, like 〝**SAMPLE.CNC**〞.

SAMPLE.CNC	Description
;Sample program	〝;〞: Comment
' for manual	〝'〞: Comment
N1 Gxx Xxxx.x Yxxx.x ;	1st block
/ Gxx Xxxx.x Yxxx.x Fxxx.x	2nd block, /: Block Skip
M03 S3600;Turn on spindle in 3600rpm	3rd block, with comment
N10 ... ;	4th block
...	.....
...	.....
M30	End of program

- 〝;〞and 〝'〞: Comment
- /: Block Skip
- 〝**N1**〞, 〝**N10**〞 indicate the line numbers, and it is not necessary to be specified in every line.

Next section lists the summary of G code and M code, and will explain more detail .

4.1.4 Axis and Coordinate Definition

We usually use X, Y, Z to indicate the three normal axes in Cartesian coordinate system. The 4th axis〔option〕is a rotation axis added from Cartesian coordinate system , the figure is as below.

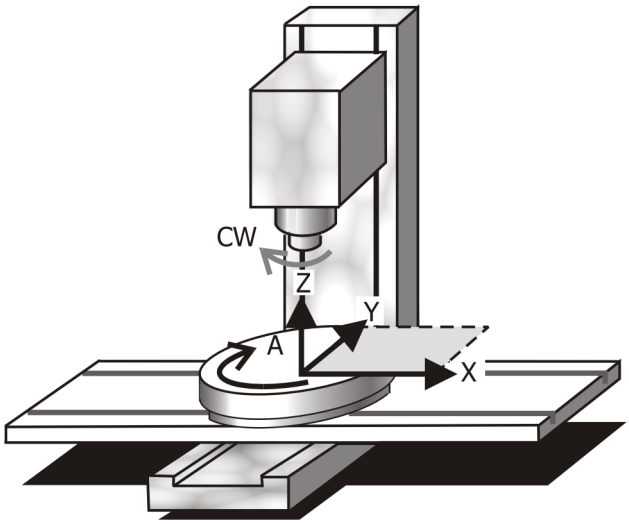


Figure 4-1 The coordinate of X, Y, Z and the 4th axis of program

The rotation axis could select one of A,B,C,U,V and W to denote its axis name.〔set by〝**parameter→the 4th axis**〞〕, the 4th axis named by **A_** temporarily in the manual. Its value usually means rotary degrees.

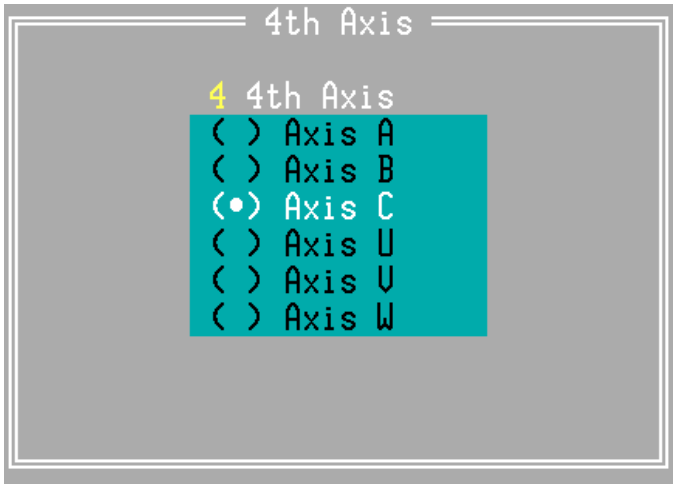


Figure 4-2 A dialog box: the 4th axis named〔option〕

4.1.5 List of G-code

Group	G code		Format	Function
01	#	G00	G00 X_Y_Z_A_;	Positioning〔rapid traverse〕
	#	G01	G01 X_Y_Z_A_;	Linear interpolation
			G01 X_Y_R_;	G01 for corner rounding
			G01 X_Y_L_;	G01 for chamfering
		G02	Radius, Center and Angle	G02 CW〔Clockwise〕circular interpolation
		G03		G03 CCW〔Counter-clockwise〕circular interpolation
			G17 G02(G03) X_Y_R_F_; G18 G02(G03) X_Z_R_F_; G19 G02(G03) Y_Z_R_F_;	Radius Method
			G17 G02(G03) X_Y_I_J_F_; G18 G02(G03) X_Z_I_K_F_; G19 G02(G03) Y_Z_J_K_F_;	Center Method
			G17 G02(G03) I_J_A_F_; G18 G02(G03) I_K_A_F_; G19 G02(G03) J_K_A_F_;	Angle Method
00		G04	G04 P_; or G04 X_;	Dwell time
		G09	G09	One Shot Exact Stop
00		G10	G10(P_R_L_);	Data setting mode on
		G11	G11;	Data setting mode off
17	*	G15	G15;	Cartesian Coordinate〔cancel G16〕
		G16	G16;	Polar Coordinate
02	*	G17	G17;	XY plane selection
		G18	G18;	ZX plane selection

Group	G code		Format	Function
		G19	G19;	YZ plane selection
06	^	G20	G20;	Input in imperial mode
	^	G21	G21;	Input in metric mode
00		G28	G28 X_Y_Z_A_;	Return to Primary Reference Position by middle point
		G29	G29 X_Y_Z_A_;	Return from Target Position by middle point
		G30	G30 X_Y_Z_A_P_;	Return to 2nd, 3rd and 4th Reference Position by middle point
00		G31	G31 X_Y_Z_A_F_;	Optional Skip
01		G33	G33 X_Y_Z_A_F_;	Thread Cutting
00		G37	G37 Z_Q_D_F_E;	Automatic tool length measurement
07	*	G40	G40;	Tool Radius Compensation off〔Cancel G41/G42〕
		G41	G41;	Tool Left Radius Compensation
		G42	G42;	Tool Right Radius Compensation
08		G43	G43;	Tool Length Compensation on
	*	G49	G49;	Tool Length Compensation off〔Cancel G43〕
11	&	G50	G50 X_Z_;or G50 S_;	Scaling/Mirror off〔Cancel G51〕
		G51	G51 X_Y_Z_I_J_K_ or G51 X_Y_Z_P_	Scaling/Mirror on
00	&	G52	G52 X_Y_Z_A_;	Set Local〔Child〕Coordinate Offset
00		G53	G53 X_Y_Z_A_;	Positioning in Machine Coordinate
14	*	G54	G54;	Work Coordinate System 1 Selection
		G55	G55;	Work Coordinate System 2 Selection
		G56	G56;	Work Coordinate System 3 Selection
		G57	G57;	Work Coordinate System 4 Selection

Group	G code		Format	Function
		G58	G58;	Work Coordinate System 5 Selection
		G59	G59;	Work Coordinate System 6 Selection
15	#	G60	G60;	Exact Stop Mode〔In Position Check Off〕〔Cancel G60〕
	#	G61	G61;	Exact Stop Mode On〔In Position Check On〕
16		G68	G68 X_Y_Z_R;	Rotation on
	*	G69	G69;	Rotation off〔Cancel G68〕
00		G70.00	G70.00 (G02/G03) P_X_Y_Z_U_V_W_C_J_K_Q_(R_/L_)D_I_E_ ;	Pocket, Rectangular
		G70.01	G70.01 (G02/G03) P_X_Y_Z_R_(U_V_W_)C_J_K_Q_I_E_F_;	Pocket, Circular
		G70.02	G70.02 (G02/G03) P_X_Y_Z_U_V_W_A_B_C_J_K_Q_(R_/L_)D_;	Dowel, Rectangular
		G70.03	G70.03 (G02/G03) P_X_Y_Z_R_(U_V_W_)C_J_K_Q_I_E_F_;	Dowel, Circular
		G70.04	G70.04 (G02/G03) I_P_X_Y_Z_U_V_W_(R_/L_)D_F_;	Side Surfacing, Rectangular
		G70.05	G70.05 (G02/G03) I_P_X_Y_Z_R_(U_V_W_)F_;	Side Surfacing, Circular
		G70.06	G70.06 D_X_Y_Z_U_V_W_C_I_F_;	Surfacing
		G70.07	G70.07 G_X_Y_Z_R_I_J_L_Q_P_F_;	Drill Pattern, Line
		G70.08	G70.08 C_G_X_Y_Z_R_I_J_A_B_U_V_W_D_Q_P_F_;	Drill Pattern, Grid
		G70.09	G70.09 G_X_Y_Z_R_C_A_I_L_B_Q_P_F_;	Drill Pattern, Arc
		G70.10	G70.10 (G41/G42) P_X_Y_Z_(U_V_W_)C_J_K_Q_R_I_E_F_;	Contour Pocket

Group	G code		Format	Function
		G70.11	G70.11 (G41/G42) P_X_Y_Z_(U_V_W_)C_R_I_F_;	Contour Pocket Side Surfacing
		G70.21	G70.21 I_J_L_;	Object Pattern, Line
		G70.22	G70.22 C_I_J_A_B_D_;	Object Pattern, Grid
		G70.23	G70.23 X_Y_Z_R_A_I_L_;	Object Pattern, Arc
09		G73	G73 X_Y_Z_R_Q_K_F_;	High speed peck drilling cycle
		G74	G74 X_Y_Z_R_P_K_F_;	Left handed tapping cycle〔rigid tapping〕
		G76	G76 X_Y_Z_R_Q_P_K_F_;	Fine boring cycle
		G80	G80;	Turns off cancycle
		G81	G81 X_Y_Z_R_K_F_;	Drilling cycle, spot boring cycle
		G82	G82 X_Y_Z_R_P_K_F_;	Drilling cycle, counter boring cycle
		G83	G83 X_Y_Z_R_Q_K_F_;	Peck drilling cycle
		G84	G84 X_Y_Z_R_P_K_F_;	Right handed tapping cycle〔rigid tapping〕
		G85	G85 X_Y_Z_R_K_F_;	Boring cycle without spindle stop on the bottom
		G86	G86 X_Y_Z_R_K_F_;	Boring cycle with spindle stop on the bottom
		G87	G87 X_Y_Z_R_Q_P_K_F_;	Back boring cycle
		G88	G88 X_Y_Z_R_P_K_F_;	Boring cycle which allows manual motion from the bottom
		G89	G89 X_Y_Z_R_P_K_F_;	Boring cycle with dwelling on the bottom
03	#	G90	G90;	Absolute dimension
	#	G91	G91;	Incremental dimension
00	&	G92	G92 X_Y_Z_A_; G92;	Set absolute zero point **G92** External Offset
05		G94	G94;	Feed per Minute
		G95	G95;	Feed per Revolution

Group	G code		Format	Function
13		G96	G96;	Constant Surface Speed Control on
	*	G97	G97;	Constant Surface Speed Control off〔Cancel G96〕
05		G98	G98;	Return to initial position in cancycle
	*	G99	G99;	Return to reference position in cancycle

4.1.6 In Groups

Class A: Status change

	03	
#	G90	Absolute dimension
#	G91	Incremental dimension

	06	
^	G20	Input in imperial mode
^	G21	Input in metric mode

	02	
*	G17	XY plane selection
	G18	ZX plane selection
	G19	YZ plane selection

	07	
*	G40	Tool Radius Compensation off
	G41	Tool Left Radius Compensation
	G42	Tool Right Radius Compensation

	17	
*	G15	Cartesian Coordinate〔cancel G16〕
	G16	Polar Coordinate

	08	
	G43	Tool Length Compensation on
*	G49	Tool Length Compensation off〔Cancel G43〕

	11	
*	G50	Scaling/Mirror off〔Cancel G51〕
	G51	Scaling/Mirror on

	16	
	G68	Rotation on
	G69	Rotation off〔Cancel G68〕

	05	
*	G94	Feed per Minute
	G95	Feed per Revolution

	10	
*	G98	Return to initial position in cancycle
	G99	Return to reference position in cancycle

	14	
*	G54	Work Coordinate System 1 Selection
	G55	Work Coordinate System 2 Selection
	G56	Work Coordinate System 3 Selection
	G57	Work Coordinate System 4 Selection
	G58	Work Coordinate System 5 Selection
	G59	Work Coordinate System 6 Selection

	13	
	G96	Constant Surface Speed Control on
	G97	Constant Surface Speed Control off〔Cancel G96〕

	15	
	G60	Exact Stop Mode〔In Position Check Off〕〔Cancel G60〕
	G61	Exact Stop Mode On〔In Position Check On〕

Class B: Motion

	01	
#	G00	Positioning〔rapid traverse〕
#	G01	Linear interpolation
	G02	G02 CW circular interpolation
	G03	G03 CCW circular interpolation
	G33	Thread Cutting
	G31	Optional Skip

00	
G10	Data setting mode on
G11	Data setting mode off
G28	Return to Primary Reference Position by middle point
G29	Return from Target Position by middle point
G30	Return to 2nd, 3rd and 4th Reference Position by middle point
G53	Positioning in machine coordinate
G37	Automatic tool length measurement
G70.00	Pocket, Rectangular
G70.01	Pocket, Circular
G70.02	Dowel, Rectangular
G70.03	Dowel, Circular
G70.04	Side Surfacing, Rectangular
G70.05	Side Surfacing, Circular
G70.06	Surfacing
G70.07	Drill Pattern, Line
G70.08	Drill Pattern, Grid
G70.09	Drill Pattern, Arc
G70.10	Contour Pocket
G70.11	Contour Pocket Side Surfacing
G70.21	Object Pattern, Line
G70.22	Object Pattern, Grid
G70.23	Object Pattern, Arc

09	
G73	High speed peck drilling cycle
G74	Left handed tapping cycle〔rigid tapping〕
G76	Fine boring cycle
G80	Turns off cancycle
G81	Drilling cycle, spot boring cycle
G82	Drilling cycle, counter boring cycle
G83	Peck drilling cycle
G84	Right handed tapping cycle〔rigid tapping〕
G85	Boring cycle without spindle stop on the bottom
G86	Boring cycle with spindle stop on the bottom
G87	Back boring cycle
G88	Boring cycle which allows manual motion from the bottom

Class C: One shot commands

	00	
	G04	Dwell time
	G09	One Shot Exact Stop
&	G52	Set Local〔Child〕Coordinate Offset
&	G92	Set absolute zero point G92 External Offset

Note:

1. About the marks:
   * : initial G-code in the group upon power on.
   # : initial G-code which could be selected by parameter setting.
   ^ : the G-code will be memorized when system is shut down.
   & : keep effective until program ends or is changed again.
2. When the machine stops in an emergency or press key **F10** to abort the execution, the tool radius compensation will be canceled automatically. For example:

   Group **07 G40/G41/G42**, the default code is **G40**〔Tool Radius Compensation off〕
   Group **11 G50/G51**, the default code is **G50**〔Scaling/Mirror off〕
   Group **16 G68/G69**, the default code is **G69**〔Rotation off〕
   Group **17 G15/G16**, the default code is **G15**〔Cartesian Coordinate〕

   When program executes normally even the default code of the above-mentioned groups were not given, the above-mentioned groups will restore the default codes.
3. The 〝**one shot G code**〞 is only effective in the specified block. These one shot G codes are all classified to group 00. The others G code are mode, and these mode G code will be kept effective until the other G code of the same group appearing and replacing it.
4. Three different G codes of class A〔Status change G codes〕allow in the same block CNC program at the same time.
   For example: **G18 G91 G16;**
   〔The three are all status-change G codes.〕
   Only one motion G-code allow in a block, but two different Status-change G codes can be

in the same block. Two of the Status-change G codes are before one of motion G codes. For example: **G54 G90 G01 X600. Y400;**
〔**G54** and **G90** are status change G codes; **G01** is the motion G code.〕

5. The one shot G code can only exist alone. No other G codes can exist simultaneously in the same block.
6. **G90** must exist with motion G-codes in the same line of a program, for example, **G09 G01 X500. Y450.;**

4.2 Positioning & Feed instructions

4.2.1 G00／G53：Positioning／Positioning in Machine Coordinate

Format：

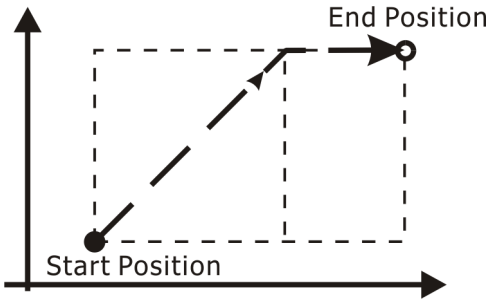
```
G00 X_Y_Z_A_;
G53 X_Y_Z_A_;
```

Command the tool move to the specified target position **X_Y_Z_A_** rapidly. The positioning speed is set by 〝**Machine parameter→Axis→X〔Y or Z〕axis→G00 speed**〞 multiplied the rate of **G00** speed override. **G00** speed override is operated by rotary switch on GMPU.

The moving without cutting usually is programmed by **G00** instruction to save cutting time. The motions are such as positioning from machine origin to cutting start point, retract tool of Z axis upon finishing cutting, positioning of X or Y axis and so on.

G00 mode can position with three axes simultaneously, two axes simultaneously or single

axis. The setting is set by〝**Machine Parameter→Motion and Speed→G00 mode**〞. When you desire to change **G00** mode, please ask your machine maker for service.



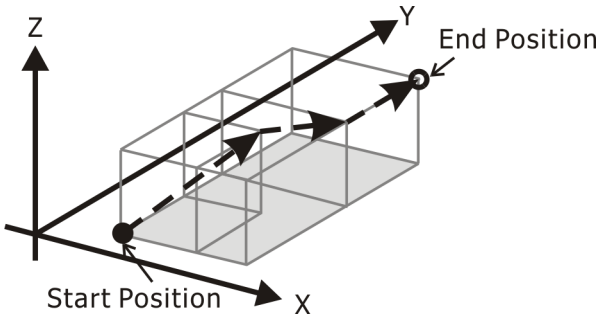


Figure4-3 G00 position with two axes simultaneously Figure4-4 G00 position with three axes simultaneously

G53 moves the tool to the specified target position rapidly like **G00**, and **G53** is specified by the absolute machine coordinate value.

G53 is an One-shot G-code, therefore it is only effective in the block of **G53** specified. This command **G53** is only effective in the absolute mode. If it is programmed with **G53 U_ W_**, the program will alarm error message.

Note: G00/G53 are not relatived with **G17/G18/G19** plane selection.

4.2.2 G01：Linear Interpolation

4.2.2.1 Normal Linear interpolation

Format：

```
G01 X_Y_Z_A_F_;
```

When the counter of work piece is straight line, cut it by **G01** instruction. The position **X_Y_Z_A_** denotes end position of cutting. It can cut with three axes simultaneously, two

axes simultaneously or signal axis. The **F_** value is specified by the feed speed which is adjusted by the rate of feed rate override operated by rotary switch on GMPU.

The unit specified **G94**〔mm / min〕of **F_** is different from **G95**〔mm / rev〕. **F_** is a sustaining command, so it doesn't need to be specified again when the feed speed is the same as one in the current block.

For example, under **G95** mode, the feed distance of **F0.1 S3000** is
0.1〔mm / rev〕× 3000〔rev / min〕= 300 〔mm / min〕

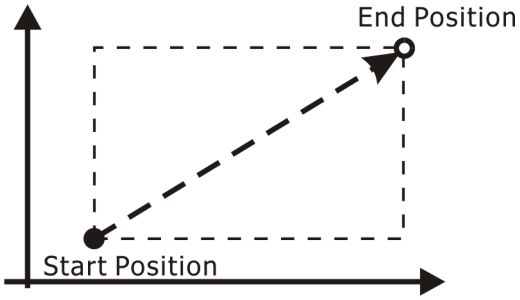


Figure4-5 G01 linear interpolation with two axes

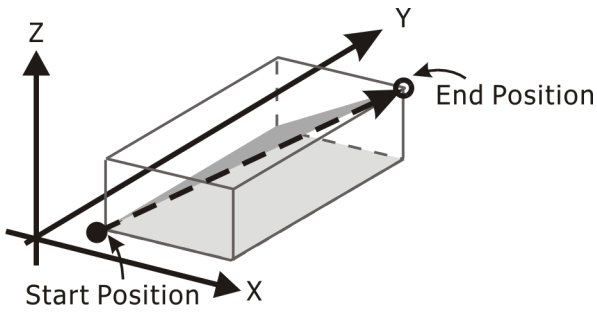


Figure 4-6 G01 linear interpolation with three axes

Note: As for linear interpolation with two axes, **G01** is not related to **G17/G18/G19** plan selection; when execute chamfering or corner rounding, **G01** is related to **G17/G18/G19** plan selection.

4.2.2.2 Chamfering or Corner Rounding

Format：

```
G01 X_Y_L_;
G01 X_Y_R_;
```

Chamfering or corner rounding just executes on a special plane, and its current command and next command both must be **G01**.

Chamfering will add a chamfering command to the corner at the distance **L_** from the corner

and make the corner not so sharp.

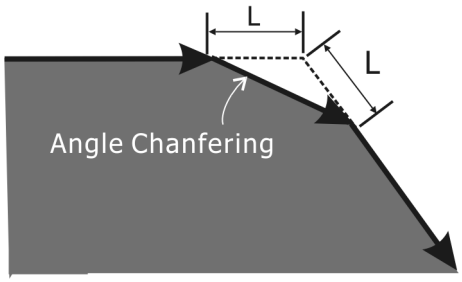


Figure 4-7 Chamfering

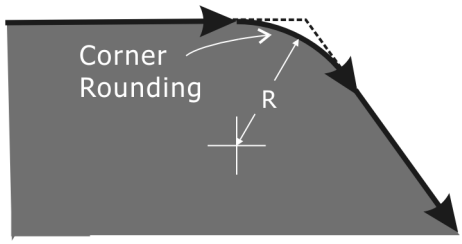


Figure 4-8 Corner Rounding

Corner rounding means add a arc command with the radius **R_** automatically at intersection of two straight lines. It makes two lines tangent with straight line respectively and generate a corner rounding instead of chip.

Note: the length of chamfering 〔**L_**〕 or the radius of corner rounding 〔**R_**〕must be less than the length of both commands.

4.2.2.3 Direct drawing dimensions programming

Angles of straight lines, chamfering value, corner rounding values, and other dimensional values on machining drawings can be programmed by direct input of these values. In addition, the chamfering and corner rounding can be inserted between straight lines having an optional angle.

Example 1：

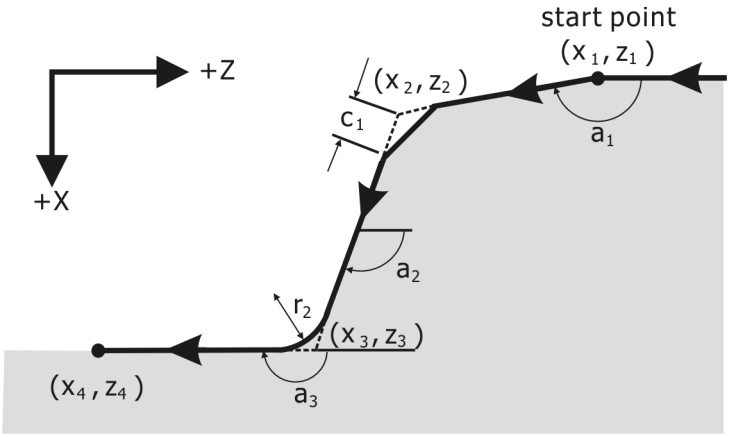
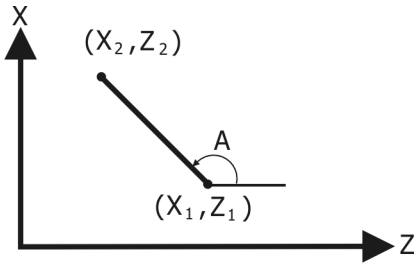
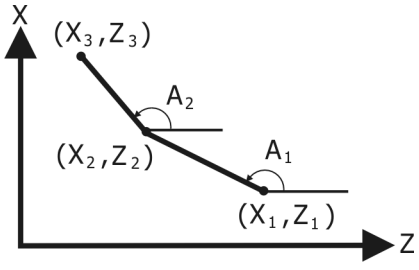


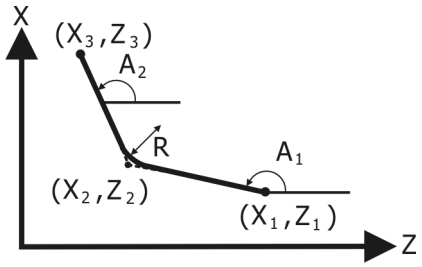
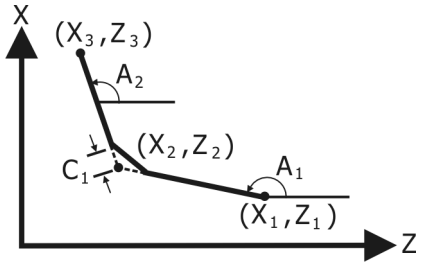
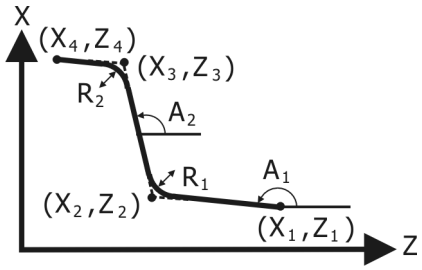
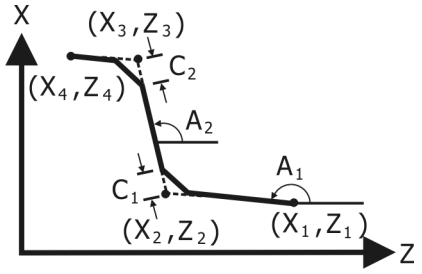
Figure 4-9 Input the angle between lines

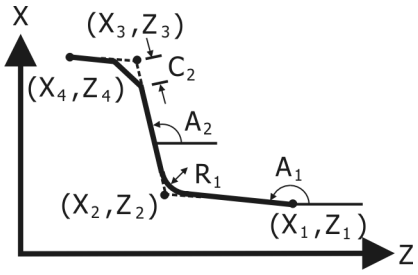
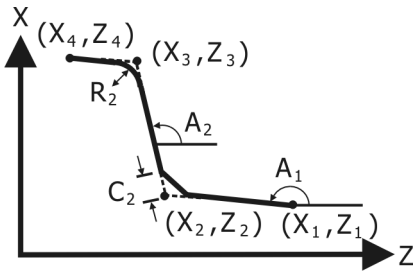
```
X(x2) Z(z2) ,C(c2);          or    A(a1) C(c1);
X(x3) Z(z3) ,R(r2);                X(x3) Z(z3) A(a2) ,R(r2);
X(x4) Z(z4);                       X(x4) Z(z4);
```

Note: To command a straight line, specify one or two out of **X_**, **Z_**, and **A_**. If only one is specified, the straight line must be primarily defined by a command in the next block.

Instruction table

	Instruction table	Tool path
1.	**X2____(Z2____)A____;**	
2.	**A1____;** **X3____ Z3____A2____;**	

	Instruction table	Tool path
3.	**X2_____ Z2_____R1_____;** **X3_____ Z3_____;** or **A1_____ R1_____;** **X3_____ Z3_____ A2_____;**	
4.	**X2_____ Z2_____C1_____;** **X3_____ Z3_____;** or **A1_____ C1_____;** **X3_____ Z3_____ A2_____;**	
5.	**X2_____ Z2_____R1_____;** **X3_____ Z3_____R2_____;** **X4_____ Z4;** or **A1_____ R1_____;** **X3_____ Z3_____ A_____** **R2_____;** **X4_____ Z4;**	
6.	**X2_____ Z2_____C1_____;** **X3_____ Z3_____C2_____;** **X4_____ Z4;** or **A1_____ C1_____;** **X3_____ Z3_____ A2_____** **C2_____;** **X4_____ Z4;**	

	Instruction table	Tool path
7.	**X2_____ Z2_____R1_____;** **X3_____ Z3_____C2_____;** **X4_____ Z4;** or **A1_____ R1_____;** **X3_____ Z3_____ A2_____** **C2_____;** **X4_____ Z4;**	
8.	**X2_____ Z2_____C1_____;** **X3_____ Z3_____R2_____;** **X4_____ Z4;** or **A1_____ C1_____;** **X3_____ Z3_____ A2_____** **R2_____;** **X4_____ Z4;**	

Example 2：Program by the metric system in the diameter mode

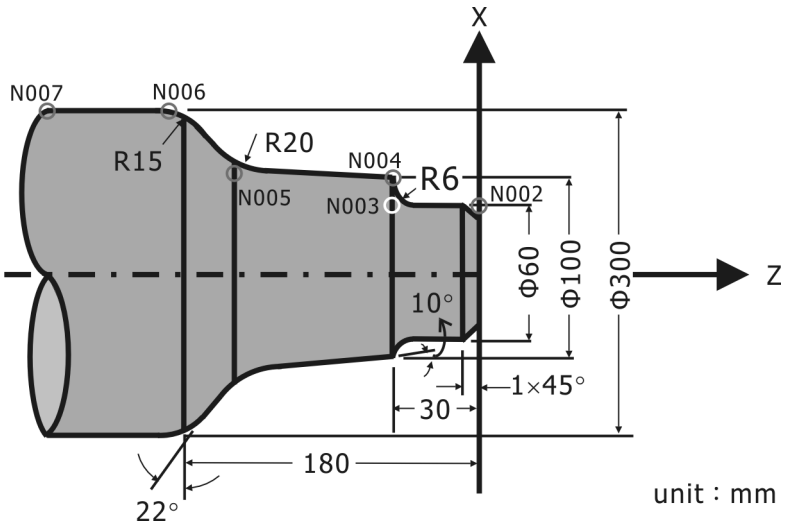


Figure 4-10 Program by the metric system in the diameter mode

```
N001 G50 X0.0 Z0.0
N002 G01 X60.0 A90.0 C1.0 F80;
N003 Z-30.0 A180.0 R6.0;
N004 X100.0 A90.0
N005 A170.0 R20.0
N006 X300.0 Z-180.0 A112.0 R15.0;
N007 Z-230.0 A180.0;
          .
          .
          .
          .
```

4.2.3 G31：Optional Skip

Format：

```
G31 X_ Y_ Z_ A_ F_;
```

G31 executes linear interpolation with specified feed rate. The feed speed is specified by **F_**. The end position is specified by **X_ Y_ Z_ A_**. The feed speed can be adjusted by the rate of feed speed override.

However, when an optional skip input signal is on from PLC, the instruction will skip the rest of the process and stop right away. For example:

```
G31 X100. Y100. Z100. A100. F100.
```

The instruction means the machine moves to **(100.,100.,100.,100.)** with the speed 100 mm⁄mm. If the machine moves at **(X80.,Y80.,Z80.,A80.)** and an optional skip input signal is on from PLC, the instruction will skip the rest of the process and stop at **(X80.,Y80.,Z80.,A80.)** right away.

4.2.4 G02/G03：: CW/CCW Circular

4.2.4.1 Definition of G02/G03

G02: CW〔Clockwise〕Circular Interpolation
This code executes clockwise circular interpolation.

G03: CCW〔Counter-clockwise〕Circular Interpolation
This code executes counter-clockwise circular interpolation.

The arc contour of work piece is programmed by **G02** or **G03** instruction. Because work piece of miler is 3D, the figures of **G02** or **G03** on different plane shows as bellow. The definition denotes look at the specified plane from positive to negative direction along normal axis, clockwise is **G02** but counter clockwise is **G03**. The arc path differs from selected planes, so **G17/G18/G19** must be specified before **G02/G03**.

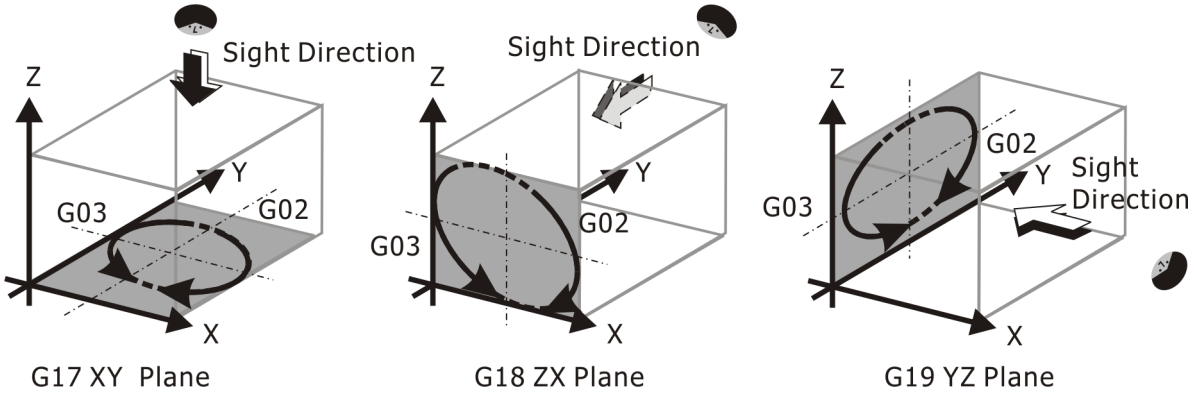


Figure 4-11 The relation of the arc direction and selected plane

1. G02/G03 in 2D motion

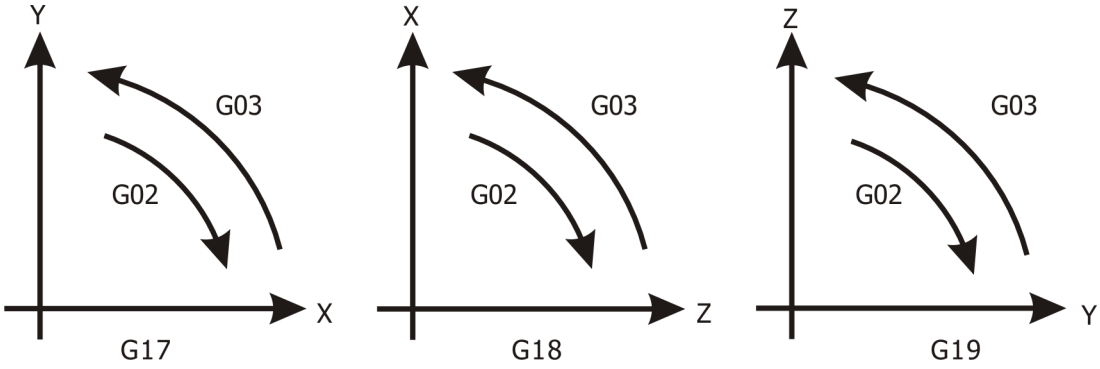


Figure 4-12 G02/G03 in 2D motion

2. G02/G03 in 3D motion

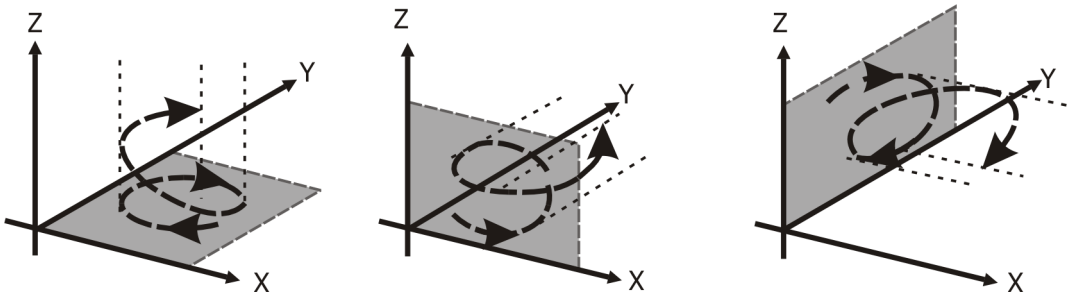


Figure 4-13 G02/G03 in 3D motion〔The figure example is under G02〕

4.2.4.2 Format of G02/G03

There are three types of format in **G02/G03**:

	2D 〔Arc or circle〕	3D 〔Helical〕
Radius	G17 G02(G03) X_Y_R_F_; G18 G02(G03) X_Z_R_F_; G19 G02(G03) Y_Z_R_F_;	G17 G02(G03) X_Y_R_Z_F_; G18 G02(G03) X_Z_R_Y_F_; G19 G02(G03) Y_Z_R_X_F_;
Center	G17 G02(G03) X_Y_I_J_F_; G18 G02(G03) X_Z_I_K_F_; G19 G02(G03) Y_Z_J_K_F_;	G17 G02(G03) X_Y_I_J_Z_F_; G18 G02(G03) X_Z_I_K_Y_F_; G19 G02(G03) Y_Z_J_K_X_F_;
Angle	G17 G02(G03) I_J_A_F_; G18 G02(G03) I_K_A_F_; G19 G02(G03) J_K_A_F_;	G17 G02(G03) I_J_A_Z_F_; G18 G02(G03) I_K_A_Y_F_; G19 G02(G03) J_K_A_X_F_;

I. **Radius Mode**:

A. **Normal mode**:

1. When the code **R_** exists, this mode has the highest priority.
2. Define the end position by the code **X_ Z_** and the radius by the code **R_**.
3. When **R_** is negative, the arc of greater than 180 degree will be executed.

Format：**G18 G02(G03) X_Z_R_;**

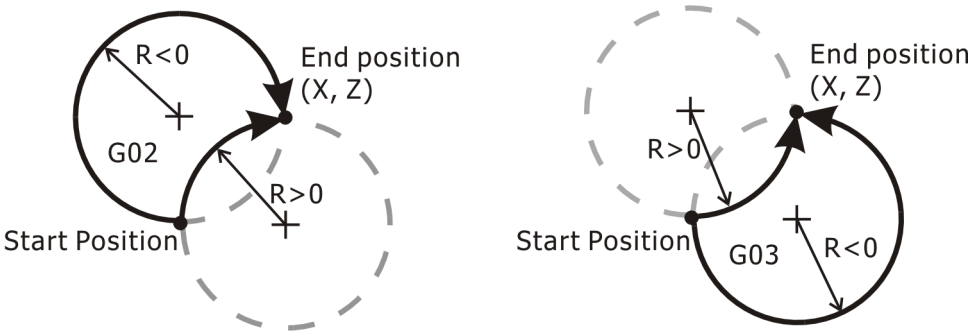


Figure 4-14 normal tool path in radius mode

B. **When d > 2R**:

1. When a radius is less than half of the distance between starting position and end position, a semicircle of the radius **R_** is executed first.
2. The rest of the distance is executed like the mode **G01**.
3. The **G02** mode still keeps effective.

Format: **G18 G02(G03) X_Y_R_;**

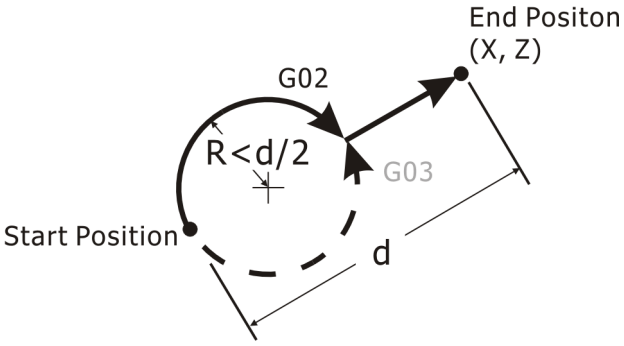


Figure 4-15 tool path in radius mode when d > 2R

C. **When R= 0** : When at least one of **X_,Z_** exists and the code **R_** equals to zero, **G02** is executed like **G01**.

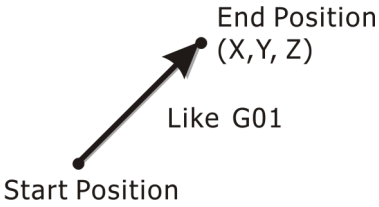


Figure 4-16 Format: **G02(G03) X_Y_Z_R0;** or **X_Y_Z_;**

D. When **X_,Y_,Z_** and **R_** are gone, no motion in the block.

II. Center Mode:

A. **Normal mode**:

1. Define the end position by **X_,Z_** and the center by the codes **I_,K_**.
2. The actual arc of the center **I_,K_** is not sure located on the end position **X_,Z_**.

3. To be located on the end position **X_,Z_**, the center and the radius of the arc will be calculated again.
4. The new radius R' is the average of from the original center to start point and to end point.

Finally, the Center Mode will switch to the Radius Mode with the new radius R'.

Note： 〝**e**〞 is the distance from end point to the center。

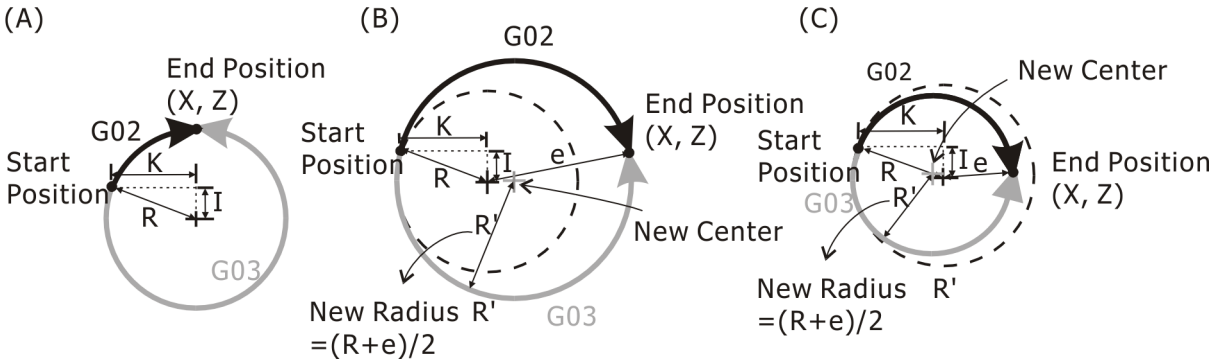


Figure 4-13 Format：**G02(G03)X_Z_I_K_F_;** (A)Normal mode。(B)When e > R時。(C)When e < R時。

B. When **X_,Y_,Z_** are gone, a full circle of the center **I_,J_,K_** is executed.

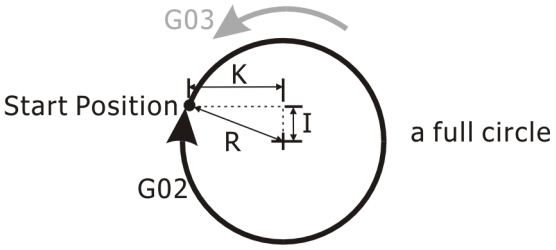


Figure 4-18 G18 G02(G03) I_K_;

When **X_,Y_,Z_** exists, and the codes **I_,J_,K_** are all zero, the center mode is executed like **G01**.

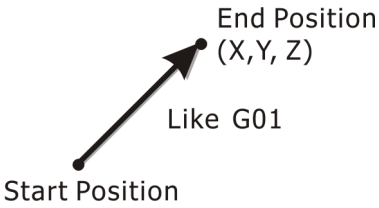


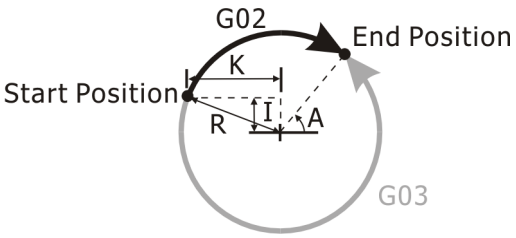
Figure 4-19 **G02(G03)X_Y_Z_I0 J0 K0;** or **G02(G03) X_Y_Z_;**

III. Angle mode

A. **Normal mode**:

1. If no **X_,Y_,Z_** exists, and **A _** is commanded.
2. Define commanded position by **A_**, and center by **I_,J_,K_** codes.

Format : **G90 G18 G02(G03) I_K_A_;**



Format : **G91 G18 G02(G03) I_K_A_;**

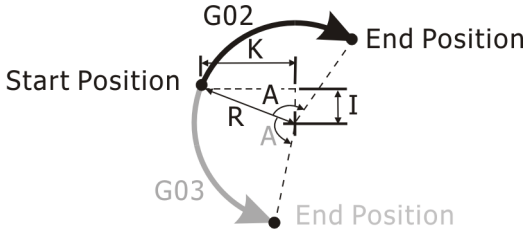


Figure 4-20 Angle mode. (A) Absolute format (B) Incremental format.

If **A_** code equals to zero under absolute mode, the motion is as below.

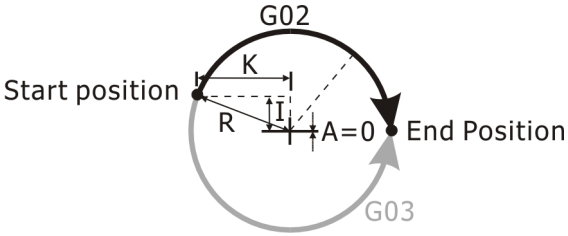


Figure 4-21 When A_ code equals to zero under absolute mode

If **I_,J_,K_** codes all equals to zero, no motion for the section.

If **I_J_K_** codes all equals to zero or **A_** code equals zero under increment mode, no motion for the section.

4.2.5 G33：Thread Cutting

Format：

```
G33 X_Y_Z_A_F_;
```

G33 works with **G95** at the same time. **G95** set the feedrate per revolution by the code **F_** , and make tools do surface cutting according to the rate of the spindle shifting the distance **F_** per revolution in X ,Y ,Z or A axis. The difference of **G33** and **G01** is that **G33** lead thread cutting with start angle specified by **A _**.That is all the reason why **G33** can thread cutting.

Note：**G95** must be specified previously to the **G33** command. And the spindle should have implemented the encode in order to do the synchronous control.

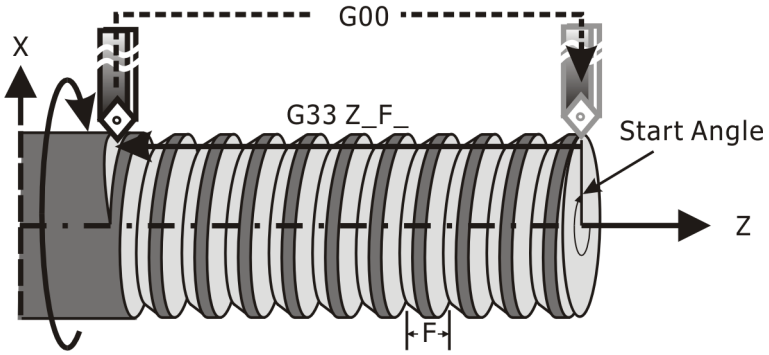


Figure 4-22 G33：Thread Cutting

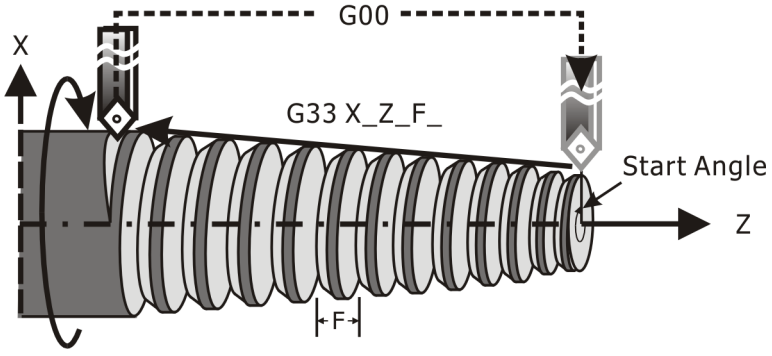


Figure 4-23 G33 X_Z_F_

Note: The dotted line of the coming back is added by **G00**, but not included by **G33**.

4.3 Tool Compensation Instructions

4.3.1 G43/G49：Tool Length Compensation on／off

Format：

```
G43 H_;
G49;
```

G43 initializes the tool length compensation.

The sign and value of tool length in the tool table decides the +/- direction and length of the compensation in 3rd axis〔for example, Z axis in **G17**〕.
G49 cancels the tool length compensation, and restores to normal condition in 3rd axis.

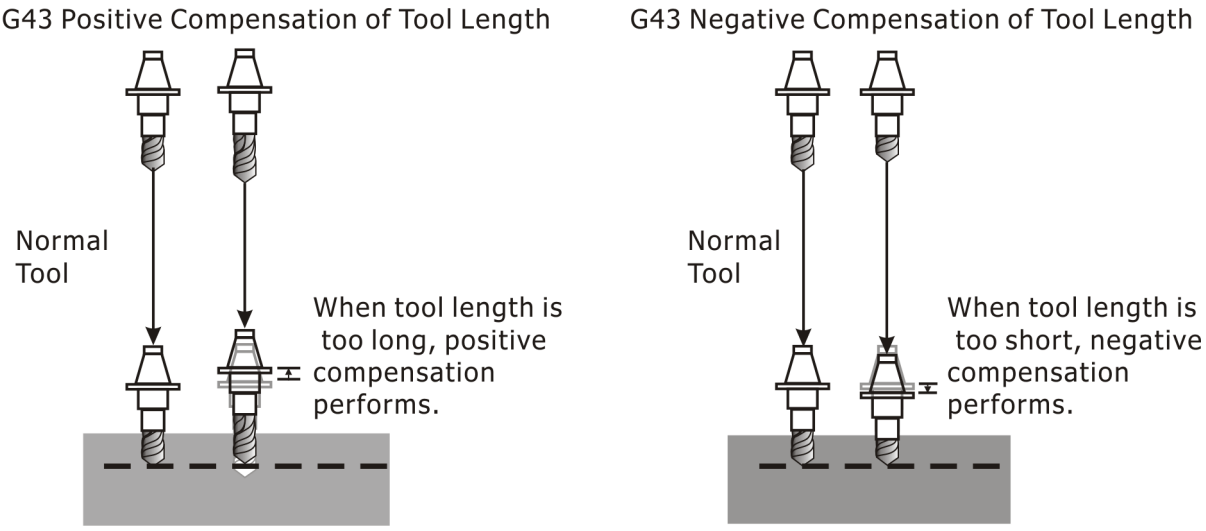


Figure 4-24 Tool length compensation

4.3.2 G37：Automatic tool length measuring

Format：

```
G37 Z_ Q_ D_ F_ E_;
```

where

desired machine position : defined by **Z_**〔**G37**〕
probe retract distance : defined by **Q_**
probe detect distance : defined by **D_**
probe searching speed : defined by **F_**
probe detect speed : defined by **E_**

G37 performs automatic tool measurement. It moves the tool towards measurement device in probe search speed until probe is detected. Then moves backward until the tool leaves the measurement surface. The tool is then retracted in search speed by retract distance. After that, probe moves toward the measurement device again in detect speed until probe is detected again. After all these process, the difference between the desired machine position and the current machine position is set into the current tool table entry automatically.

Before uses the **G37** command, make sure:

1. The machine tool has implemented the tool measurement sensor〔probe〕
2. **H_** value has been specified before giving the **G37** command.(This **H_** value can not be **H00**〕
3. The tool is to be measurement, has been moved to the upper place of the sensor.

The following procedure of tool measurement will be performed automatically:

1. At beginning, the tool moves in middle speed〔**F_** value〕to approach the sensor, until the tool tip touched the sensor. If exceeding **D_** distance and no touch the sensor, an alarm will be popped out and abort the tool measurement.
2. If the tool tip touches the sensor within **D_** distance, then it will retract **Q_** distance in **F_** speed.
3. And then use very low speed〔**E_** value〕to approach and to touch the sensor again. As

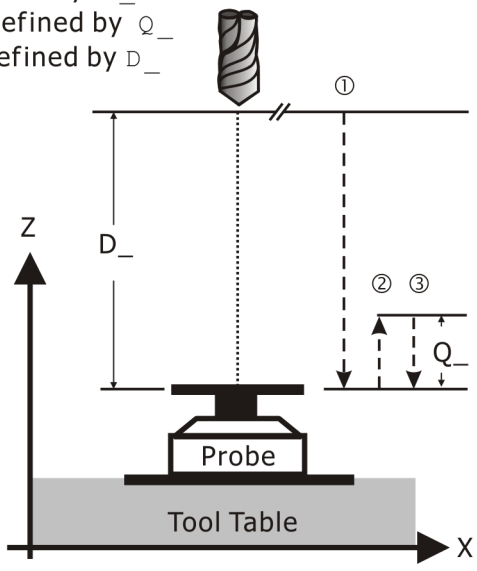
soon as the tool tip touches the sensor, the motion stops.

4. Read the current spindle axis value〔absolute position〕, and then calculate the difference from the expected value. Add this difference value to the respective length item 〔specified by **H_**〕in the tool table, to make the current tool tip position to match the expected value.

Note:

1. If **Z** code does not exist, **X/Z**=0 is assumed.
   Other codes〔**Q_ D_ F_ E_**〕can use the value defined in **Z** tool length measurement of user parameter.
2. The searching direction is negative if **Q_** value is positive.

Corresponding parameter：
probe searching speed : defined by F_
probe detect speed : defined by E_
probe retract distance : defined by Q_
probe detect distance : defined by D_



① Touch the sensor with middle speed F_
② Retract distance Q_ with speed F_
③ Touch the sensor with speed E_
④ Calculate the offset and update the tool table automatically

Figure 4-25 Automatic tool offset measurement

4.3.3 G40/G41/G42：Tool Radius Compensation off / on

Format：

```
G41 D_;
G42 D_;
G40;
```

G40: Tool Radius Compensation off

G40 turns off tool nose compensation, the next motion command following **G40** has to be a **G00** or **G01** to transit out of the nose compensation mode.

G41: Left Tool Radius Compensation

G41 turns on tool nose compensation along the left side of the original profile path. The tool radius used is the value stored in tool table of the current tool offset number. The motion codes between **G41** and **G40** must be **G01**, **G02** or **G03**.

G42: Right Tool Radius Compensation

G42 turns on tool nose compensation along the right side of the original profile path. The tool radius used is the value stored in tool table of the current tool offset number. The motion codes between **G42** and **G40** must be **G01**, **G02** or **G03**.
The address **D_** is a tool number of the tool table. The tool number **D_** can be specified before the command **G41/G42**, and it keeps effective until the next new tool number **D_** is executed. The tool number **D00** means no compensation 〔the radius is zero〕.

G40 cancels the tool radius compensation〔transit out〕, and restores to the normal tool path. The new tool nose is very sharp with an angle 〔as shown in the left figure〕. It becomes round after a period of cutting〔as shown in the right figure〕.

This system follows the following rules to execute tool radius compensation in the mode :

A. When the angle to the compensated side between two continuous blocks is larger than 180 degree, this system inserts an arc between the two block motion.

(a>180°)

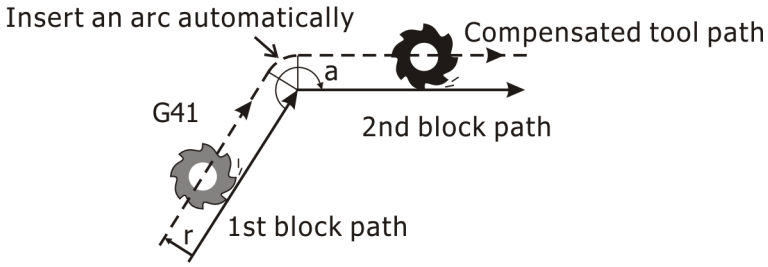


Figure 4-26 the tool compensation inserts an arc between the two block motion

2. When the angle to the compensated side between two continuous blocks is smaller than 180 degree, this system calculates the intersection of the two paths automatically. This intersection point becomes the end point of the first block motion and the start point of the second block motion.

(a<180°)

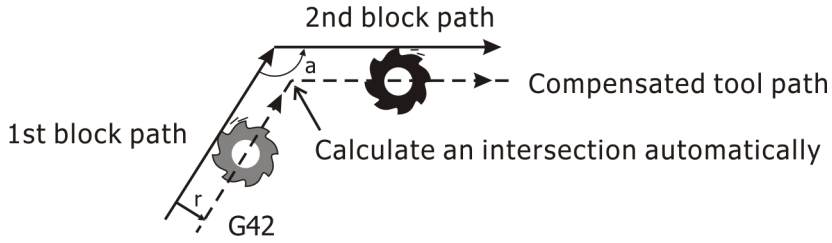


Figure 4-27 the tool compensation inserts an intersection point between the two block motion

Note:

1. Linear Interpolation〔**G01**〕must be used when tool transits into or out the compensation. Use circular interpolation 〔**G02/G03**〕at the same time, the machine will alarm and turn off automatically.
2. To change of radius , exchange from **G41** to **G42** or exchange from **G42** to **G41** is not allowed in the tool compensation mode. A full circle format of **G02/G03** can't be executed in **G41/G42** mode.

4.3.3.1 Transit into the compensation〔G41/G42 begins compensating〕

Must enable compensation at the last block before transiting into the compensation. When transit compensation into target position, the tool will shift left / right the distance of a radius. In the following figures, the broken line denotes compensated tool path, but the straight line denotes original programmed tool path.

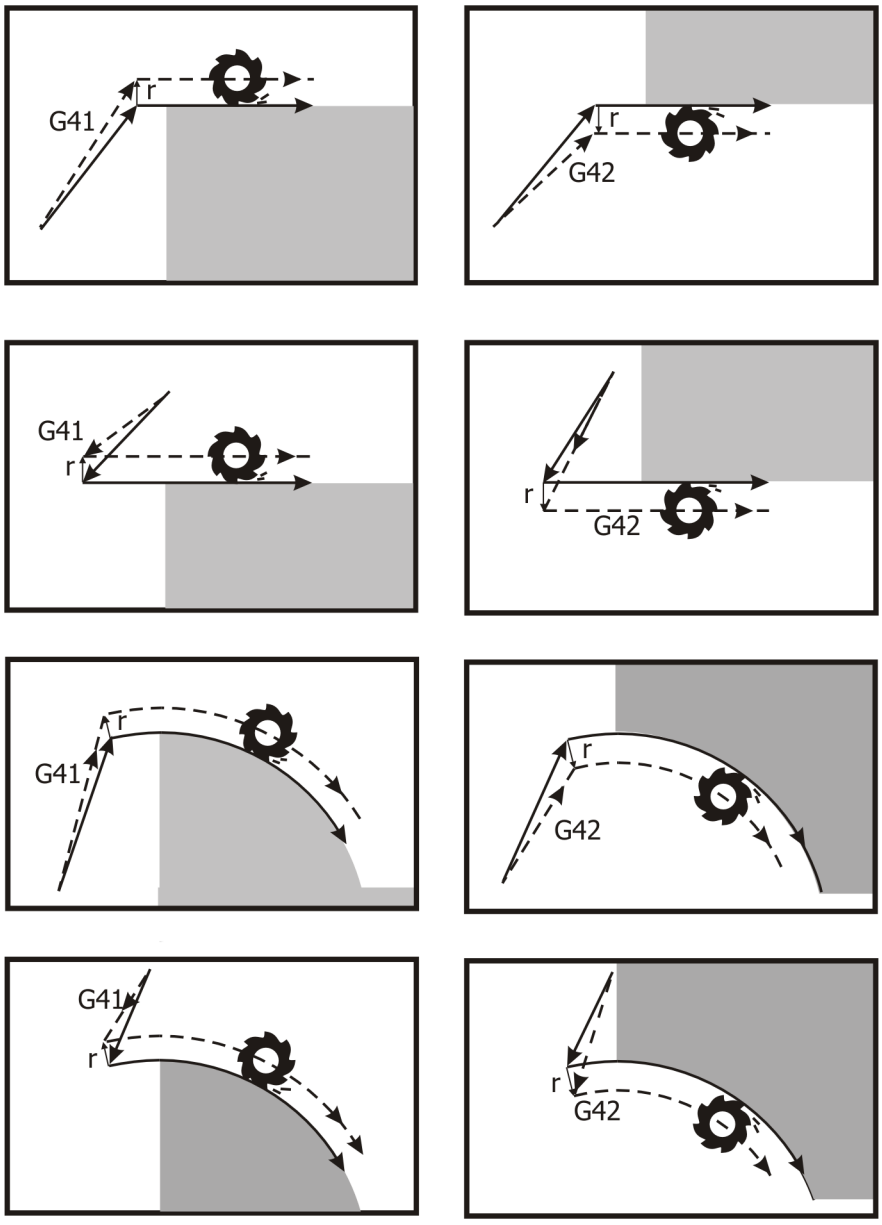


Figure 4-28 Transit into the compensation〔G41/G42 begins compensating〕

4.3.3.2 In the compensation mode

1. From straight line to straight line

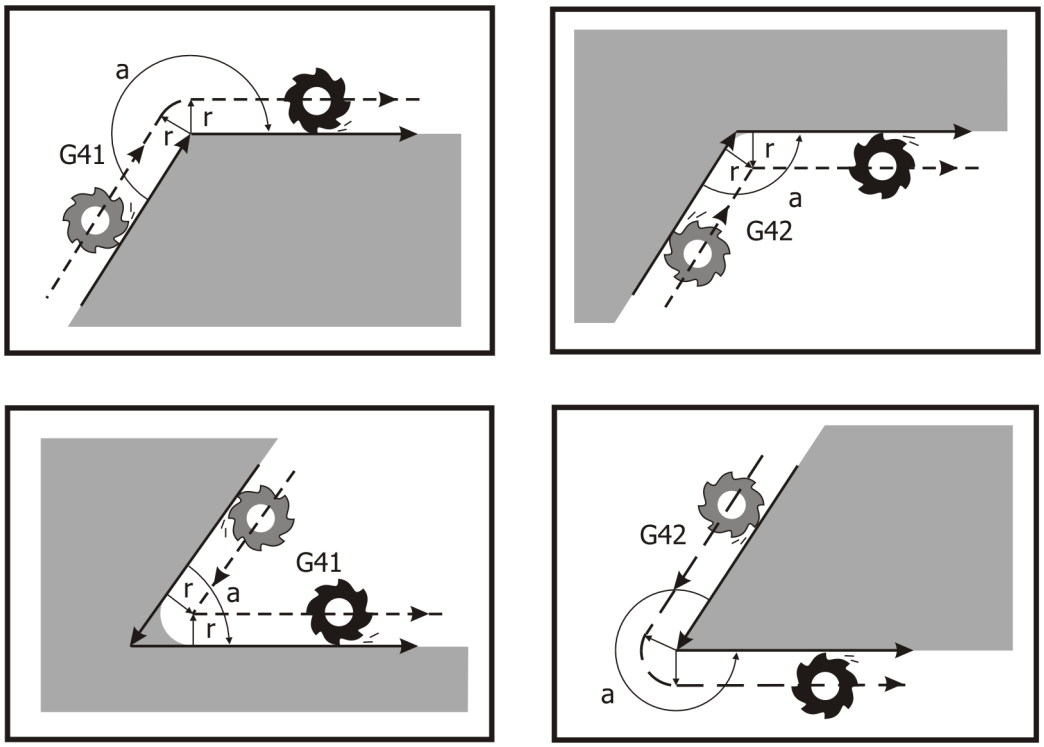


Figure 4-29 In the compensation mode: From straight line to straight line

2. From straight line to arc

(1) From straight line to CW arc

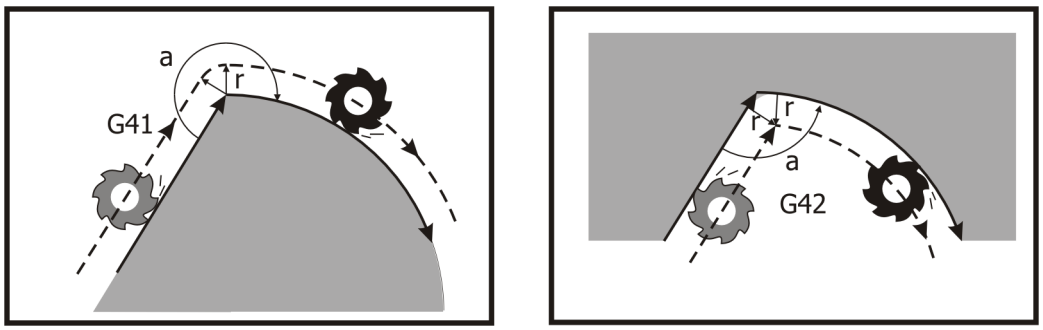


Figure 4-30 In the compensation mode: From straight line to CW arc

(2) From straight line to CCW arc

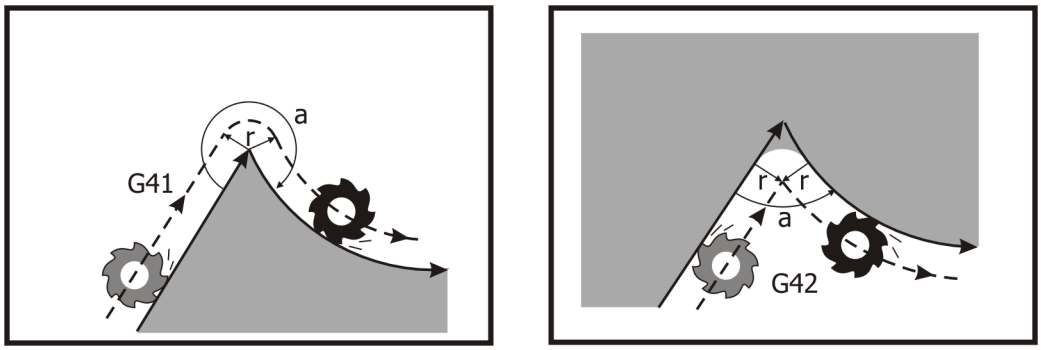


Figure 4-31 In the compensation mode: From straight line to CCW arc

3. From arc to arc

(1) From CW arc to CW arc

Figure 4-32 In the compensation mode: From CW arc to CW arc

(2) From CW arc to CCW arc

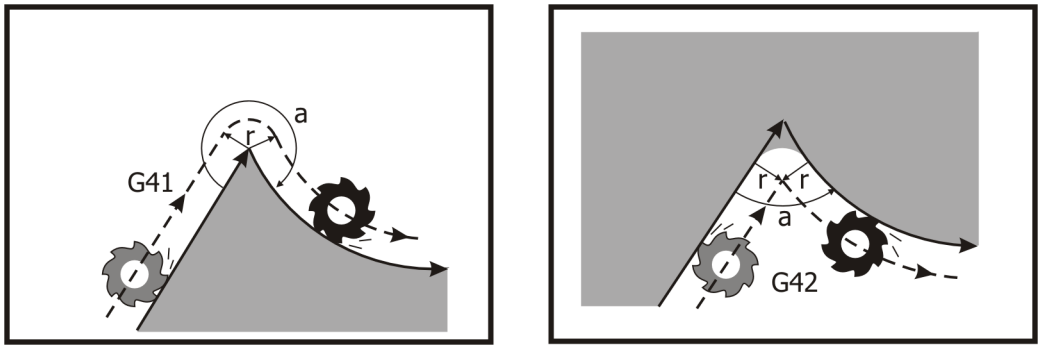


Figure 4-33 In the compensation mode: From CW arc to CCW arc

(3) From CCW arc to CCW arc

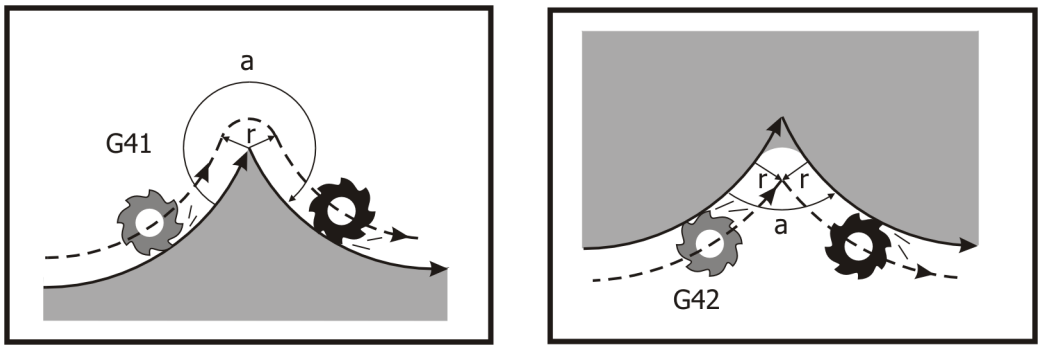


Figure 4-34 In the compensation mode: From CCW arc to CCW arc

4.3.3.3 Transit out the compensation〔G40 finishes compensating〕

Upon finishing compensation, the tool will return to the original programmed tool path.

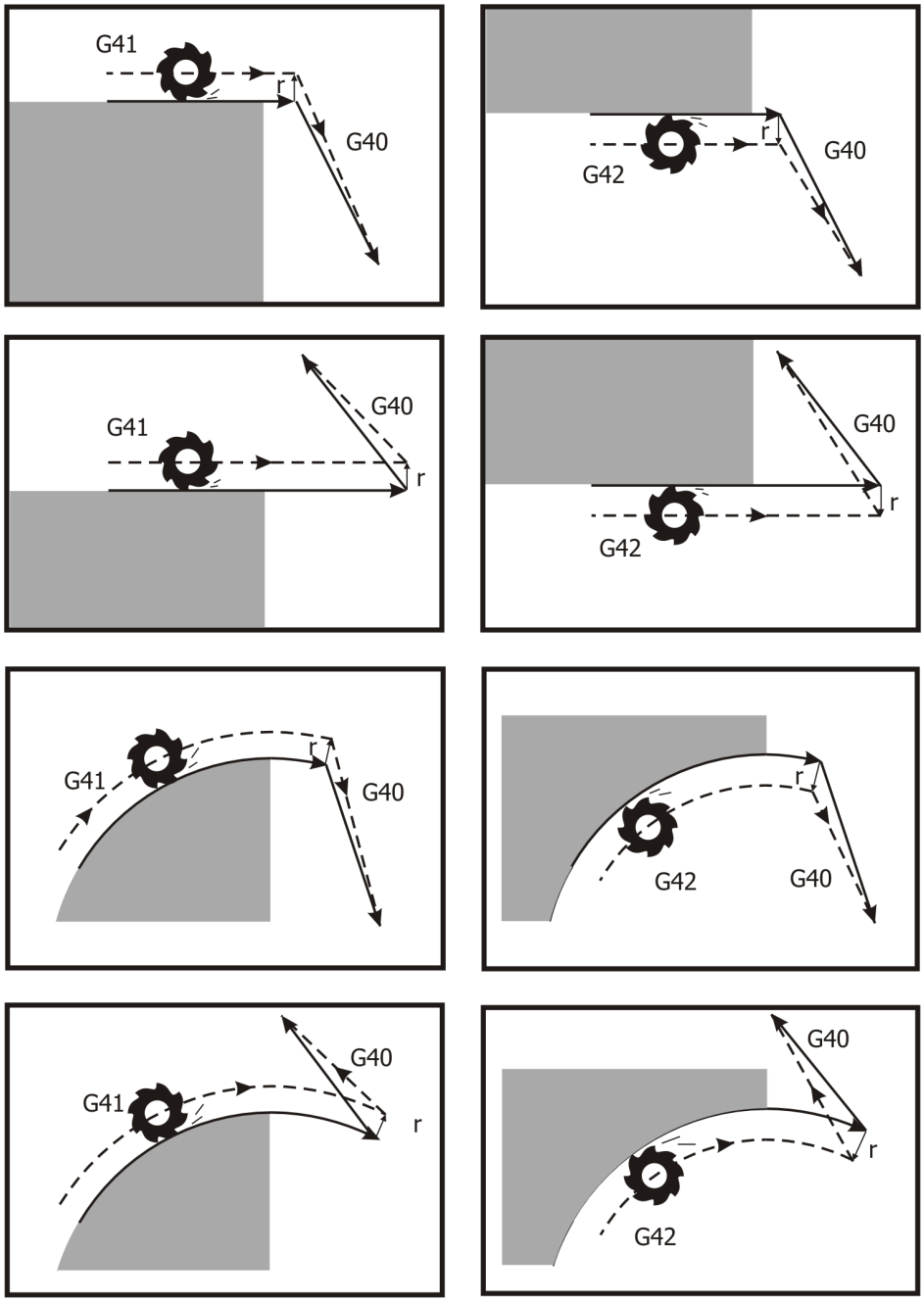


Figure 4-35 Transit out the compensation〔G40 finishes compensating〕

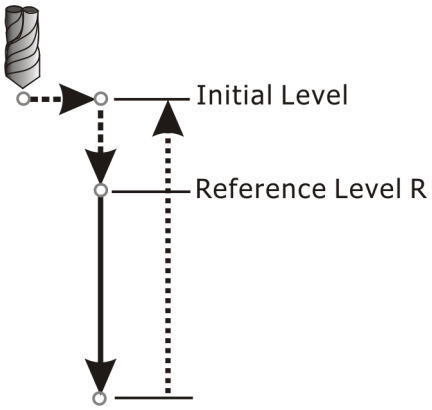
4.4 Cancycles Instructions

4.4.1 G98/G99：return to initial position /reference position in cancycle

Format：

G98; return to initial position in cancycle
G99; return to reference position in cancycle

Can Cycle instructions can execute drilling, boring and tapping. All the motions of these instructions are similar, as the following two figures:



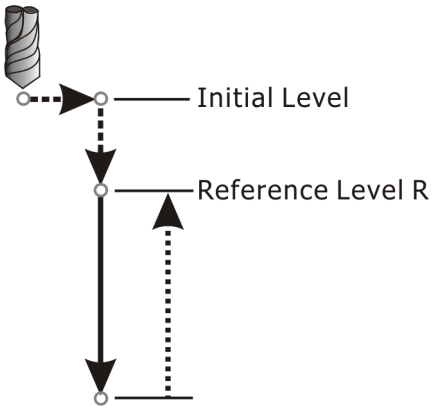


Figure 4-36 G98 returns to initial position in cancycle Figure 4-37 G99 returns to reference position in cancycle

The left figure **G98** denotes the tool returns to initial position upon finishing; the right figure **G99** denotes the tool returns to reference position upon finishing.

These cancycle instructions give the cutting mode of the first hole, the following holes just give their cutting positions, and then they will do repeating motions until **G80** cancel cancycle. The Class A G-codes , such as **G00/G01/G02/G03/G33**, occurring is treated as cancycle cancel.

As for reference position in cancycle instructions, **R_** denotes the value of reference position under **G90**, absolute dimension. Under **G91**, increment dimension, **R_** denotes the distance from initial position to reference position, and the value of **R_** is always positive.

As for drilling depth **Z_** specified by **G90/G91**: **Z_** denotes the value of drilling depth under **G90**, but **Z_** denotes the increment distance from reference position to drilling depth under **G91** and the distance could be positive or negative.

The motions of above-mentioned commands are specified respectively by **G90/G91** and the figures is as below:

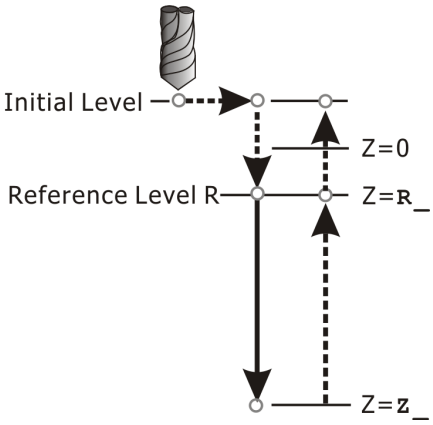


Figure 4-38 G90

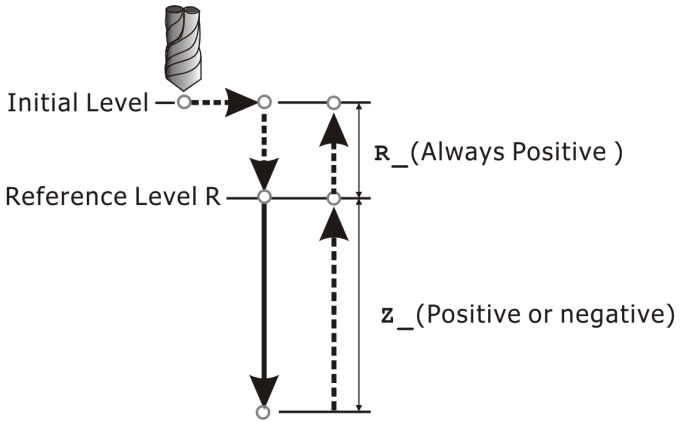


Figure 4-39 G91

In addition, all cancycle instructions can give **K_** to repeat the specified cancycle instruction **K_** times alone a straight or tile line, such as **Line Drill Pattern**.

For example, "**G91 Gxx Xdx Ydy Z_ R_ F_ Kk;** "means repeat to execute some cancycle instruction **Gxx** 〔**G73~G89**〕**k_** times, **dx** and **dy** denotes respectively the interval between consecutive holes of X and Y axis. The instruction repeats cancycle along a straight or tile line makes cutting more convenient.

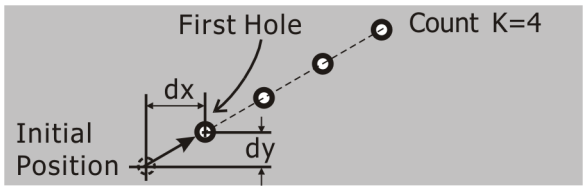


Figure 4-40 Repeat cancycle under G91 mode

Note: **K_** is nonsense under **G90** mode, because it will repeat cancycle at the same hole.

Some of cancycle instructions, such as **G74,G76,G82,G84,G87,G88** and **G89**, can give **P_** to dwell **P_** seconds at the drilling depth **Z_**. This commands can make the drilling depth more precise or make the spindle having buffer time to reverse.

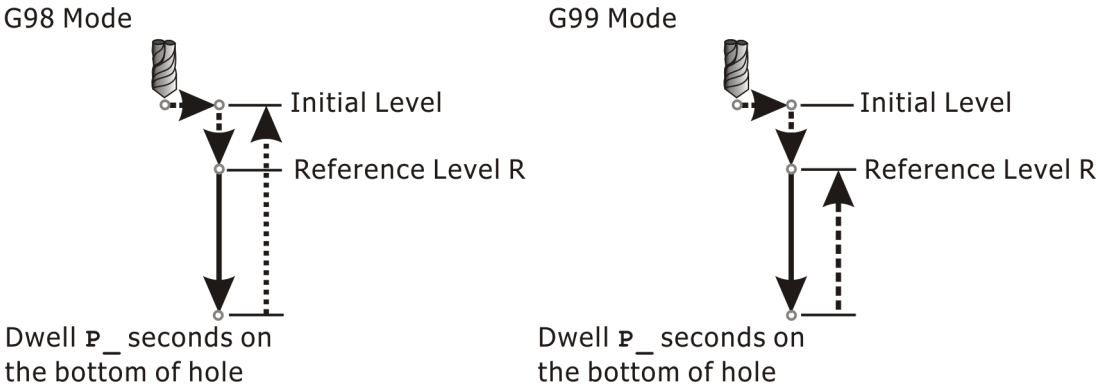


Figure 4-41 Dwell P_ seconds at the drilling depth Z_

4.4.2 G80：Turns off cancycle

Format：

G80;

Because cancycle instructions functions are sustained, don't specify again in every block when executing the same cancycle instruction. When finishing cancycle and don't execute it anymore, must specify **G80** to cancel cancycle and restore normal mode, such as **G00**, **G01**, **G02** and **G03**.

4.4.3 G81/G82, G73/G83：Drilling Cycle

Format：

```
G81 X_Y_Z_R_K_F_;
G82 X_Y_Z_R_P_K_F_;
G73 X_Y_Z_R_Q_K_F_;
G83 X_Y_Z_R_Q_K_F_;
```

G81： drilling cycle, spot boring cycle
G82： drilling cycle, counter boring cycle

G81/G82 is a simple drilling, it can drill to the bottom once. Execute the simple boring sometimes by **G81/G82**. Executing **G81/G82** motions show as the following figure:

1. Move the drill rapidly to position specified by X and Y axis.
2. Position 〔**G00**〕to reference position **R_**.
3. Drill to the bottom **Z_** of specified hole with the specified feed speed **F_**.
4. Retract the drill rapidly to initial position〔**G98**〕or reference position〔**G99**〕to finish cancycle.

Note: Straight line denotes feed speed **F_** and broken line denotes **G00** speed in the following figures.

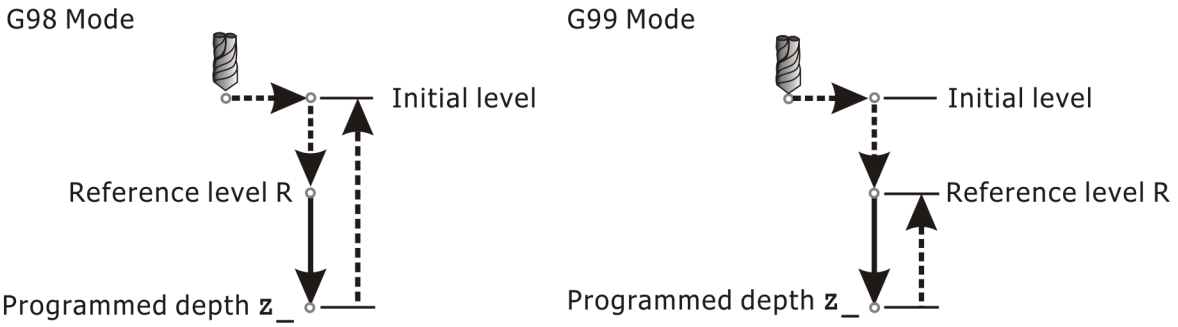


Figure 4-42 G81： drilling cycle, spot boring cycle

Besides pause **P_** seconds in the bottom of specified hole, other motions of **G82** are the same as ones of **G81**. **G82** makes the drill pause **P_** seconds in the bottom of specified hole to

improve the precision of bottom of blind hole, beam hole and taper hole.

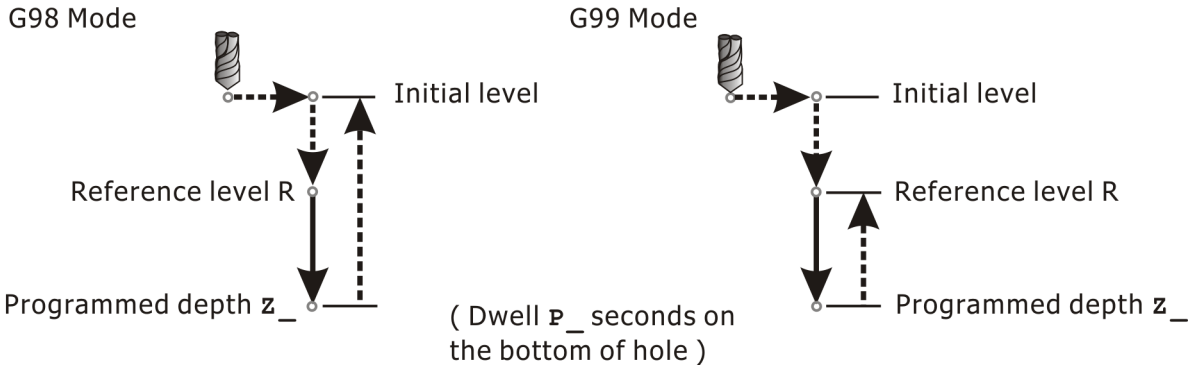


Figure 4-43 **G82**： drilling cycle, counter boring cycle

G73：high speed peck drilling cycle
G83：peck drilling cycle

G73/G83 is a complex drilling, it can peak section by section instead of drilling to the bottom once. Every section motion can retract the tool and remove from filings.

The motion of **G73** shows as the following figures :

1. Move the drill rapidly to position specified by X and Y axis.
2. Position 〔**G00**〕to reference position **R_**.
3. Drill the specified distance **Q_**〔**Q_** is always a positive value〕along Z axis with the specified feed speed **F_**.
4. Retract the drill back in a distance d rapidly.
5. Repeat Step 3. ~ Step 4. once and once until reach to the specified bottom position **Z_**.

The interval drilling make filings broken and cutting oil reach to the contour easily.

Only retract a fixed distance d which is set by 〝**Parameter→Cancycle Parameters**〞 when retract tool and remove from filings under **G73** mode.

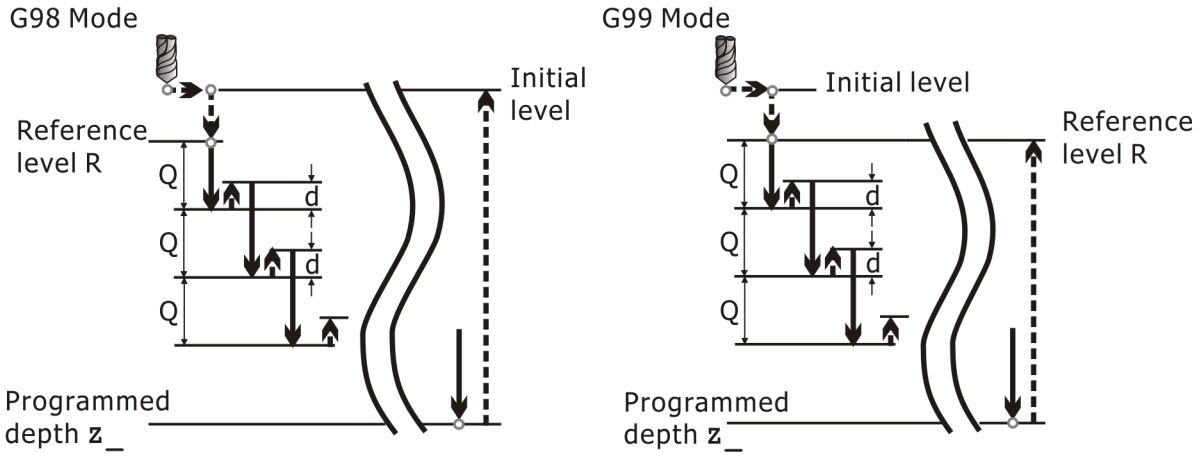


Figure 4-44 G73：high speed peck drilling cycle

The motions of **G83** are similar to ones of **G73**. The difference between **G83** and **G73** is that retract tool to reference position **R_** under **G83** mode. This motion can remove the filings out of hole and avoid the hole full of filings to increase cutting resistance or make cutting oil not reach to the contour. This is the reason why **G83** is suited to deep hole drilling.

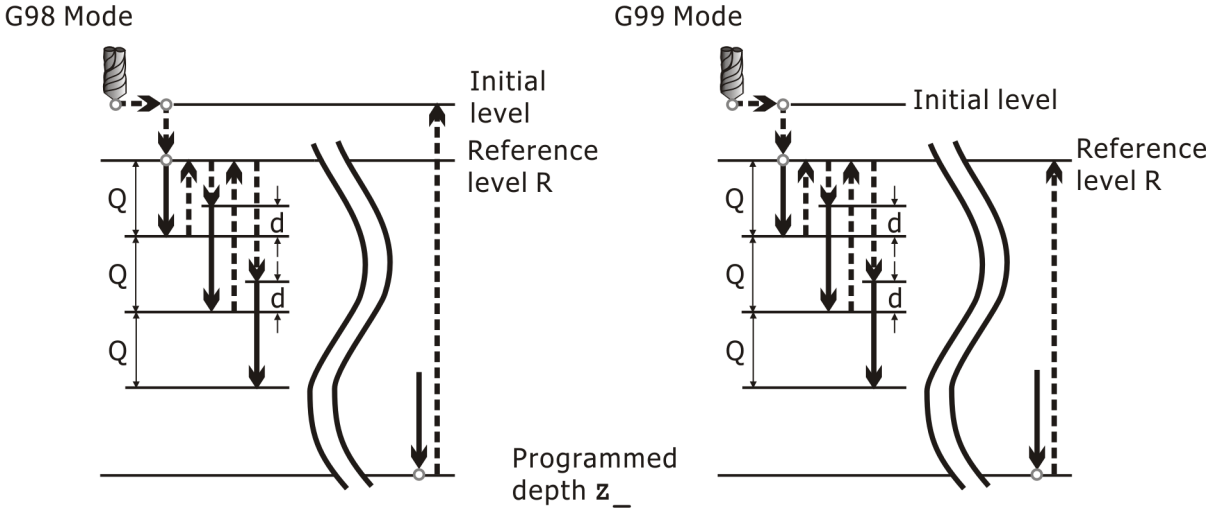


Figure 4-45 G83：peck drilling cycle

4.4.4 G85/G86、G76/G87、G88、G89：boring cycle

Format：

```
G85 X_Y_Z_R_K_F_;
G86 X_Y_Z_R_K_F_;
G76 X_Y_Z_R_Q_P_K_F_;
G87 X_Y_Z_R_Q_P_K_F_;
G88 X_Y_Z_R_P_K_F_;
G89 X_Y_Z_R_P_K_F_;
```

4.4.4.1 G85/G86：boring cycle without / with spindle stop on the bottom

The motion of **G85** shows as the following figures :

1. Move the reamer rapidly to position specified by X and Y axis.
2. Position 〔**G00**〕to reference position **R_**.
3. Ream downward to the specified position **Z_** in the bottom of specified hole with the specified feed speed **F_**.
4. Retract the reamer back with feed speed **F_**. This is the reason why **G85** is suited to reaming.

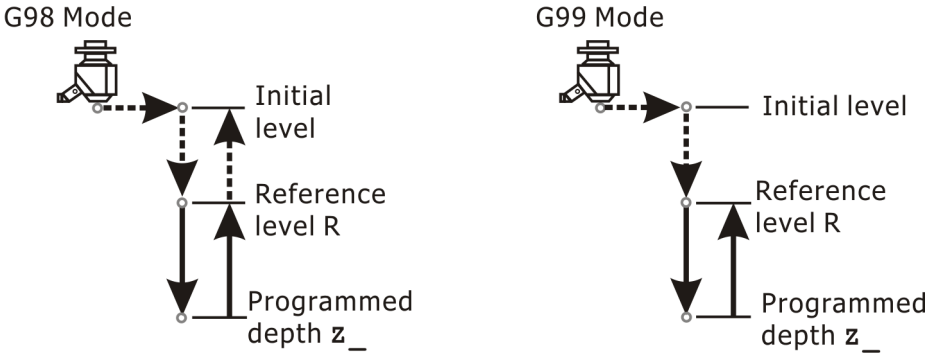


Figure 4-46 G85: boring cycle without spindle stop on the bottom

Stop the spindle upon the reamer reams at the cutting depth **Z_**, retract to initial position 〔**G98**〕or reference position〔**G99**〕, and then restore the spindle clockwise〔CW〕.

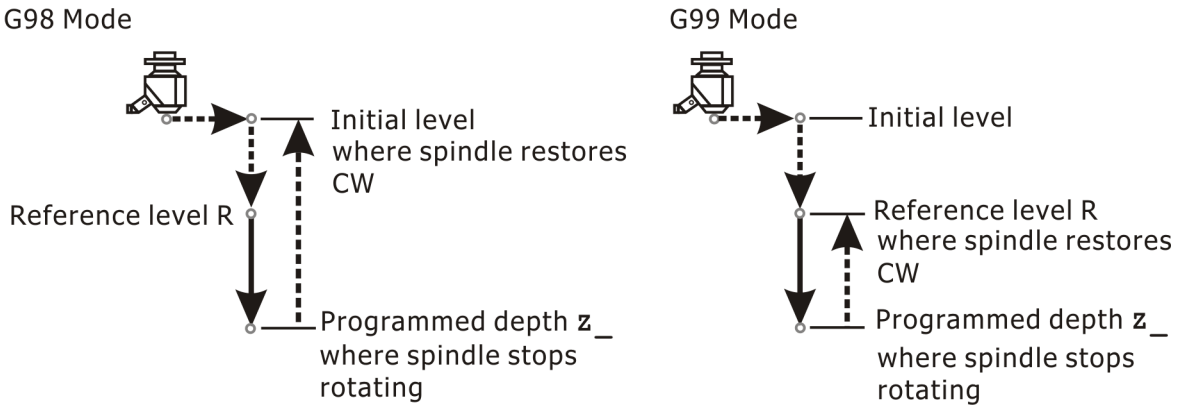


Figure 4-47 G86: boring cycle with spindle stop on the bottom

4.4.4.2 G76/G87：Fine Boring Cycle / Back Boring Cycle

Both **G76** and **G87** execute the motion which is stopping the spindle at the specified direction. That is the reason why **G76** and **G83** can do fine boring.

The motion of **G76** shows as the following figures :

1. Move the boring cutter rapidly to position **X_,Y_**.
2. Position 〔**G00**〕to reference position **R_**.
3. Bore downward to the specified position **Z_** in the bottom of specified hole with the specified feed speed **F_** .
4. Stop the spindle toward a specified direction.
5. The center of the boring cutter toward a specified direction shift a little distance **Q_**, and make the boring cutter leave the cutting contour.
6. Retract the boring cutter back to initial position〔**G98**〕or reference position 〔**G99**〕with **G00** speed. Step 5.~Step 6. protects the cutting contour from scraping.
7. Position the boring cutter to position **X_,Y_** and restore the spindle clockwise〔CW〕.

Note: the shift value **Q_** must be positive and can't be too large to crash work piece.

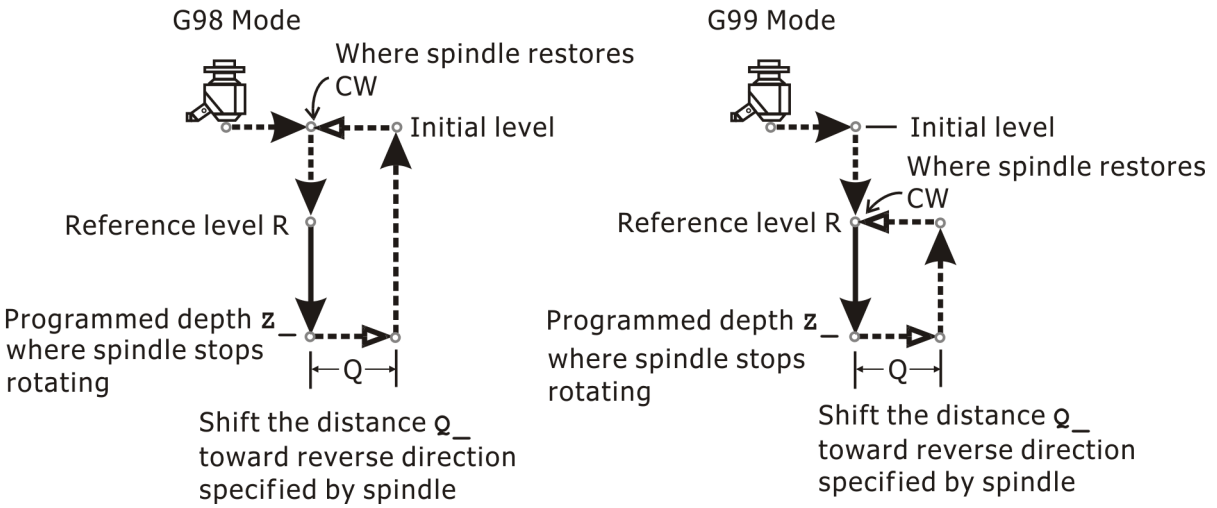


Figure 4-48 G76: fine boring cycle

The motion of **G87** shows as the following figures :

1. Move the boring cutter rapidly to position **X_,Y_**.
2. Stop the spindle toward a specified direction.
3. The center of the boring cutter toward a specified direction shift a few distance **Q_**, make the boring cutter leave the cutting contour and move the boring cutter downward. This protects the cutting contour from scraping.
4. Position 〔**G00**〕to reference position **R_** on the bottom of hole.
5. Shift the distance **Q_** toward original direction and restore the spindle clockwise〔CW〕.
6. Bore upward to the specified position **Z_** with the specified feed speed **F_** .
7. Stop the spindle and make the boring cutter leave the cutting contour a few distance **Q_**.
8. Retract the boring cutter back to initial level.
9. Shift the distance **Q_** toward original direction and go back to initial position **X_Y_**. That is all motions of **G87**.

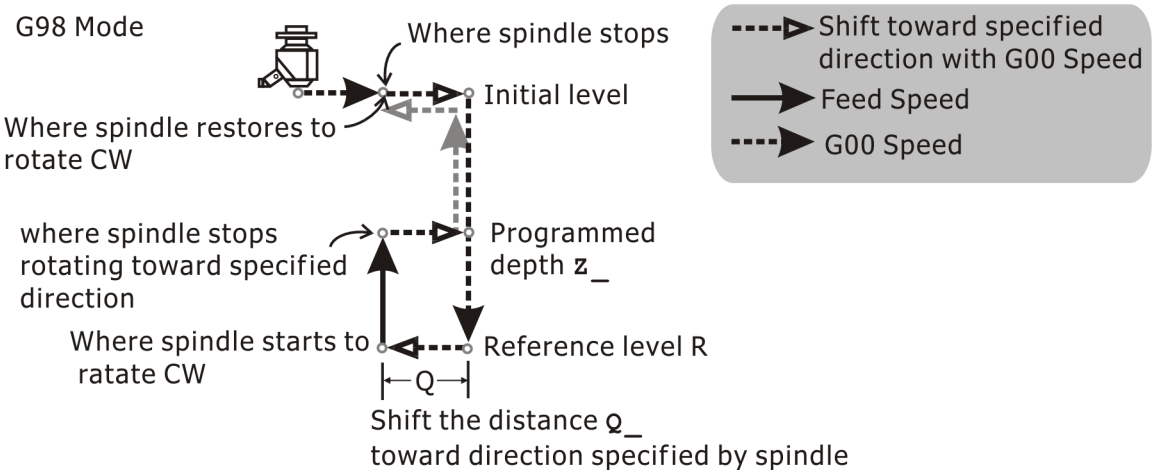


Figure 4-49 G87: back boring cycle

Note: in **G87**, the definitions of reference level **R_** and programmed depth **Z_** are different from ones of other cancycle instructions. The reference level **R_** locates at the lowest position whereas the programmed depth **Z_** locates above the **R_** level. Due to this fact, **G87** cannot be programmed in **G99** mode.

4.4.4.3 G88：boring cycle which allows manual motion from the bottom

Under **G88** mode, when boring to the programmed depth **Z_**, stop the spindle **P_** seconds temporarily. Then, enable manual mode to make tool back to reference position **R_** and leave manual mode by 2nd operating panel. If tool doesn't reach over the reference position **R_**, the alarm dialog box will pop up and can't allow to leave manual mode.

Note: Upon making sure the tool position has been over reference position **R_**, restore the spindle clockwise〔CW〕.For safety, please keep in mind to let head and hands far to spindle. If **G88** is programmed by **G98** dimension, tool will continue upward to initial level, but by **G99** dimension tool will not move anymore. The finish position doesn't restore to the position specified by **G88**.

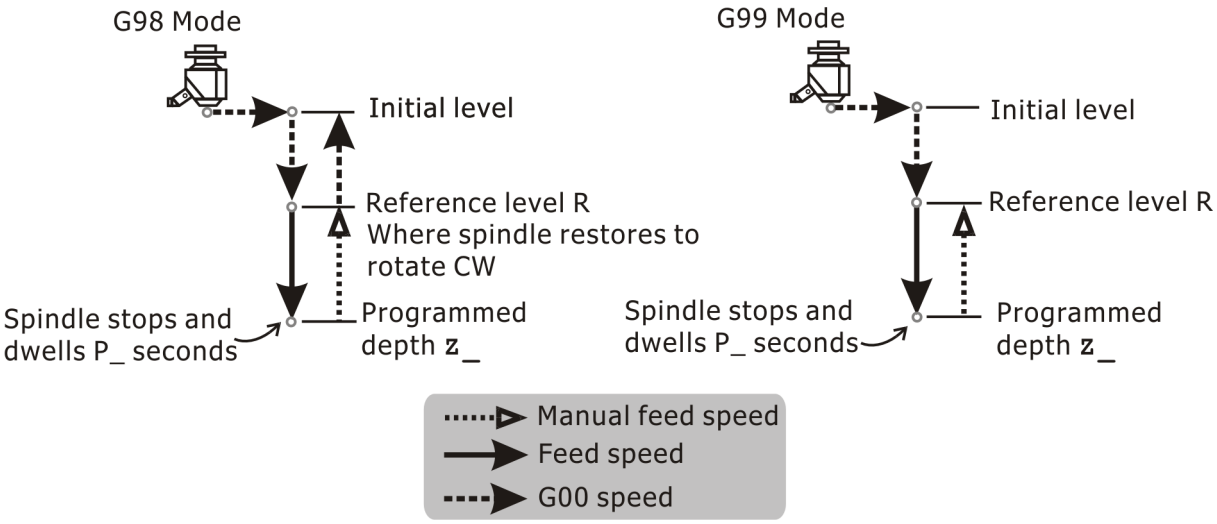


Figure 4-50 G88: boring cycle which allows manual motion from the bottom

4.4.4.4 G89： boring cycle with dwelling on the bottom

Under **G89** mode, when boring to the programmed depth **Z_**, stop the spindle **P_** seconds temporarily. Then, spindle upward to reference level **R_** with feed speed **F_**. If **G89** is programmed by **G98** dimension, tool will continue upward to initial level, but by **G99** dimension tool will not move anymore.

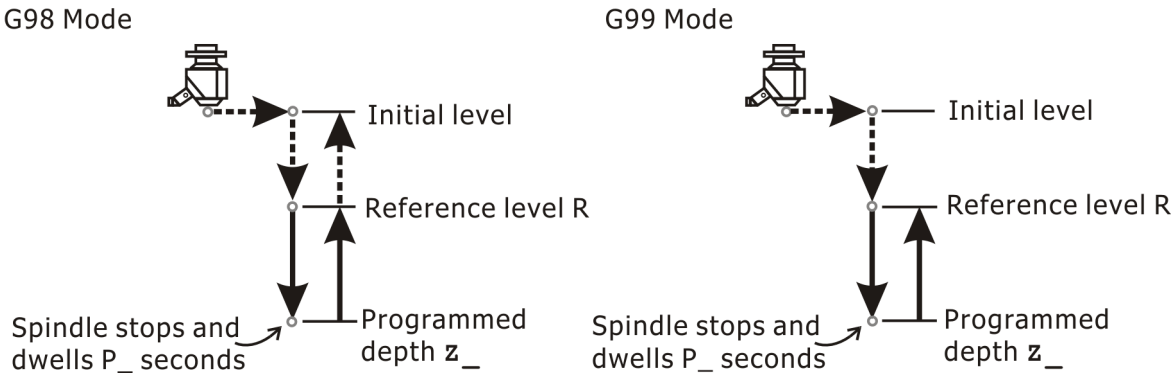


Figure 4-51 G89: boring cycle with dwelling on the bottom

4.4.5 G74/G84：Rigid Tapping Cycle

Format：

G74 X_Y_Z_R_P_K_F_; left handed tapping cycle〔rigid tapping〕
G84 X_Y_Z_R_P_K_F_; right handed tapping cycle〔rigid tapping〕

G74 specify left handed tapping cycle. Must make spindle CCW〔**M04**〕first and then execute **G74**.

The motions of **G74** are as below:

1. Position tool to the specified position **X_,Y_** first.
2. Position tool downward to reference level **R_**.
3. Tapping to the specified position **Z_** on the bottom with feed speed **F_**.
4. Reverse the spindle CW and back to reference level **R_** along the positive direction of Z axis.
5. Restore the spindle CCW at the reference level **R_**.

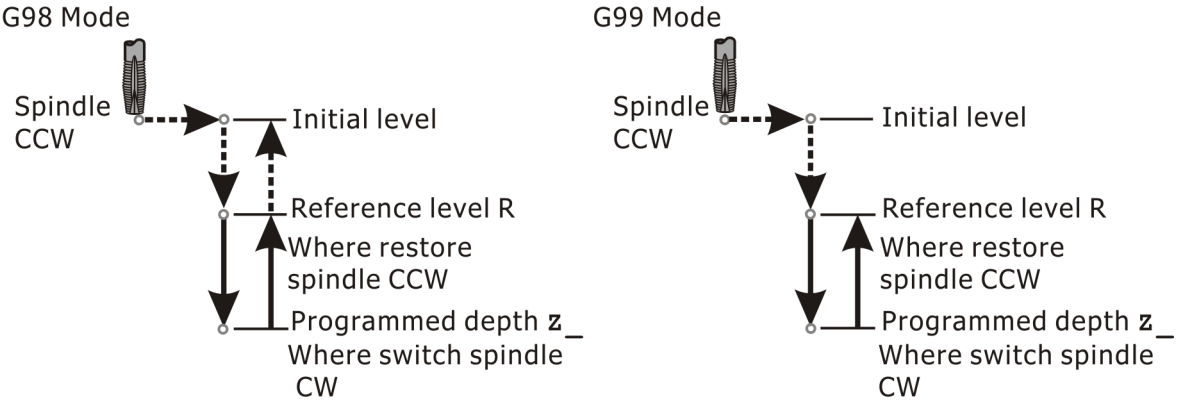


Figure 4-52 G74 left handed tapping cycle〔rigid tapping〕

G84 specify right handed tapping cycle. Must make spindle CW〔**M03**〕first and then execute **G84**.

The motions of **G84** are as below:

1. Position tool to the specified position **X_,Y_** first.

2. Position tool downward to reference level **R_**.
3. Tapping to the specified position **Z_** on the bottom with feed speed **F_**.
4. Reverse the spindle CCW and back to reference level **R_** along the positive direction of Z axis.
5. Restore the spindle CW at the reference level **R_**.

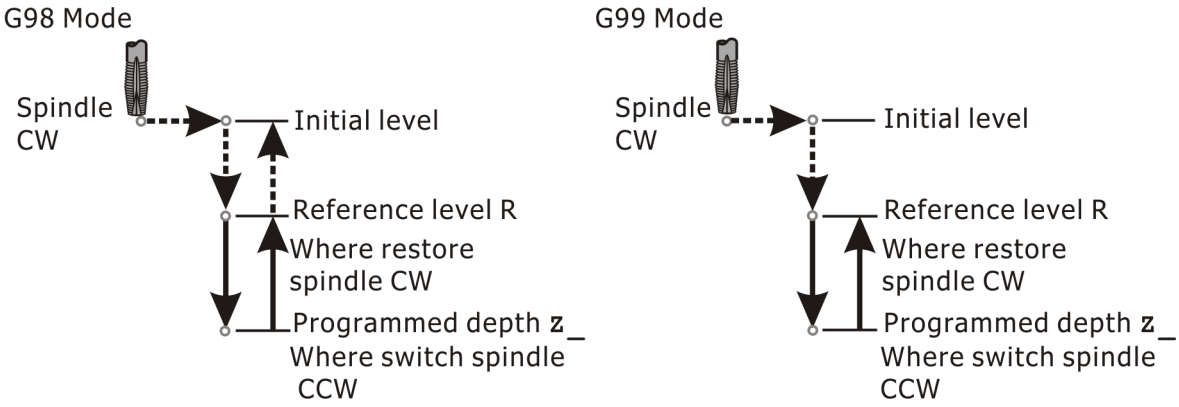


Figure 4-53 G84 right handed tapping cycle〔rigid tapping〕

4.4.5.1 Rigid Tapping

INCON-ME3 controller features rigid tapping if the machine uses variable speed spindle driver and mounts an encoder on the spindle for spindle orientation angle detection. Use specific M code to turn on rigid tapping. It is usually **M29**. Format is **M29 Sxxxx**.

When use AC motor instead of spindle driver, the depth of the tapping will be a little distance over-tapped or short-tapped. The over／short-tapped distance will depend on the speed performance of the AC motor and the machine parameter setting of the spindle related about tapping. Ask your CNC machine maker about this situation.

Format：

```
M29 S_;
M03
G84 X_Z_R_F_;
```

Before performing the rigid tapping, the **M29 S_;** command must be specified previously, in order to set the rotation speed and enable the rigid tapping mode. This rotation speed of tapping is usually in low speed. And this speed will be clamped by the setting of rigid tap maximum speed in machine parameter.

This **M29 S_ ;** command will be kept effective until tapping cycle cancels.

If the spindle has implemented the high / low speed gear change, before doing the rigid tapping, make sure that the gear change should be switched to the dedicated gear for rigid tapping to work with spindle rate ratio. We can get the correct tapping pitch. No gear change needs no change the gear.

The tapping pitch in rigid tapping can be calculated in **G94** or **G95** mode, as follows:

1. The lead of the tap is equal to **F_** code value divided by **S_** code value if system is in **G94** mode.

   Example: If F=100 mm / min, S=200 rpm, and then the pitch =100 / 200=0.5 mm.

   The program can be as follows:

```
G90 G0 X0 Y0 Z0          ; move tool to part center and cutting point in Z direction
G49 G54 G94              ; Select G94 mode
M29 S200                 ; Enable rigid tapping, S=200 rpm
M03                      ; Turn on spindle in CCW rotation
G84 X0 Y0 Z-10. F100     ; Do tapping in 100 / 200=0.5 mm lead, 10 mm depth
G80                      ; Turn off cancycle and rigid tapping
M30
```

2. The lead is equal to **F_** code value if system is in **G95** mode.
   Example: If F=3 mm / rev, then the pitch =F=3 mm, no matter what rpm will be.
   The **F_** value can be less then 1 mm, therefore it is very convenience to use the rigid tapping in **G95** mode.

   The program can be as follows:

```
G90 G0 X0 Y0 Z0        ; move tool to part center and cutting point in Z direction
G49 G54 G95            ; Select G95 mode
M29 S200               ; Enable rigid tapping, S=200 rpm
M03                    ; Turn on spindle in CW rotation
G84 X0 Y0 Z-10. F0.5   ; Do tapping in 0.5 mm lead, 10 mm depth
G80                    ; Turn off cancycle and rigid tapping
M30
```

Where〝**G84 X0 Y0 Z-10. F0.5**〞figures out the 0.5 mm lead directly. And make the program more readable.

4.5 Rotation, Mirrorand Scaling Instructions

4.5.1 G68/G69：Rotation on／off

Format：

```
G68 X_Y_Z_R_;
G69;
```

G68 enables system to rotate coordinate. In the case of absolute dimension〔**G90**〕, the center of rotation is defined by two axes among X, Y, or Z corresponding to which of **G17**, **G18**, or **G19** is specified.

The angle of rotation is defined by **R_**. The sign〝+〞and〝**-**〞in front of **R_** value denotes CCW or CW rotation. For angle, Absolute or Incremental is set by a 〝**Parameter→Rotation angle...→Rotation Angle Abs / Inc**〞.
If selected〝(●) **Incremental**〞, the rotation angle could be progressive until canceled by **G69** or power off to reset the angle to zero.

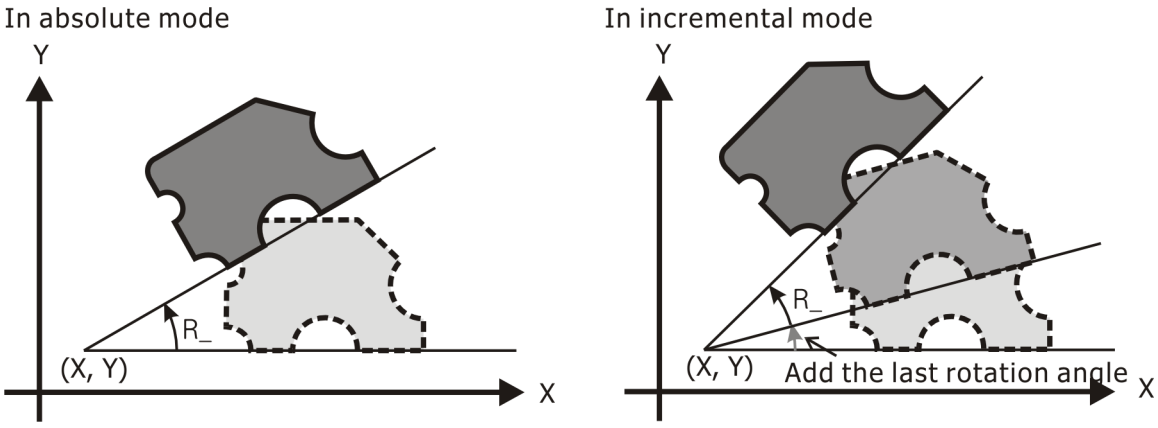


Figure 4-54 Rotation

The relation between **G68** and **G17/G18/G19** is as below:

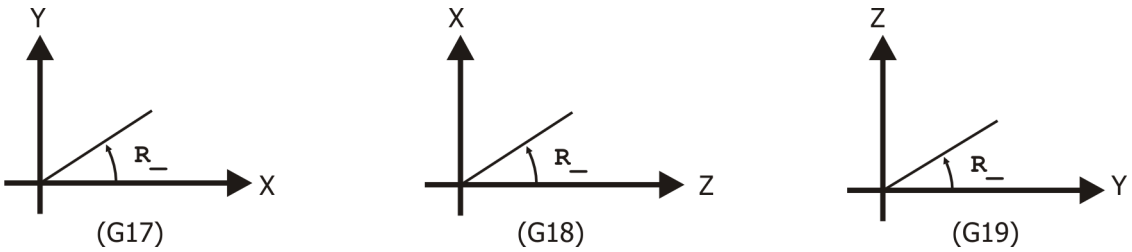


Figure 4-55 The relation between G68 and G17/G18/G19

4.5.2 G50/G51：Scaling or Mirror off / on

Format：

G50;

G51 X_Y_Z_I_J_K_ ; or **G51 X_Y_Z_P_;**

G51 enables system to scale up / down and mirror. The center of scaling or mirror is defined by **X_**, **Y_** and **Z_** in the instruction. If **X_**, **Y_** and **Z_** are all missing from the command line, the current position will be used as the center.

The magnification is defined by **I_**, **J_** and **K_** for each axis separately or by **P_** for all axes. If one or more but not all of **I_**, **J_** and **K_** is defined, the missing component will be defined

by the default parameter.
If **I_**, **J_** , **K_** and **P_** are all missing, the default **P_** parameter will be used.
As for scaling, **I_**, **J_** , **K_** or **P_** has to be greater than zero. For mirror, **I_**, **J_** or **K_** will be negative, which corresponds to mirror image with respect to axis **X_**, **Y_**, or **Z_**.

Note:

1. The 4th axis will be no effected by **G51**. Mirror and scaling function are only effective on X,Y and Z axes.
2. The center point〔**X_Y_Z_**〕of scaling can be absolute or incremental dimension 〔specified by **G90/G91**〕.
3. When the scaling factor of any two among axes are different, the radius of the arc will be specifed by the larger scale and the center position of the arc will renew to calculate by the larger scale.
4. If **G50/G51** and **G68/G69** exist simultaneously in the program, the edit sequence is that **G51** first and then **G68** when enable, but **G69** first and then **G5**0 when cancel. This way must be observed, or the system will pop up a alarm dialog box.

The part contour is in proportional to the same scaling factor of three axes, and it is illustrated as below:

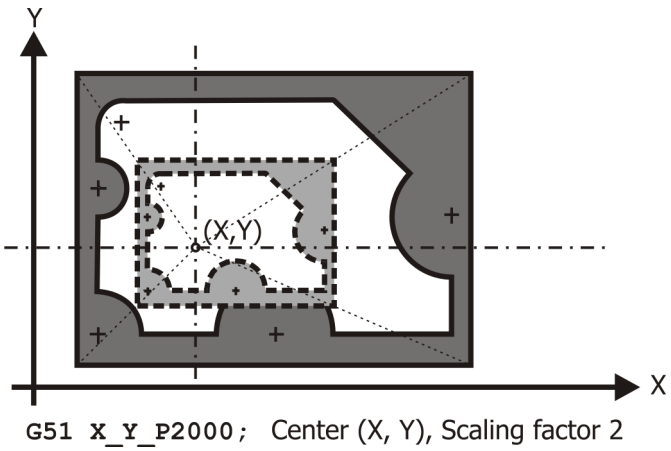
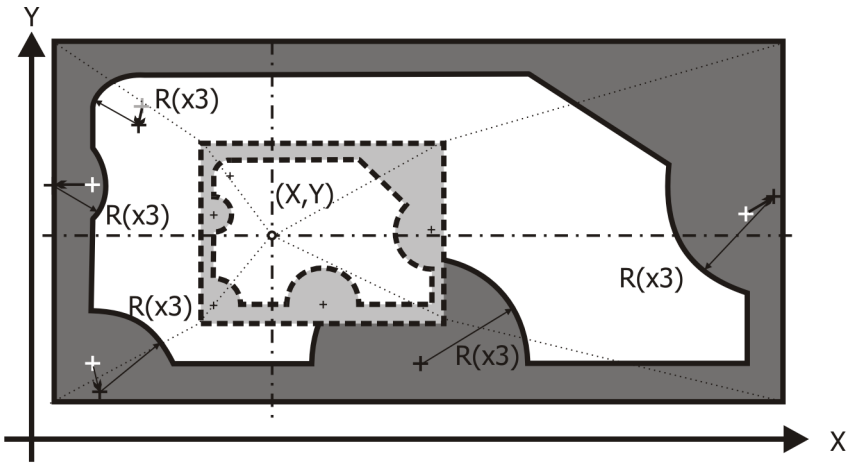


Figure4-56 The part contour is in propotional to the same scaling factor of three axes

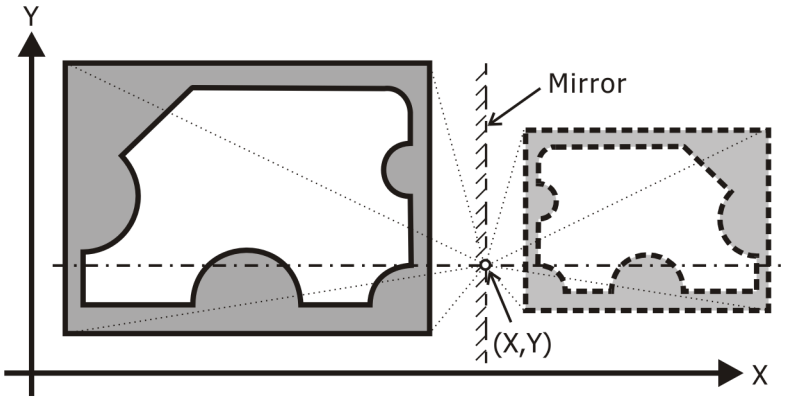
The part contour isn't in propotional to the scaling factor of two axes, and it is illustrated as below:〔Note: The arc center is be shifted, and the radius is enlarged in propotional to the larger scaling factor.〕



G51 X_Y_ I3000 J2000; Center (X, Y), Scaling factor: ×3 for X axis, ×2 for Y axis

Figure 4-57 The part contour isn't in propotional to the scaling factor of two axes

The part contour is in propotional to the same scaling factor of two axes, do mirror image of Y axis,and it is illustrated as below:



G17 G51 X_Y_ I1500 J-1500; Mirror to Y axis in G17 mode (X-Y plane), Scaling factor 1.5

Figure4-58 The part contour is in propotional to the same scaling factor of two axes, do mirror image of Y axis

4.6 Spindle Rotation Instructions

4.6.1 List of M Codes

M codes can be distinguished into two group:

1. Group 1: For Program Execution Control

M code	Format	Function description
M00	**M00;**	program hold
M01	**M01;**	Optional hold
M02	**M02;**	end of program and stay at the end of program
M30	**M30;**	end of program and rewind to the start of program
M95	**M95 P_P_L;**	internal program nested execution〔from 1st **P_** to 2nd **P_**, **L_** times〕
M96	**M96 H_ P_**	conditional jump〔 to **P_** if **H_** is TRUE〕
M97	**M97 P_L_;**	internal subroutine call
M98	**M98 (filename)L_;**	external subprogram call **L_** times
M99	**M99;**	return from subroutine／subprogram／endless loop
M100	**M100**	Program hold for manual feed operation
M101	**M101**	Optional hold for manual feed operation

Note: **M-code for Program Execution Control** must exist alone in a block. No other codes exist simultaneously in the same block.

2. Group 2: Spindle / Tool / Miscellaneous Control

	M code	Format	Function description
	M03	**M03 S_;**	spindle on cw with speed S rpm
	M04	**M04 S_;**	spindle on CCW with speed S rpm
*	**M05**	**M05;**	Spindle off
	M06	**M06 T_;**	Tool change
	M08	**M08;**	Turn on cooling／clean device
*	**M09**	**M09;**	Turn off cooling／clean device
#	**M29**	**M29;**	Rigid Tapping

Note：

1. ***** ：〔Specified by PLC〕Initial M-code in the group when turning power on.
2. Whether **M06/M08/M09** exist or not, it depends on machine tool and these functions are programmed by PLC.
3. Only one **Miscellaneous Control M-code** can exist in a block. However, it can exist with G-codes simultaneously in the same block.
4. **#**： The M code of rigid tapping can be changed in machine parameter. **M29** is the default M code of rigid tapping and being used mostly.

4.6.2 M03/M04/M05：Turn On or Off Spindle

Format：

M03 S_; turn on spindle CW
M04 S_; turn on spindle CCW
M05; turn off spindle

When **M03/M04/M05** exists with **Motion G-code** in the same block, the PLC programmer

usually programs **M03/M04**〔spindle CW／CCW rotation turning on〕 before the block program execution but **M05**〔 spindle stop〕 after finishing the block program execution.

It had better program **M03/M04/M05** in a block alone, and don't combine with other motion command. So that it can guarantee the spindle speed being in steady condition when perform the cutting and the spindle stop after finished the cutting.

4.6.3 S code：Spindle Speed Setting

Format：

Sxxxxxx; Change the spindle speed by **S_** code in rpm

S_ shows usually by 4 digits to denote the value of rpm of spindle, and it shows up to 6 digits. **S_** just sets how fast spindle rotates but doesn't enable spindle to rotate. As soon as the **M03** or **M04** exists the spindle rotates.

4.6.4 G96/G97：Constant Surface Speed Control on / off

Format：

G96 S_;
G97 S_;

G96: turns on constant surface speed control

The spindle speed will be calculated so that the surface speed will be unchanged no matter how tool position changes.

G96 enables the constant surface speed control, and a cylindrical work piece which is clamped on the spindle, is under control too, and the tangent speed of tool nose relative to the surface of the cylindrical work piece will be controlled in **S_** value. This **S_** value can be up to five digits and in the unit of m / min or inch / min.

On performing the constant surface speed control, it is usually assume the Z axis as the rotation axis of the work piece, and the X axis being the radius axis. The X axis zero point of program position has to be at the center of work piece to get surface feed speed. The rotation speed of the spindle will be increase if the radius〔X value〕decreases.

G97: turns off constant surface speed control

The spindle speed is a constant. **G97** cancel the constant surface speed control, and specified the **S_** value restore to revolutions per minute.

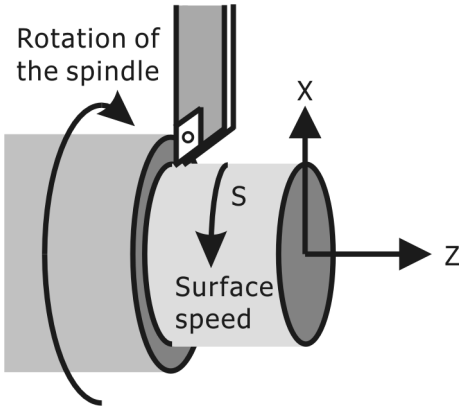


Figure 4-59 Constant Surface Speed Control

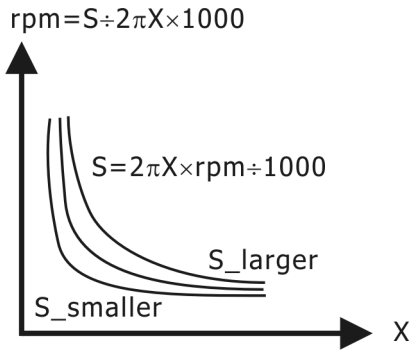


Figure 4-60 The relative figure about spindle speed and X value

4.6.5 G94/G95：Feed per Minute / Revolution

Format：

```
G94 F_;
G95 F_;
```

G94: specifies feed per minute, the **F_** value commanded represents feed per minute.

G95: specifies feed per revolution, the **F_** value commanded represents feed per spindle revolution.

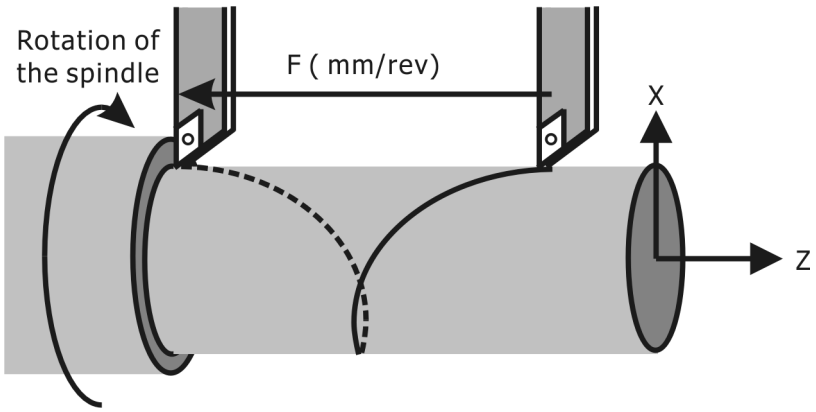


Figure 4-61 feed per revolution

Note:

The digit definition of **F_** value in **G94** and **G95** are different:

1. In **G94** mode, the integer value of **F_** data will be the same as that of specify a decimal point in behind .**F300** will be the same as **F300.** or **F300.0**. And the dot in **F300.05** or **F300.6** will still effective.
2. In **G95** mode, the integer value of **F_** data will be converted to be in the least unit of that system. Therefore **F300** will be converted to **F0.300** in **G99** mode. And the dot in **F300.05** or **F300.6** will still effective.

The ranges of **F_** data in metric / inch system:

Mode	G21〔Metric system〕	G20〔Inch system〕
G94	F0.001～F60000 〔0.001～60000 mm / min〕	F0.0001～F2362 〔0.0001～2362 inch / min〕
G95	F0.001～F999.999 〔0.001～999.999 mm / rev〕	F0.0001～F39.3700 〔0.0001～39.3700 mm / rev〕

4.7 Unit & Coordinates Instructions

4.7.1 G17/G18/G19：Plane Selection

Format：

G17;XY plane selection
G18;ZX plane selection
G19;YZ plane selection

Some G codes only work on the specified plane. For example, chamfering/corner rounding of **G01**, **G02/G03** circular interpolation, **G16**, **G41/G42**, **G68** and so on. We must give plane selection G code〔**G17/G18/G19**〕in front of these G codes. The forth axis does not work at these G codes. The initial code is **G17** in this G code class.

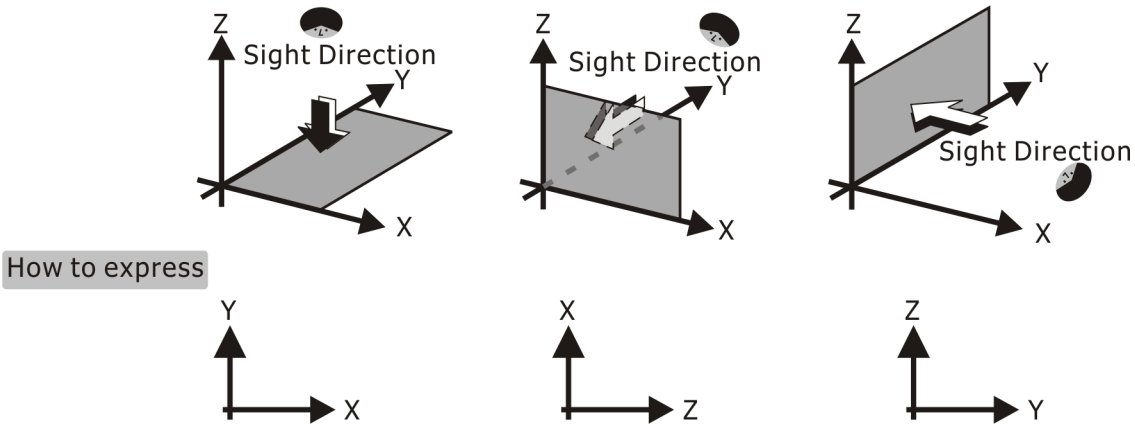


Figure 4-62 G17, G18 and G19 plane selection

4.7.2 G20/G21：Input in Inch / Metric

The value of each address in the programming block can be programmed in Metric or Inch. The default unit when power on depends on the last turning off setting. **G20** or **G21** must be programmed in the first block of the CNC program. As soon as the instruction **G20** or **G21** is specified the mode can't change anymore. We can also switch **G20** or **G21** by 〝**Parameter→Attribute**〞 or 〝**Run→MDI**〞.

The acceptable range in Metric / Inch mode:

Inch mode〔**G20**〕: 0.0001~999.9999 inch
Metric mode〔**G21**〕:0.001~9999.999 mm

4.7.3 G90/G91：Absolute / Incremental Dimension

Format：

G90; Programming in absolute dimension
G91; Programming in incremental dimension

Must input end position **X_**,**Y_** and **Z_** which can be programmed in absolute or incremental dimension in **Motion G Code**s. This is specified by **G90** or **G91**.

When move the tool from **A(150,300,120)** to **B(100,240,90)**, 〝**G90 X100 Y240 Z90;**〞 denotes program in absolute dimension〔**G90**〕 illustrated as the following figure.

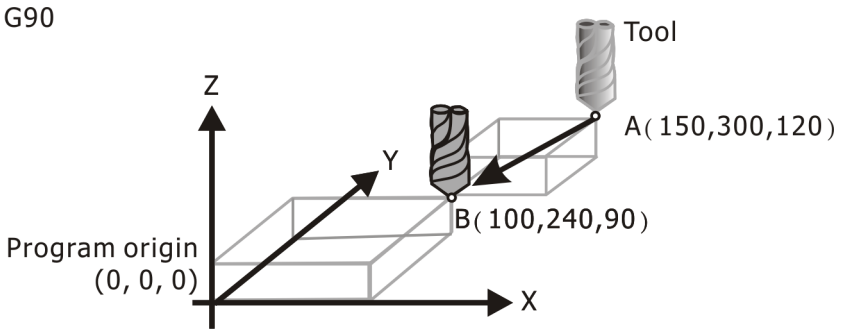


Figure 4-63 G90; Programming in absolute dimension

〝**G90 X-50 Y-60 Z-30;**〞 denotes program in incremental dimension〔**G91**〕 illustrated as the following figure.

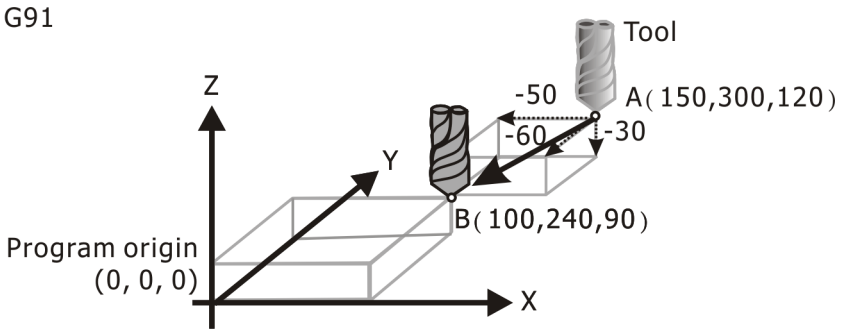


Figure 4-64 G91; Programming in incremental dimension

4.7.4 G15/G16：Cartesian / Polar Coordinate System

Format：

G15; Programming in Cartesian coordinate
G16; Programming in polar coordinate

Must input end position **X_**, **Y_** and **Z_** which can be programmed in Cartesian or polar coordinate in **Motion G Code**s. This is specified by **G15** or **G16**. **G15** is the default code of this class. When program in polar coordinate, must specify **G16** previously.

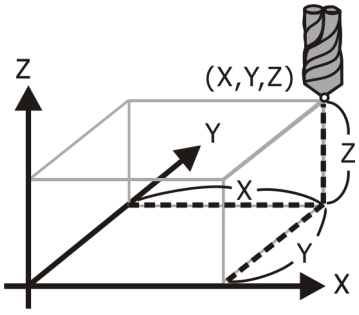


Figure 4-65 G15: Programming in Cartesian coordinate

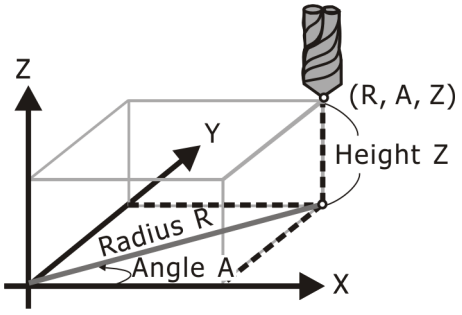


Figure 4-66 G16: Programming in polar coordinate

Note: polar coordinate is closely related to plane selection. The plane selection 〔**G17/G18/G19**〕has the priority to set up.

Polar coordinate use **X_**, **Y_** and **Z_** to denote its address code. The meanings of **X_**, **Y_** and **Z_** are listed below：

	X	Y	Z
G17	radius	angle	height
G18	angle	height	radius
G19	height	radius	angle

The definition of positive angle is the angle of rotating counter clockwise from the 1st axis on the specified plane, and list as below:

G17: rotate from X axis to Y axis.
G18: rotate from Z axis to X axis.
G19: rotate from Y axis to Z axis.

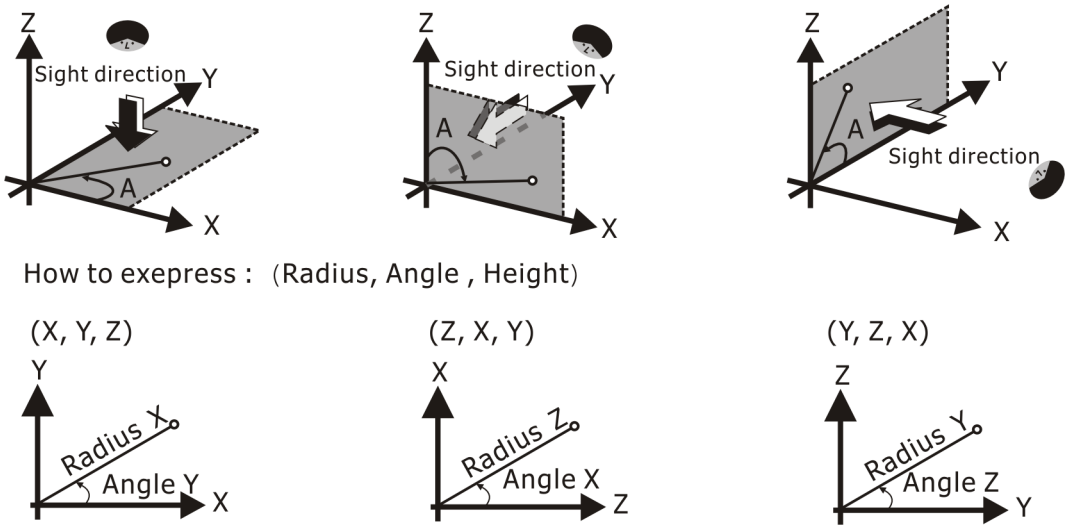


Figure 4-67 The definition of positive angle in polar coordinate

G90 and **G91** are still effective in polar coordinate,such as radius and angle in absolute or increment dimension. When initially switching coordinate to the polar coordinate, the definitions of radius and angle in **G90/G91** mode are varied conditions as follows:

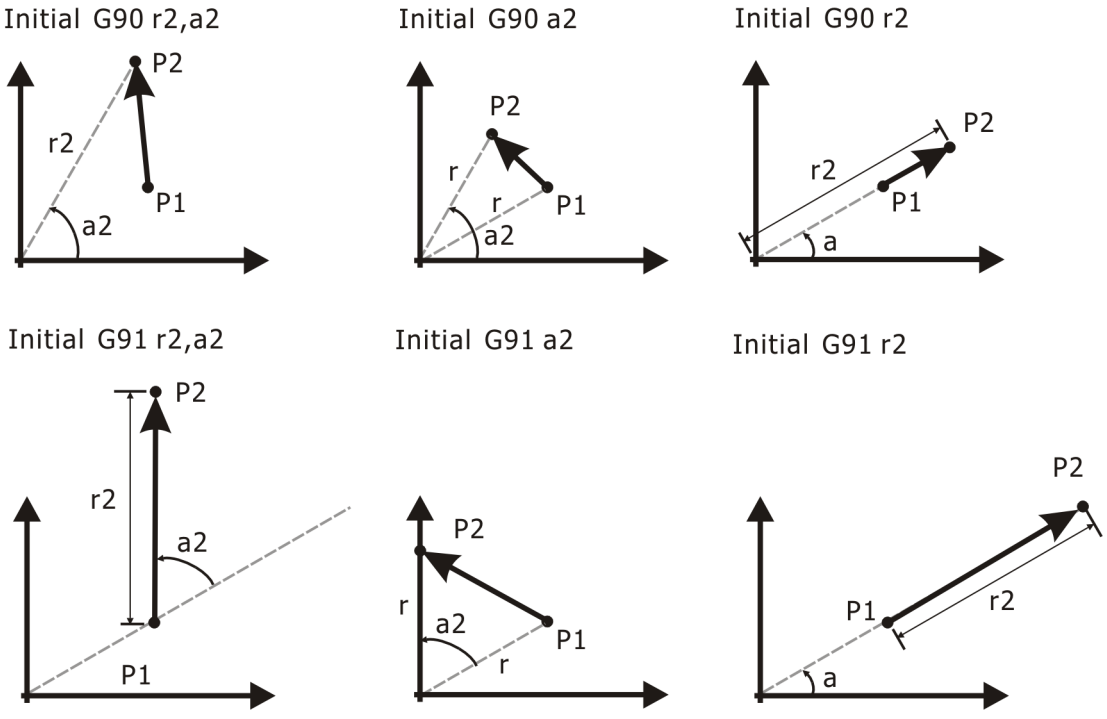


Figure 4-68 When initially switching coordinate to the polar coordinate, the definitions of radius and angle in G90/G91 mode are varied conditions

Note: When the angle is programmed in incremental dimension, the initial angle is defined as the absolute angle of current position P1. Then, add the incremental angles to the initial angle. If the current point is located at the zero point〔origin〕, then the initial angle is zero.

The radius and angle of polar coordinate are specified in absolute or incremental dimension by **G90/G91**.The end position are different because of the sequence of radius and angle setting by **G90** or **G91**. The possible conditions are illustrated as below:

1. When the radius is specified in absolute dimension first

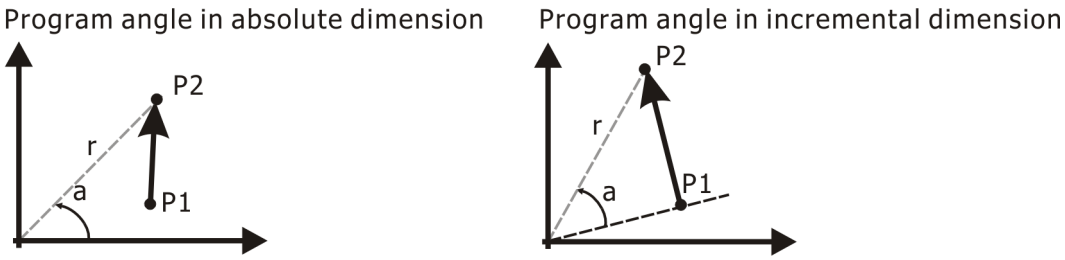


Figure 4-69 When the radius is specified in absolute dimension first

2. When the radius is specified in incremental dimension first

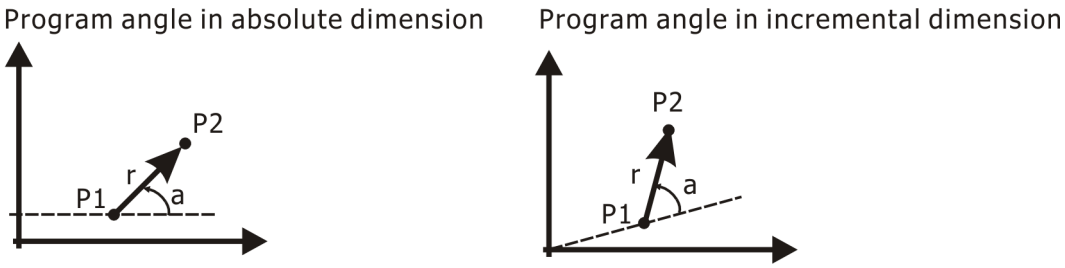


Figure 4-70 When the radius is specified in incremental dimension first

The varied conditions according to the sequence of radius or angle in absolute or incremental dimension are illustrated as below:

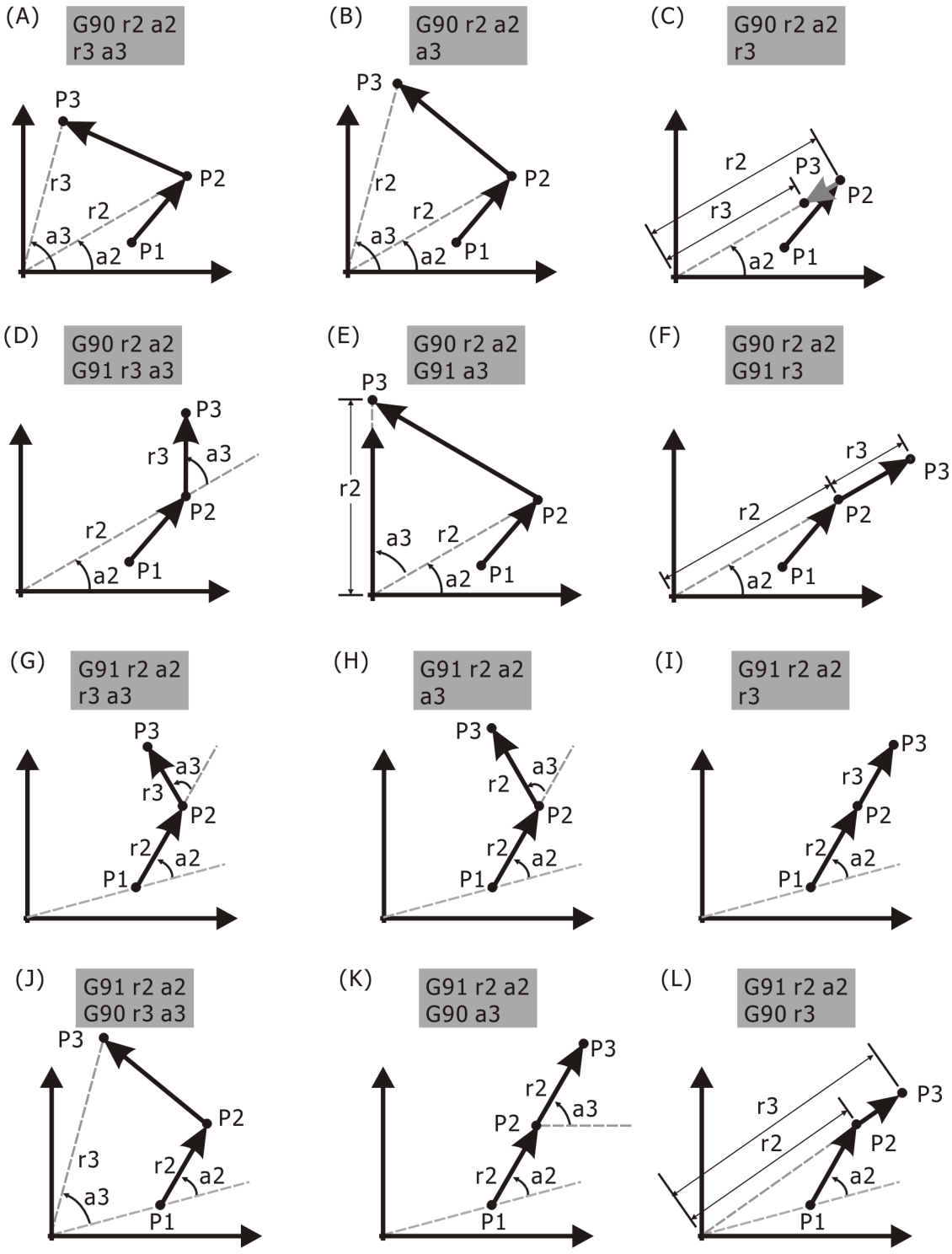


Figure 4-71 The varied conditions according to the sequence of radius or angle in absolute or incremental dimension

Note: If the current position P1 were located at zero point origin, **r1**=**a1**=0.

4.7.5 G54～G59：Work Coordinate System 1~6 Selection

Format：

G54
G55
G56
G57
G58
G59

Up to 6 work coordinate systems can be selected by **G54** to **G59**. The offsets of these 6 work coordinate systems are set in the parameter which will add on to the offset created by **G92** to form the total offset.

The default work coordinate system when power turns on is the coordinate system defined by **G54**.

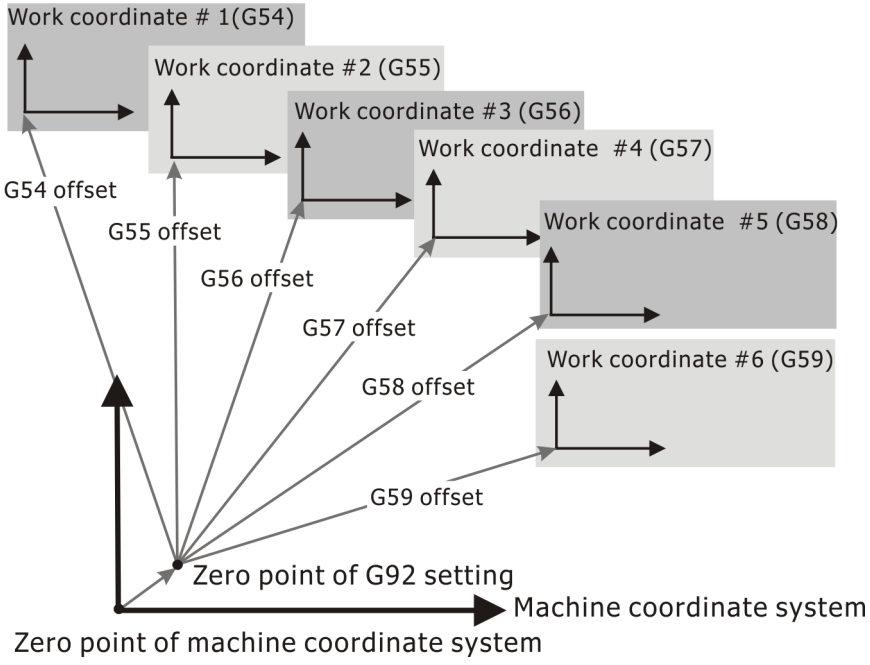


Figure 4-72 Work Coordinate System 1~6 Selection

4.7.6 G52：Set Local〔Child〕Coordinate Offset

Format：

G52 X_ Y_ Z_ A_;

G52 sets the offset of a local〔child〕coordinate system relative to the origin of the work coordinate system.

The offset will be commanded by **X_Z_**. Cancel the local coordinate by **G52 X0 Z0**.

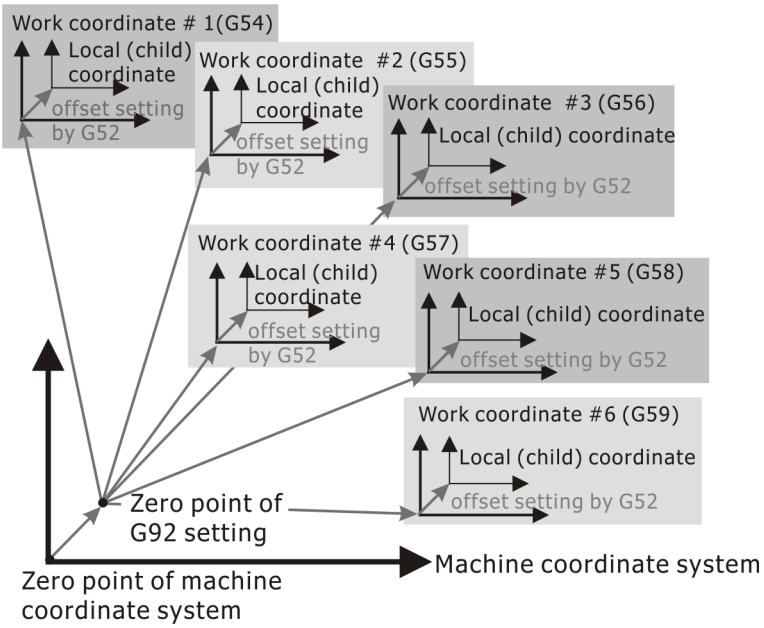


Figure 4-73 G52: Set Local［Child］Coordinate Offset

4.7.7 G92：Set Program Position/ G92 External Offset

Format：

G92 X_ Y_ Z_ A_; Set Program Position

G92; **G92** External Offset

G92 sets the program position of the current machine position as the position commanded by **X_,Y_,Z_**. Therefore , sets an external offset of the program position from the current machine position, which then sets the origin of work coordinate system.

Note: When tool radius or length compensation is on, **G92** can not be applied.

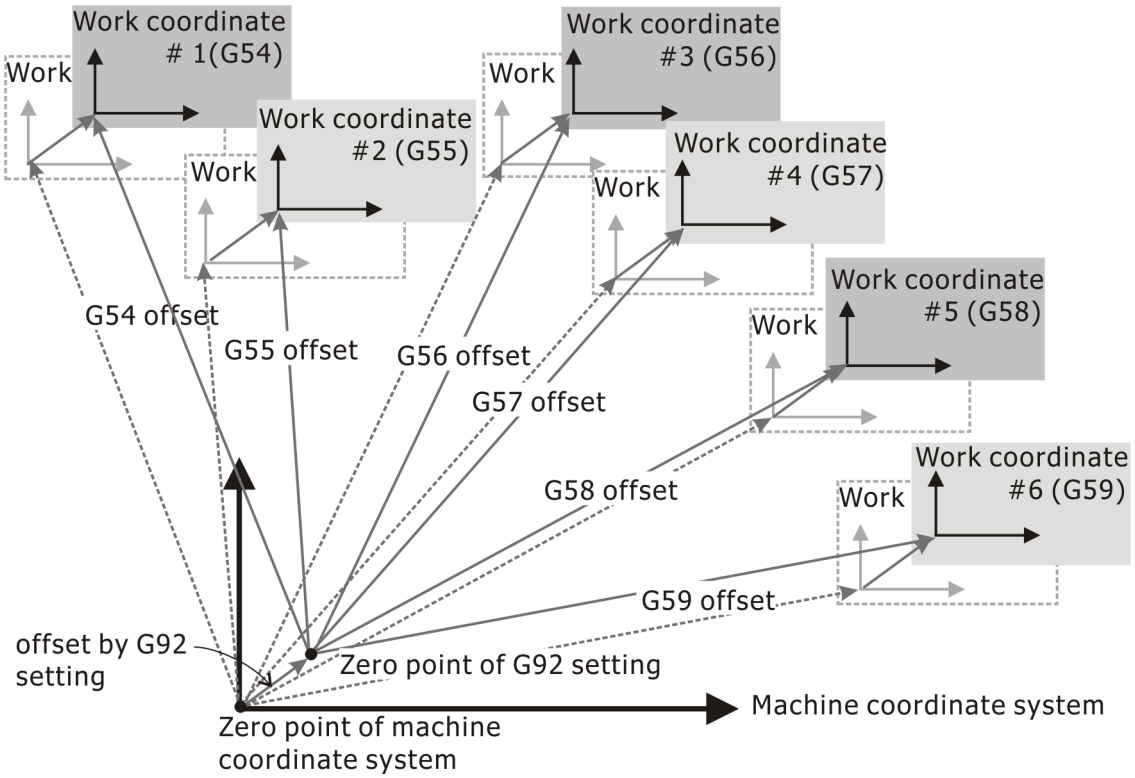


Figure 4-74 G92 : Set Program Position/ G92 External Offset

4.8 Program Executing Instructions

4.8.1 G04：Dwell Time

Format：

```
G04 X(U)_;
G04 P_;
```

G04 is specified to pause program in **X_**, **U_** or **P_** seconds.

The **G04** instruction can be applied to the following conditions:

1. When spindle shifts gear, execute **G04** to pause some seconds upon finishing M05. Make sure spindle stop and then shift gear to protect servo of spindle from damage.
2. When cutting on the bottom of hole, execute **G04** to pause some seconds. That can improve the precision of the depth and the surface on the bottom, such as beam hole, taper hole, spot facing and so on.
3. When tapping thread with large radius, execute **G04** to pause some seconds. Make sure spindle speed steady and then tap thread can make the screw pitch correct.

Dwell time is from 0.001 second to 9999.999 second in metric system. Dwell time is from 0.0001 second to 999.9999 second in imperial system.

If no X or P is commanded, **G04** will act exact like the instruction **G09** for exact stop mode.

4.8.2 G09：One Shot Exact Stop

This is a **one shot command**, it only effects in the line of G09 commanded.

The **G09** commands tool to be located at the specified position and execute in-position check. **G09** is just effective when it is programmed with the **Motion G Code**s. **G09** makes the **Motion G code** of specified axis confirm its end position and cut desired acute angle.

Both **G09** and **G61** can execute **Exact Stop**〔**In Position Check**〕. The difference between **G09** and **G61** is that **G09** is **One Shot Command** but **G61** is sustained command.

4.8.3 G60/G61：Cancel / G61 Exact Stop Mode

Format：

G60;Cancel **G61** Exact Stop Mode〔In Position Check Off〕
G61;Exact Stop Mode On〔In Position Check On〕

G61 turns on the exact stop mode and the following cutting commands, such as **G01**, **G02**, **G03**, **G33** and so on, will perform in-position check until G60 cancel it.

The CNC programs designed by CAD will cut many line segments. If **G60** canceled the exact stop mode, these line segments get a smooth contour by cutting continuously.

Note: Some instructions related to positioning, such as **G00**, **G53**, **G28** and **G29**, have executed **Exact Stop Mode** automatically upon reaching end position. Therefore **G61/G60** is not effective to these G codes.

4.8.4 M00/M01/M100/M101/M02/M30： Program Hold／Stop

Format：

M00;	Program Hold
M01;	Optional hold
M100;	Program hold for manual feed operation
M101;	Optional hold for manual feed operation
M02;	End of Program
M30;	End of Program Rewind

M00：Program Hold

Program pause immediately. Press F9 to resume program execution.

M01：Optional hold

The function of **M01** is the same as one of **M00**. **M01** is just effective when it highlights in the system state window. When **M01** doesn't highlight, **M01** will be skip.〔 In other words, the program will not stop because of **M01**.〕

M100/M101：Program hold / Optional hold for manual feed operation

M100/M101 is similar to **M00/M01**, But **M100/M101** enables feed toward or backward operation after program hold. That can check correction of cutting path. Press **Cycle Start** button or **F9** to restore program execution.

M02：End of Program

M02 is terminate program execution, and stay at the end of the program. **M02** makes spindle stop 〔**M05**〕, cutting oil turn off〔**M09**〕,and the cursor stay at the **M02** block instead of the first block in a program. Pogrom doesn't execute anymore even press **Cycle Start** button or F9. Must reload the program to restore execution.

M30：End of Program Rewind

M03 is terminate program execution, and rewind to the start of the program. **M30** makes spindle stop 〔**M05**〕, cutting oil turn off〔**M09**〕,and the cursor return automatically to the first block in a program. That is convenient to execute program again.

Note: Press Cycle Start button of F9 depends on machine parameter. Please ask your machine maker for service.

4.8.5 M95/M97/M98/M99：Inner program / subroutine / subprogram call or return

Format：

M95 P_P_L; Internal Program Nested Execution〔from 1st **P_** to 2nd **P_**, **L_** times〕
M97 P_L_; Subroutine Call **L_** times
M98 (filename)L_; Subprogram Call **L_** times
M99; Return from Subroutine／Subprogram

Note:

1. If using **M99** instead of **M02/M30** at the end of program, then the program will be repeated execution.
2. All these subprogram or subroutine call can be used in nesting. The maximum nest layer

number of these program or subroutine call can be up to eight layers.

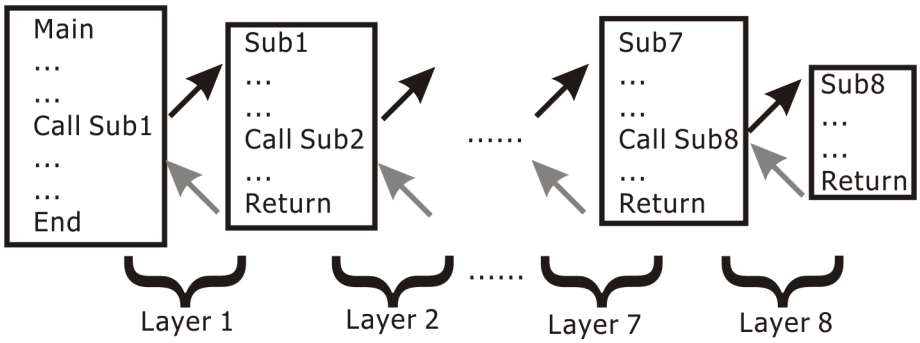


Figure 4-75 The maximum nest layer number of these program or subroutine call can be up to eight layers.

4.8.5.1 M95：Internal Program Nested Execution

Format：

M95 P_P_L;

M95 executes a nested program run inside the current program from a commanded start line to a commanded end line which both have to be in front of **M95** command.
In command line, start and end line will each be defined with a **P_** code. Also, an **L_** code defines the repetition count.

In program, the specified start and end line will be defined with **N_** codes.

SAMPLE95.CNC	Description
;SAMPLE OF M95 CALL	
N10 M_;	
N20 G_Y_;	Start of nest 1〔**N20**〕
N30 X_;	Start of nest 2〔**N30**〕
N40 G_X_;	
N50 Y_;	
N60 Z_;	End of nest 2〔**N60**〕

N70 X_;	
N80 M95 P30 P60 L5;	Execute from **N30** to **N60** five times
N90 G_X_Y_;	End of nest 1〔**N90**〕
M95 P20;	Execute from **N20** to **N90** one times
M02;	Program end

4.8.5.2 M97：Internal Subroutine Call L_ times

Format：

M97 P_ L_

M97 calls a subroutine **L_** times. The subroutine and the main program must be in the same file, and the subroutine follows the main program.

The subroutine is named〝**Oxxxx**〞. 〝**xxxx**〞denotes the value of **P_** in **M97** block by four digits. The subroutine must end with **M99**〔subroutine return〕

Example1:

SAMPLE97.CNC	**Description**
;SAMPLE OF M97 CALL	Main program
G50 X0 Y0 Z0;	
M03;	
M97 P1005 L3;	Call subroutine **O1005** three times
M97 P2001;	Call subroutine **O2001**
M05;	
...	
M02;	End of main program
O1005	Start of subroutine **O1005**
...	
M99;	End of subroutine **O1005**
;	

O2001;	Start of subroutine **O2001**
...	
M99;	End of subroutine **O2001**

Example 2:

SAMPLE97.CNC	**Description**
;SAMPLE OF M97 CALL	Main program
G50 X0 Y0 Z0;	
...	
M97 P1005 L3;	Call subroutine **O1005** three times
...	
M30;	End of main program
;	
O1005	Start of subroutine **O1005**
...	
M97 P2001;	Call subroutine **O2001**
...	
M99;	End of subroutine **O1005**
;	
O2001;	Start of subroutine **O2001**
...	
M99;	End of subroutine **O2001**

4.8.5.3 M98： External Subprogram Call L_ times

Format：

M98 (FILENAME) L_

M98 calls an external subprogram which has existed in the file list. The subprogram must end with **M99**〔subprogram return〕. A repetition count can also be commanded with an **L_**

code.

Note: the format of **M98** is 〝**M98 Program name L___**〞, and doesn't need any leading alphabet before the program name, but it must need a space between the program name and the **L_** character.

Example:

SAMPLE98.CNC	**Description**
;SAMPLE OF M98 CALL	
G92 X0 Y0 Z0;	
G91 G00 X100 Y100;	
M03;	
M98 SUB1OF98 L3;	Call external subprogram **SUN1OF98** three times
M98 SUB2OF98;	Call external subprogram **SUN2OF98**
M05;	
...	
M02;	

SUB1OF98.CNC	Description
;SUB_PROGRAM 1 G01 Z-10; X50 Y50; G02 X50 Y-50 R50; G01 X-100; G00 Y-200; M99;	End of external subprogram **SUB1OF98.CNC**, and return to **SAMPLE98.CNC**

SUB2OF98.CNC	Description
;SUB_PROGRAM 2 ... M98 SUB3OF98; ... M99;	Call external subprogram **SUB3OF98** End of external subprogram **SUB1OF98.CNC**，and return to **SAMPLE98.CNC**

SUB3OF98.CNC	Description
;SUB_PROGRAM 3 ... ... M99;	End of external subprogram **SUB1OF98.CNC**, and return to **SAMPLE98.CNC**

4.8.5.4 M99： Subprogram Return

M99 returns the program execution to the calling program. All subroutines／subprograms must have **M99** as its very last code. However, if **M99** is not within a subroutine or subprogram, the program will do endless loop.

4.9 Reference Position Instructions

Reference point can be used as the position of tool changing, or the temporary staying position for work piece changing. Up to four reference points can be used, and the offset value of each reference point relative to the zero point of machine coordinate can be set in the 〝**Parameter→Reference positions…**〞.

The machine zero point is the index position of all reference points, therefore the 〝**Home**〞 operation must be done before using the reference position command. The machine parameter can set whether Homing first and then using machine. It usually selects the item 〝**Homing first**〞.

Note: When machine has special needs and not to select the item 〝**Homing first**〞, the tool position upon turning on is regarded as the origin of machine position and the reference positions change their positions with it. Under this situation, executing Reference Position is very possible to crash work piece to damage work piece or tool.

4.9.1 G28：Return to Primary Reference Position through Intermediate Point

Format：

```
G28 X_ Y_ Z_ A_;
```

X_ Y_ Z_ A_ is the position of intermediate point.

G28 commands tool to return to primary reference position with **G00** speed. **G28** can indicate a safety path, move tool to return to primary reference position along this safety path , and then execute the instruction about changing tool. The intermediate **(X,Y,Z,A)** point can be

programmed in absolute or incremental dimension. If just gave **G28** without specified axis, **G28** would not be effective.

The primary reference position is set by "**Parameter→Reference positions**". Input value is the vector from machine origin to reference position.

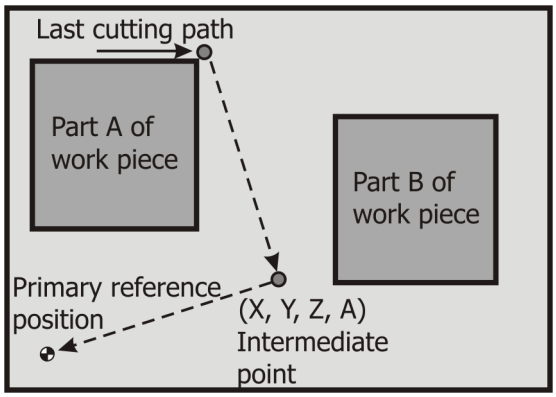


Figure 4-76 G28：Return to Primary Reference Position through Intermediate Point

4.9.2 G29：Move to End Point from Primary reference position through Intermediate Point

Format：

```
G29 X_ Y_ Z_ A_;
```

X_ Y_ Z_ A_ denotes the position which tool reaches to in the end.

G29 commands tool to reach end point from primary reference position through the intermediate point specified by **G28**. **G29** can indicate a safety path, move tool to end position along this safety path. Must give **G28** instruction before **G29**, or **G29** doesn't know where the intermediate point is and system alarms.

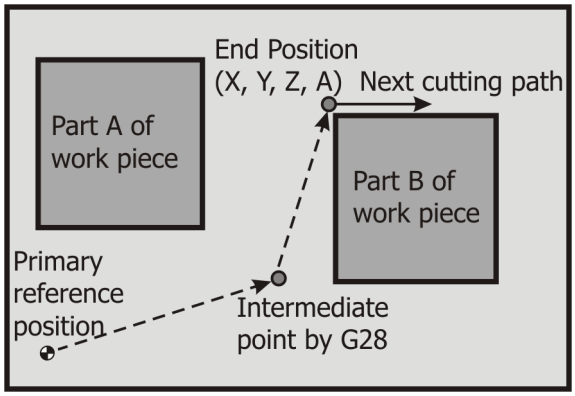


Figure 4-77 G29：Move to End Point from Primary reference position through Intermediate Point

4.9.3 G30：Return to the 2nd, 3rd or 4th Position through Intermediate Point

Format：

```
G30 X_ Y_ Z_ A_ P_;
```

P_ means the 2nd, 3rd or 4th reference point selected. **X_ Y_ Z_ A_** means intermediate position.

G30 commands tool return to selected reference point through the intermediate point. G30 is similar to **G28**. The difference between **G30** and **G28** is **G28** returns to primary reference point, but **G30** returns to the 2nd, 3rd or 4th reference point selected.

The 2nd, 3rd or 4th reference position is set by 〝**Parameter→Reference positions**〞. Input value is the vector from machine origin to each reference position.

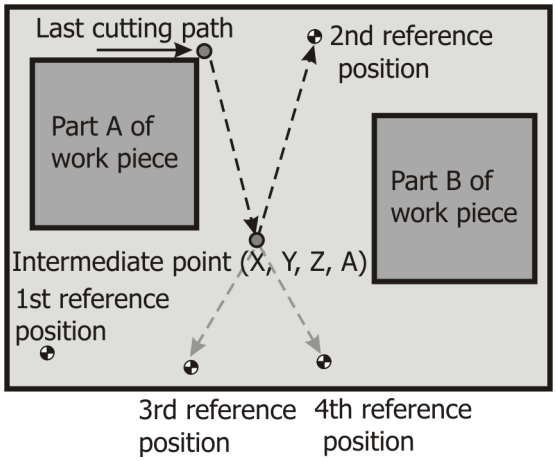


Figure 4-78 G30：Return to the 2nd, 3rd or 4th Position through Intermediate Point

Note：

1. Under **G28** or **G30** mode, after tool moves to primary, 2nd, 3rd or 4th reference point, the work coordinate system is changed to another one, and then the intermediate point is also changed with new coordinate system. When **G29** follows **G28** or **G30** later, the tool will reach to end position through the intermediate point of new work coordinate system.
2. Cancel the length or radius compensation instructions 〔**G49**、**G40**〕before **G28** or **G30**.

4.10 Miscellaneous Instruction

4.10.1 M08/M09：cooling or clean device turn on / off

Format：

M08 ; cooling / clean device turn on

M09 ; cooling / clean device turn off

Note： M codes function are defined by PLC.

4.10.2 T# : Tool Selection

Format：

Txx;
Txxxx;

T code is also named tool function which consists of address **T** and 2 digits following **T**.

How to change tool depends on tool changer machine. Some of tool changers are disk model and some are chain model. There are two methods of tool changers, no robot arm and with robot arm. The tool changer without robot arm is move tool storehouse to approach spindle first, unload the original tool on spindle, switch the other tool desired, and then load it. Most of the tool storehouse are disk model and fixed tool number〔It denotes No.1 tool must return to No.1 tool storehouse〕. The tool change in program edits as below:

```
M06 T03;
```

M06 is a tool change instruction which unload the original tool in spindle, switch it to No. 3 tool, and load No.3 tool in spindle.

Most of the tool changer with robot arm combine with chain tool storehouse and have no fixed tool number〔these words denote No.1 tool may not return to No.1 tool storehouse and tool number setting is set by PLC〕. The tool changer of this model means call tool by the digits following **T**. When **T** code enables, the called tool will switch itself to tool change position but no motion of tool change actually. Therefore T code must be specified before M06 to save the waiting time of tool change.

4.10.3 M06：Tool Setting

Format：

M06;

M06 commands ATC〔automatic tool changer〕to change tool.

It is not all available to change tool anywhere. Tool change depends on the design of machine maker. Most tool changes at a safety location where to avoid crashing of machine and work piece. The machine origin of Z axis is usually the safety position furthest work piece. Therefore execute **M06** normally after tool returns to machine origin of Z axis.

Note: Make sure ATC has been installed before executing **M06**. Please ask your machine maker for detail.

4.10.4 G10/G11：Data Setting Mode On/Off

Format：

G10 (P_R_L_);
G11;

User can use **G10** to switch on the data setting mode of length and radius compensation in CNC program. This command is modal, i.e., **G10** will be effective until canceled by **G11**. **G11** is **NECESSARY** to cancel the data setting mode, else alarm will be issued when other codes are commanded.

In data setting command line, the only commands allowed are **G10/G11** and **G90/G91**. The data is absolute or incremental depends on the current state of **G90/G91**.

For **G10**, the command arguments are:

P_ : tool entry
R_ : tool radius
L_ : tool length

G10 can stand alone in the command line to just switch on the data setting mode for the data setting command lines follows, or, it can has complete arguments with it. However, if arguments follows, **P_** must exist to specify the data entry.

Example:

```
G90 G10 P1 R1. L10.   ; Tool 1, Radius=1., Length=10.
G10 P2 R2. L20.       ; Tool 2, Radius=2., Length=20.
G10 P3 R3. L30.       ; Tool 3, Radius=3., Length=30.
G10 P4 R4. L40.       ; Tool 4, Radius=4., Length=40.
G91 G10 P5 R-0.5      ; Tool 5, Radius=-0.5.
G90 G11               ; Data setting mode off
```

or

```
G90 G10               ; Data setting mode on
P1 R1. L10.           ; Tool 1, Radius=1., Length=10.
P2 R2. L20.           ; Tool 2, Radius=2., Length=20.
P3 R3. L30.           ; Tool 3, Radius=3., Length=30.
P4 R4. L40.           ; Tool 4, Radius=4., Length=40.
G91                   ; To incremental mode
P5 R-0.5              ; Tool 5, Radius=-0.5.
G90 G11               ; Data setting mode off
```

4.11 Examples of programs of M series

The examples of G codes combination list as below: there are more example in 〝**Help→Code index**〞.

4.11.1 The example：G54~G59 Work Coordinate

SAMPL54.CNC：

```
; Set 〝Parameter→Coordinate offsets〞
; G54 X0 Y0 Z0
; G55 X200. Y200. Z0
; G56 X400. Y100. Z0 previously.
;
G92 X0 Y0 Z0;
G90;
/M03;            Spindle turn on
G54 G00 Z0;      Select work1
M98 SUBWORK1;    Call sub.
G55;             Select work2
M98 SUBWORK1;    Call sub.
G56;             Select work3
M98 SUBWORK1;    Call sub.
/M05;            Spindle turn off
G54 G00 X0 Y0 Z0;
M30;
```

SUBWORK1.CNC：

```
;subroutine of cutting a work
G90;
G00 X0 Y0;
/G01 Z-5. F500.;
G01 X100. F1500.;
X90. Y10.;
G03 X10. Y90. I-40. J40.;
G01 X0 Y100.;
Y0;
/Z0;
M99
```

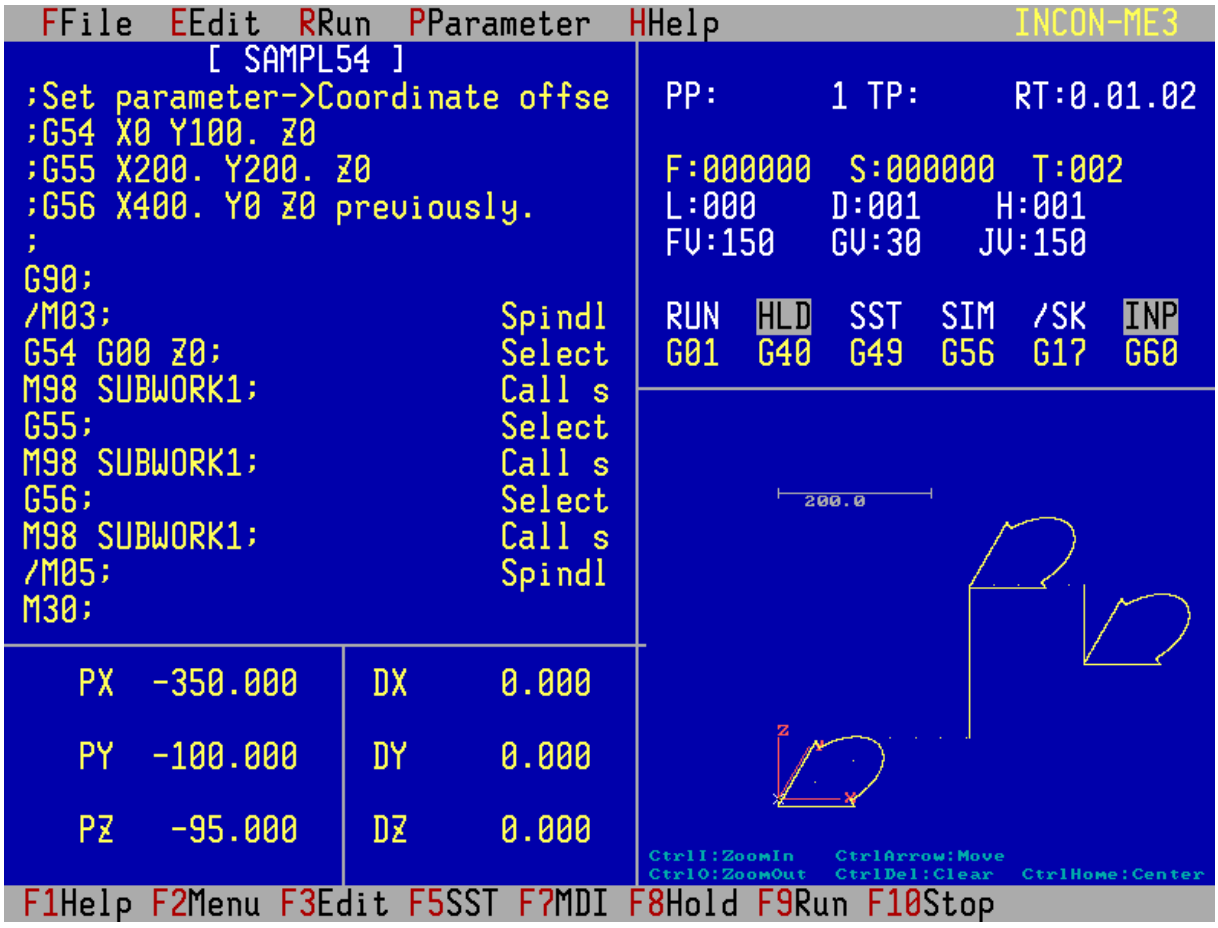


Figure 4-79 An example is illustrated

4.11.2 The example : G52 Child Coordinate

SAMPL52.CNC：

```
; Set 〝Parameter→Coordinate offsets〞
; as SAMPL54 noticed
; and then run this program
; notice the difference of tool path from SAMPL54
;
G92 X0 Y0 Z0;
G90;
/M03;              Spindle turn on
G52 X100. Y50.;    Set child coordinate.
G54 G00 Z0;        Select work1
M98 SUBWORK1;      Call sub.
G55; Select work 2
M98 SUBWORK1;      Call sub.
G56; Select work 3
M98 SUBWORK1;      Call sub.
/M05;              Spindle turn off
G54 G00 X0 Y0 Z0;
M30;
```

SUBWORK1.CNC：

```
;Subroutine of cutting a work
G90;
G00 X0 Y0;
/G01 Z-5. F500.;
G01 X100. F1500.;
X90. Y10.;
G03 X10. Y90. I-40. J40.;
```

```
G01 X0 Y100.;
Y0;
/Z0;
M99
```

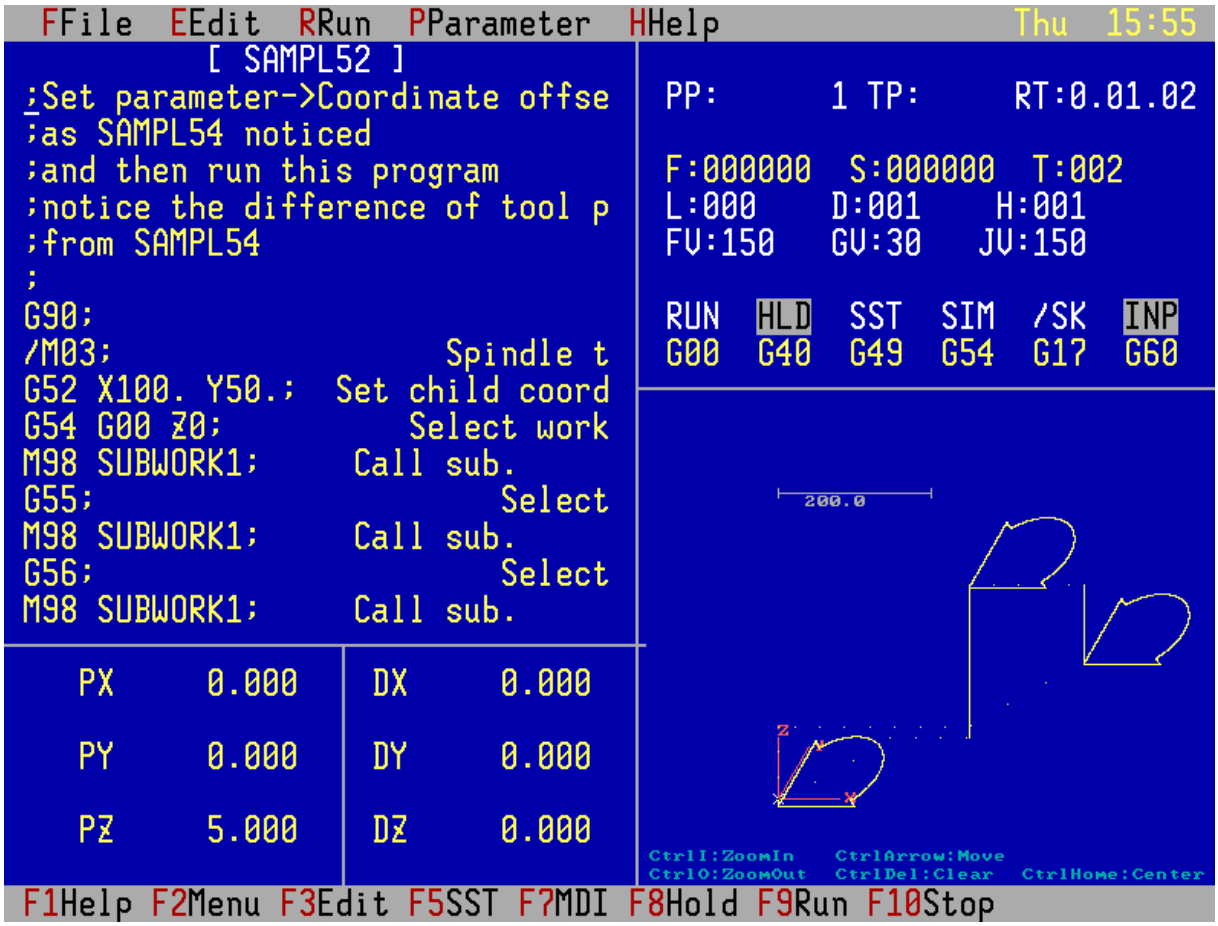


Figure 4-80 An example is illustrated

4.11.3 The example : G43 Tool Length Compensation

SAMPL43.CNC：

```
; Set〝Parameter→Tool table〞:
;T1 length to 10 mm previously.
;Then run the program
G90;
/M03;              Spindle turn on
G00 Z0;
M98 SUBWORK1;      Call sub1.
G43 H1;            Tool length compensation on
M98 SUBWORK1;      Call sub1.
G49 G00 Z10;
/M05;              Spindle turn off
M30;               Program end
;
;Change T1 length to -15 mm, and then run the program again.
;Notice the change of the path of tool tip.
```

SUBWORK1.CNC：

```
; Subroutine of cutting a work
G90;
G00 X0 Y0;
G01 Z-5. F500.;
X100. F1500.;
X90. Y10.;
G03 X10. Y90. I-40. J40.;
```

```
G01 X0 Y100.;
Y0;
Z0;
M99
```

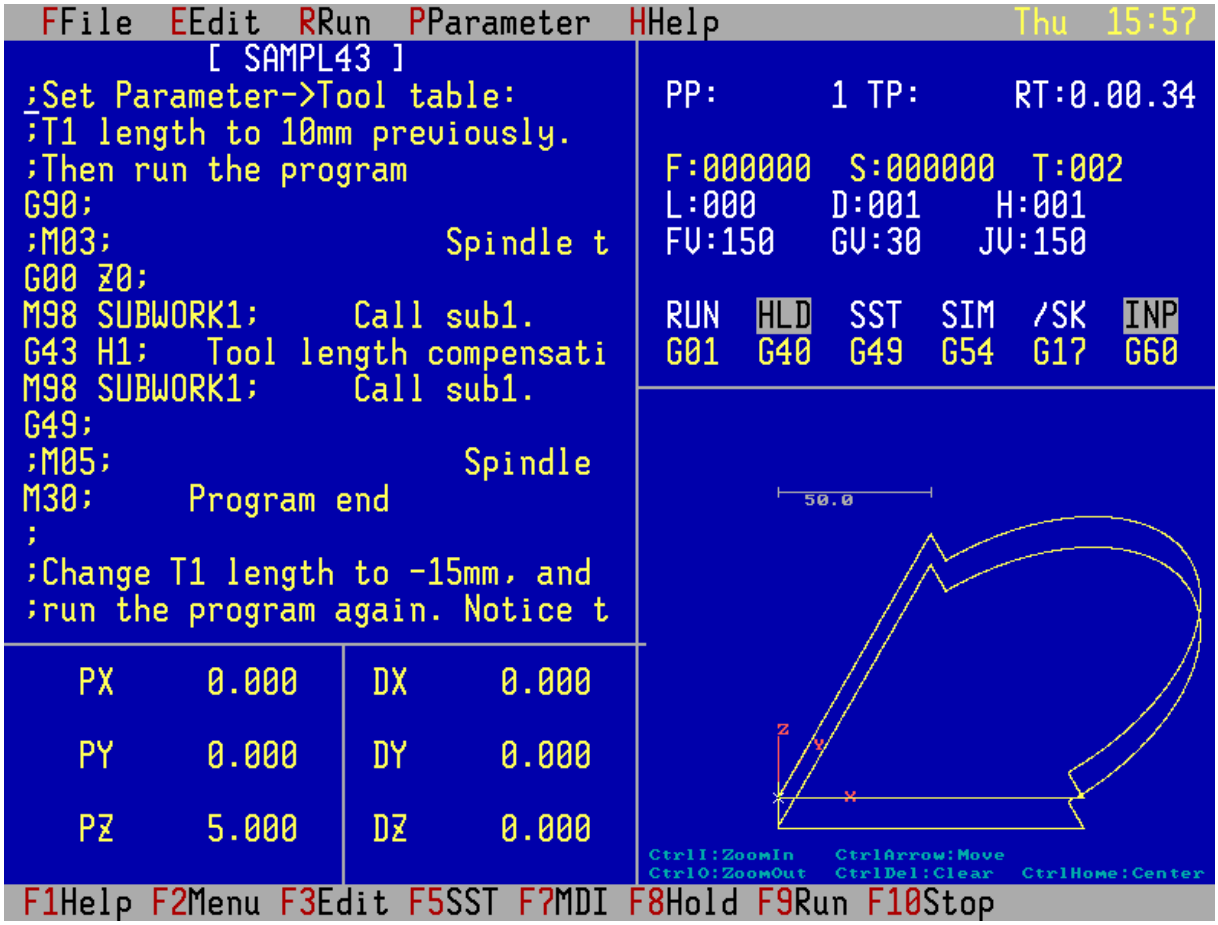


Figure 4-81 An example is illustrated

4.11.4 The example : G41/G42 Tool Radius Compensation

SAMPL41.CNC：

```
; Set〝Parameter→Tool table〞:
; T1 radius to 2.5 mm previously
; Start of normal path
;
G17 G90
G92 X0 Y0 Z0
G0 X100. Y100.
M97 P1041;          Call sub O1041
G0 X0 Y0
M01;                End of normal path
;
;Start of cutter left compensation
H1;                 Select tool No.1
G41 G0 X100. Y100.
M97 P1041;          Call sub O1041
G40 G0 X0 Y0;       End of G41
M30;                Program end
;
O1041;              Subroutine
G01 X200. F1500.
X190. Y110.
G03 X110.Y190.I-40.J40.
G01 X100. Y200.
Y100.
M99;                End of subroutine
```

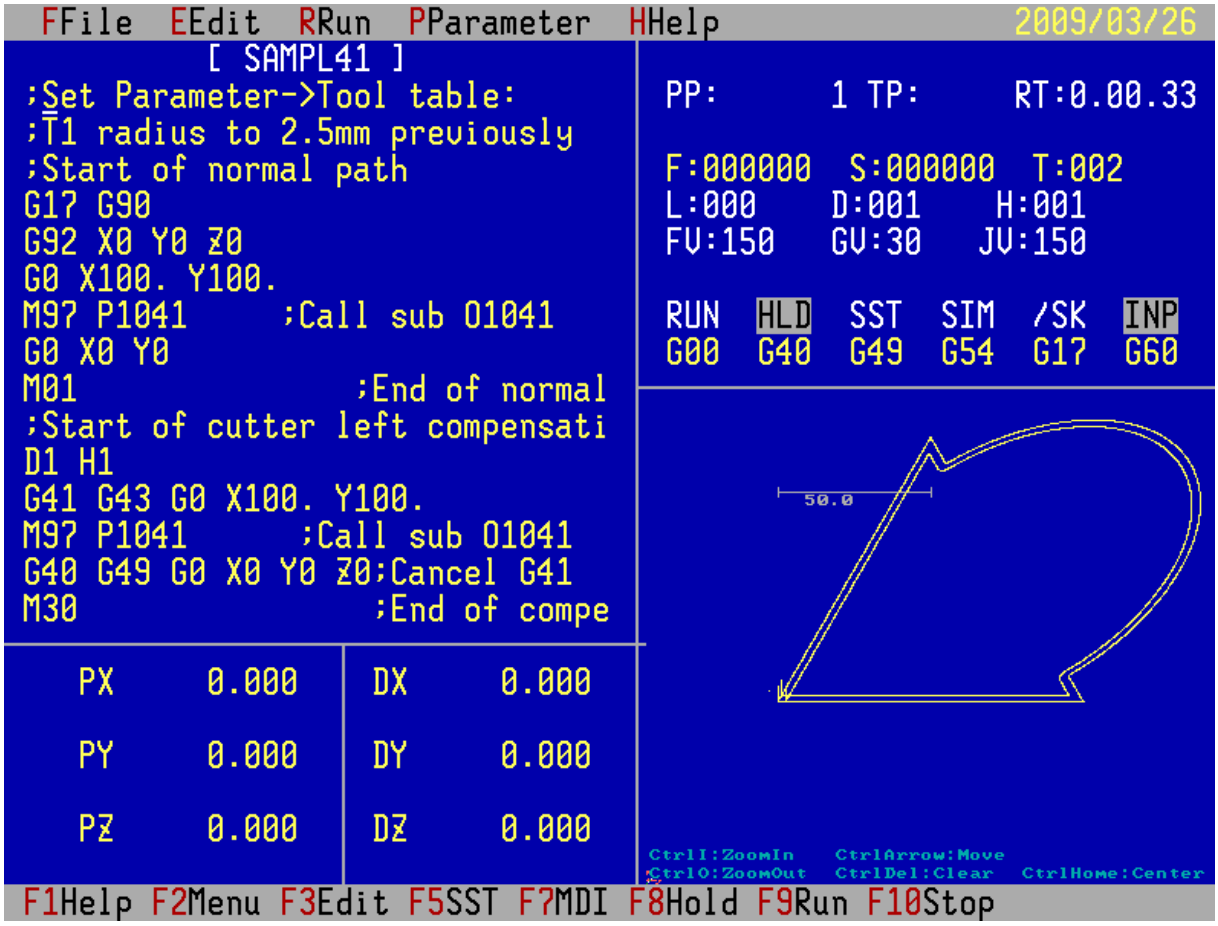


Figure 4-82 An example is illustrated

SAMPL42.CNC：

```
; Set 〝Parameter→Tool table〞：
; T1 radius to 2.5 mm previously
; Start of normal path
G17 G90
G92 X0 Y0 Z0
G0 X100. Y100.
M97 P1042;              Call sub O1042
G0 X0 Y0
M01;                    End of normal path
;
```

```
;Start of cutter right compensation
H1;                     Select tool No.1
G42 G0 X100. Y100.
M97 P1042;              Call sub O1042
G40 G0 X0 Y0;           End of G42
M30;                    Program end
;
O1042;                  Subroutine
G01 X200. F1500.
X190. Y110.
G03 X110.Y190.I-40.J40.
G01 X100. Y200.
Y100.
M99;                    End of subroutine
```

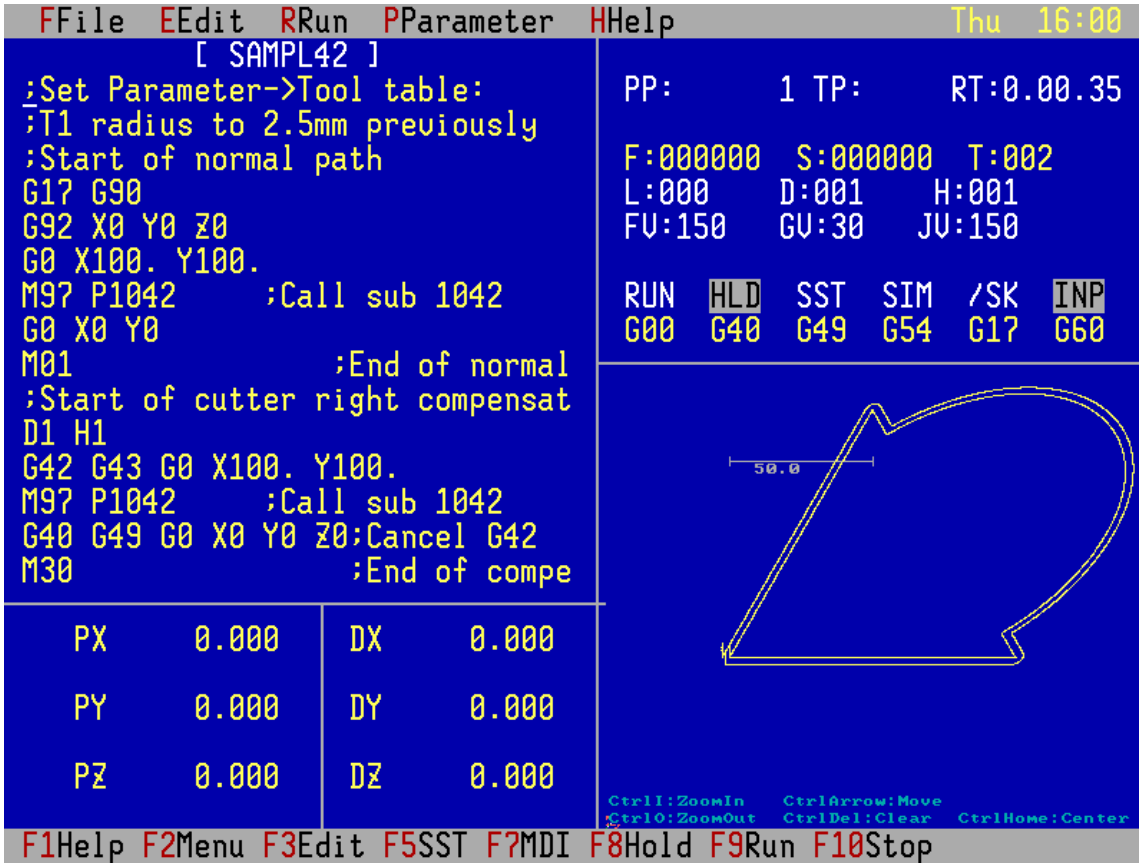


Figure 4-83 An example is illustrated

4.11.5 The example : G16 Polar Coordinate

SAMPL16A.CNC

```
; Hexagon
; Absolute radius, Absolute angle
G92 X0 Y0 Z0
G16; Switch to polar coordinate.
G90 G0 X200. Y60.;r=200. angle=60.
G1 Y120. F1500.; absolute angle
Y180.
Y240.
Y300.
Y360.
Y60.
G15; polar coordinate off
G90 G00 X0 Y0
M30
```

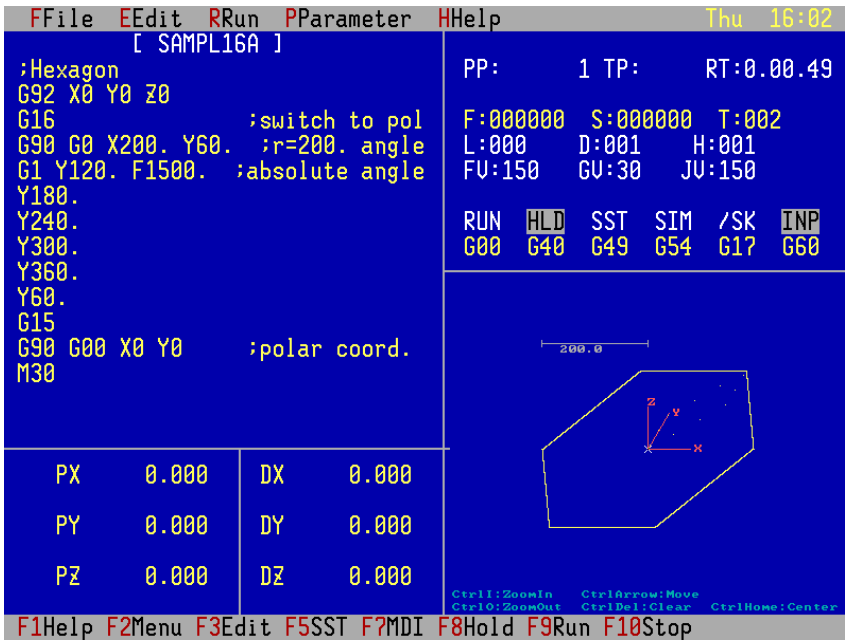


Figure 4-84 An example is illustrated

SAMPL16B.CNC

```
; Octagon
; Absolute radius, Incremental angle
G92 X0 Y0 Z0
G16; Switch to polar coordinate.
G90 G0 X200.Y45.; r=200. angle=45.
G91 G1 Y45.F1500.; Incremental angle
N10 Y45.
M95 P10 L6
G15; Polar coordinate off
G90 G00 X0 Y0
M30
```

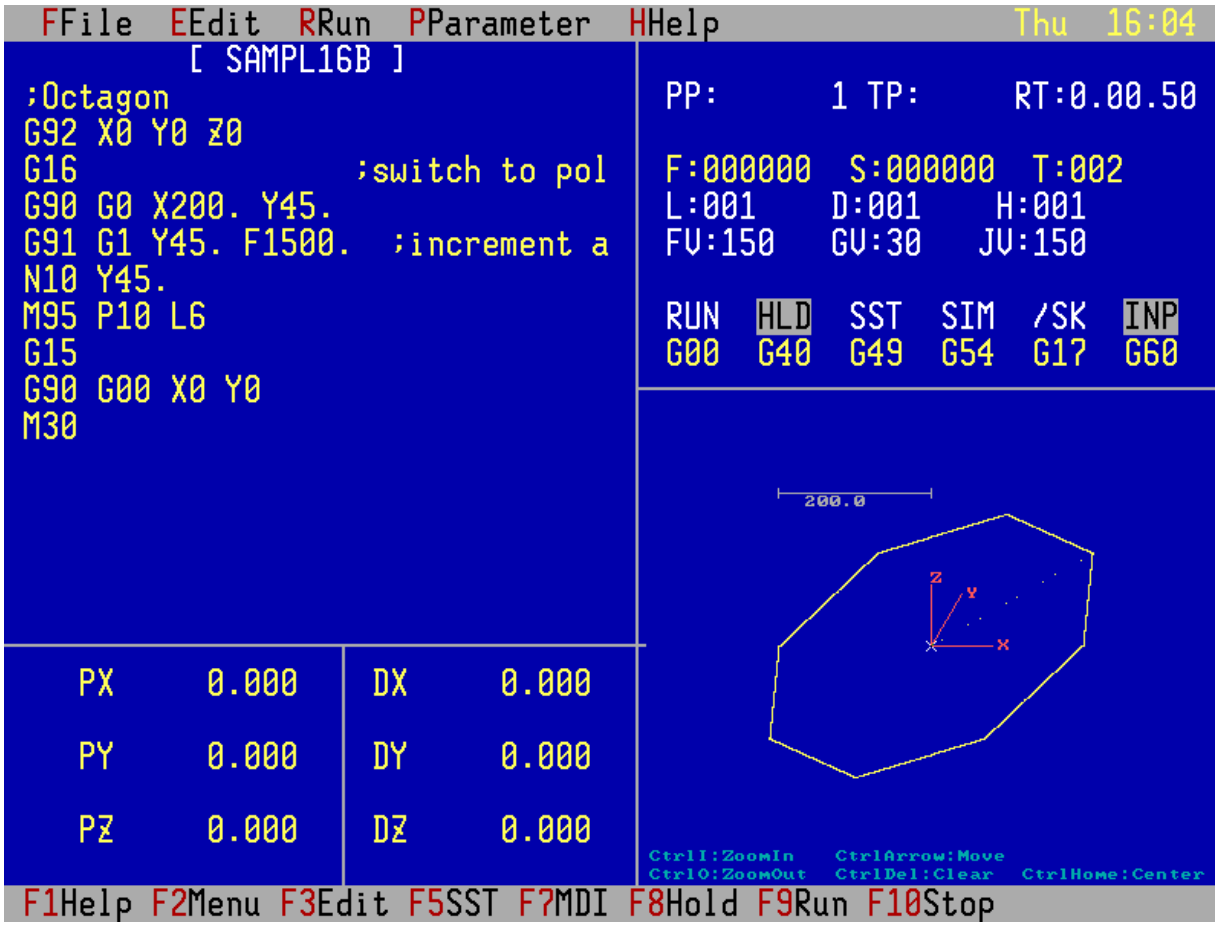


Figure 4-85 An example is illustrated

SAMPL16C.CNC：

```
; Hexagon
; Incremental radius, Incremental angle
G92 X0 Y0 Z0
G16; Switch to polar coordinate.
G91 G0 X200. Y60.
G1 X200. Y120. F1500. ;
N10 Y60.
M95 P10 L4
G15
G90 G00 X0 Y0
M30
```

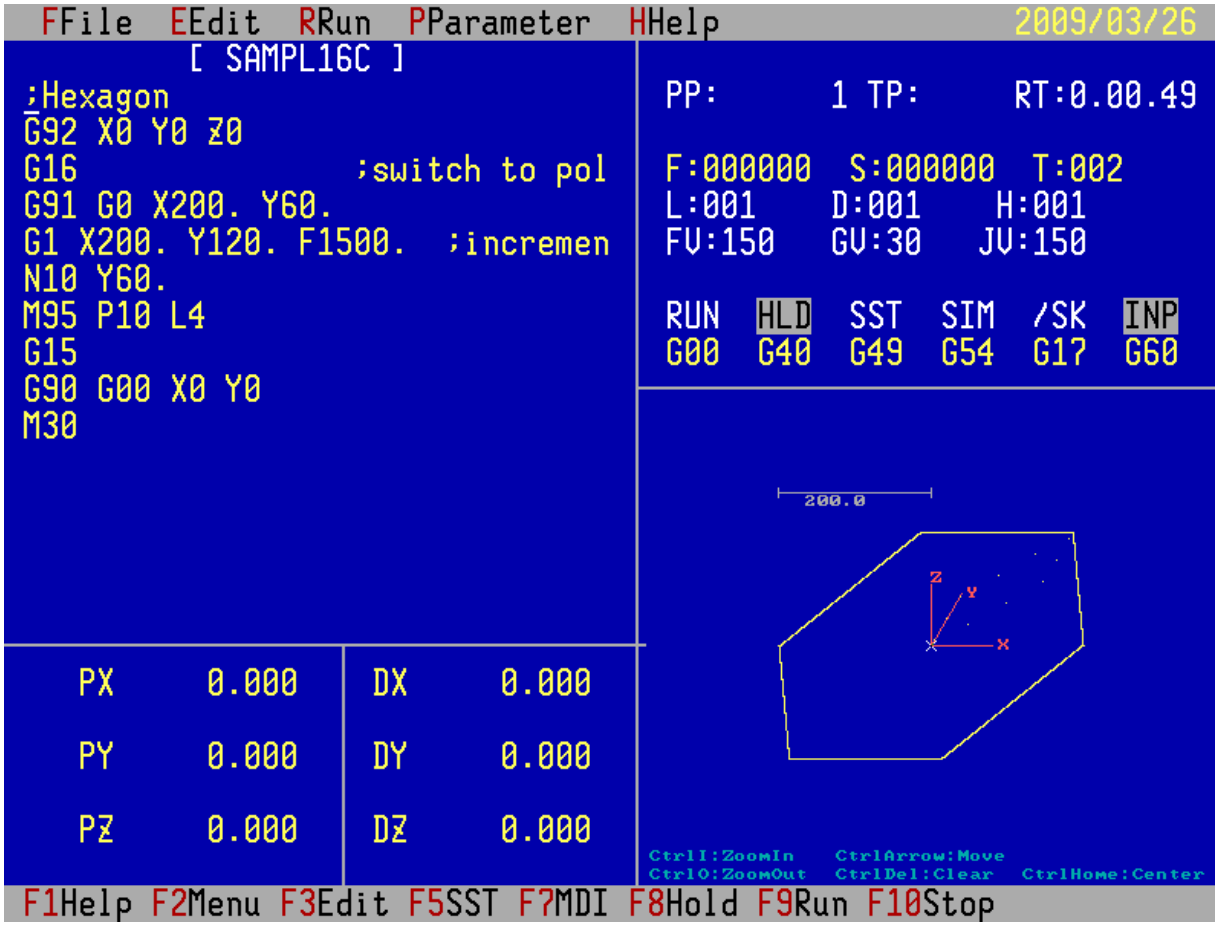


Figure 4-86 An example is illustrated

SAMPL16D.CNC：

```
; Lotus flower
; Incremental radius, incremental angle
G92 X0 Y0 Z0
G91
G16 G0 X100. Y0. F1500.
N10 G2 X250. Y160. R200.
M95 P10 L8
G15
G90 G0 X0 Y0 Z0
M30
```

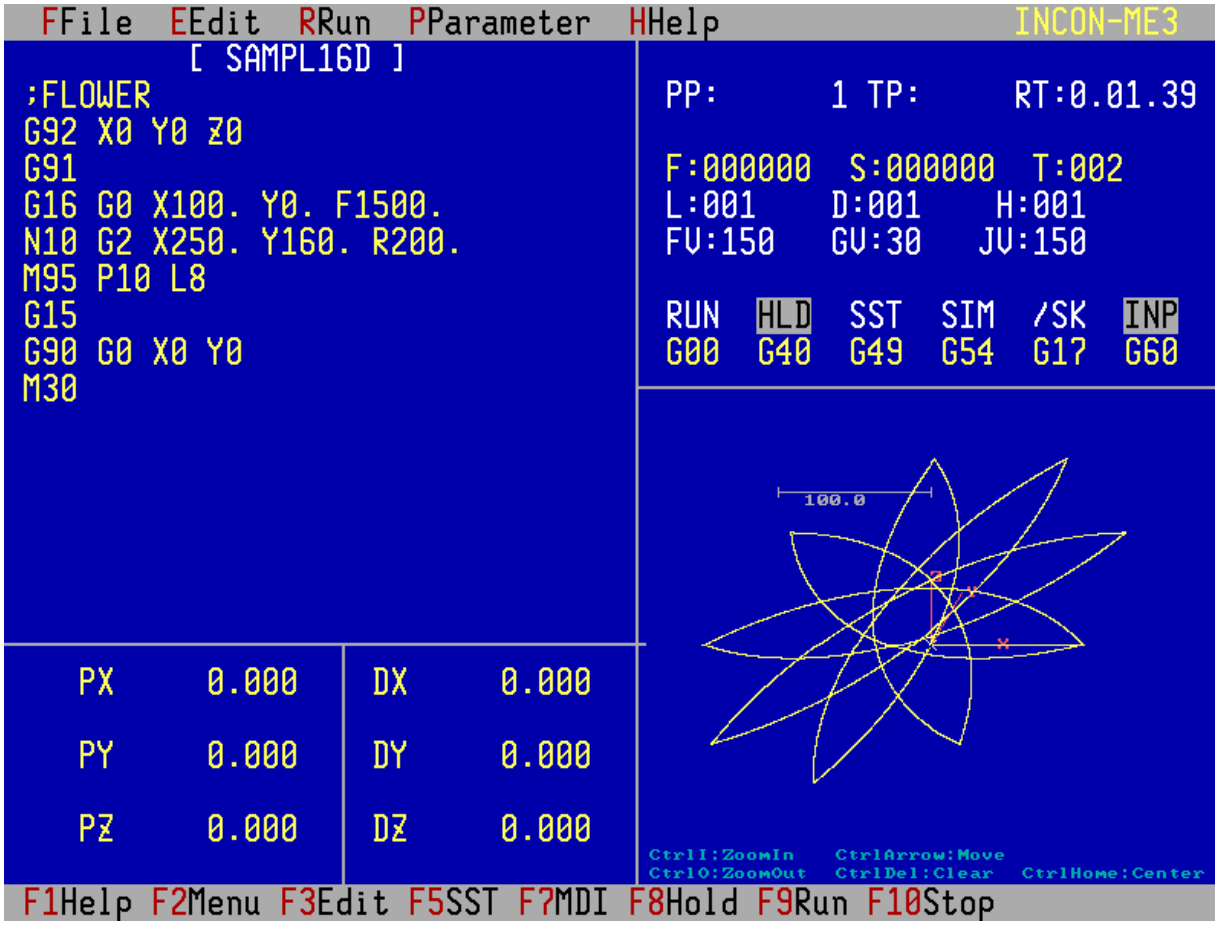


Figure 4-87 An example is illustrated

SAMPL16E.CNC

```
; Star fish
; Absolute radius, Incremental angle
G92 X0 Y0 Z0
G90 G16 G0 X200. Y0
G91 G2 Y144. R200. F1500.
N10 Y144. R200.
M95 P10 L4
;
N20 Y72. R200.
M95 P20 L4
G15
G90 G0 X0 Y0
M30
```

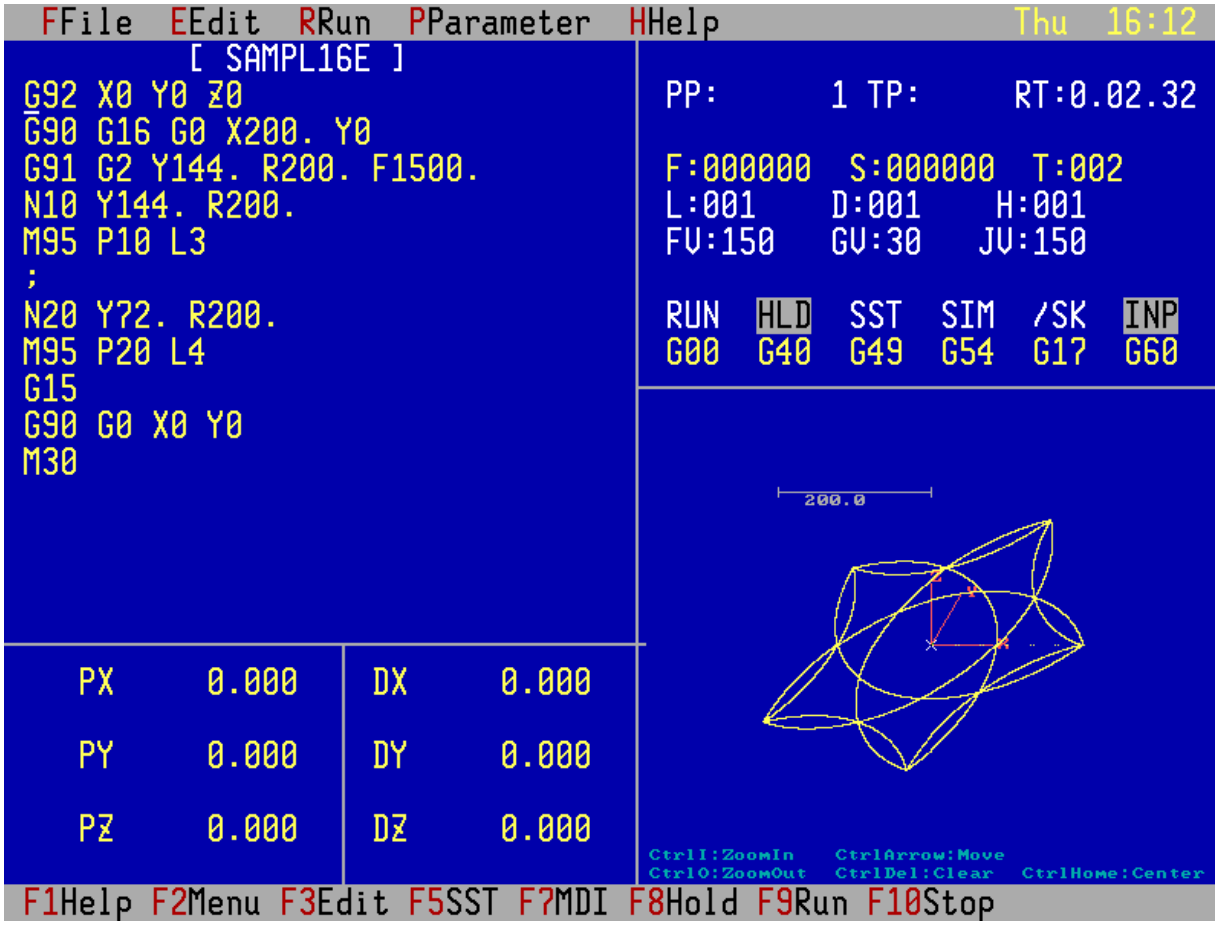


Figure 4-88 An example is illustrated

SAMPL16F.CNC

```
; Magic crystal
; Absolute radius, incremental angle
G92 X0 Y0 Z0
G90 G16 G0 X100. Y45.
G91 G1 X200. Y162. F1500.
N10 Y162.
M95 P10 L18
G15
M30
```

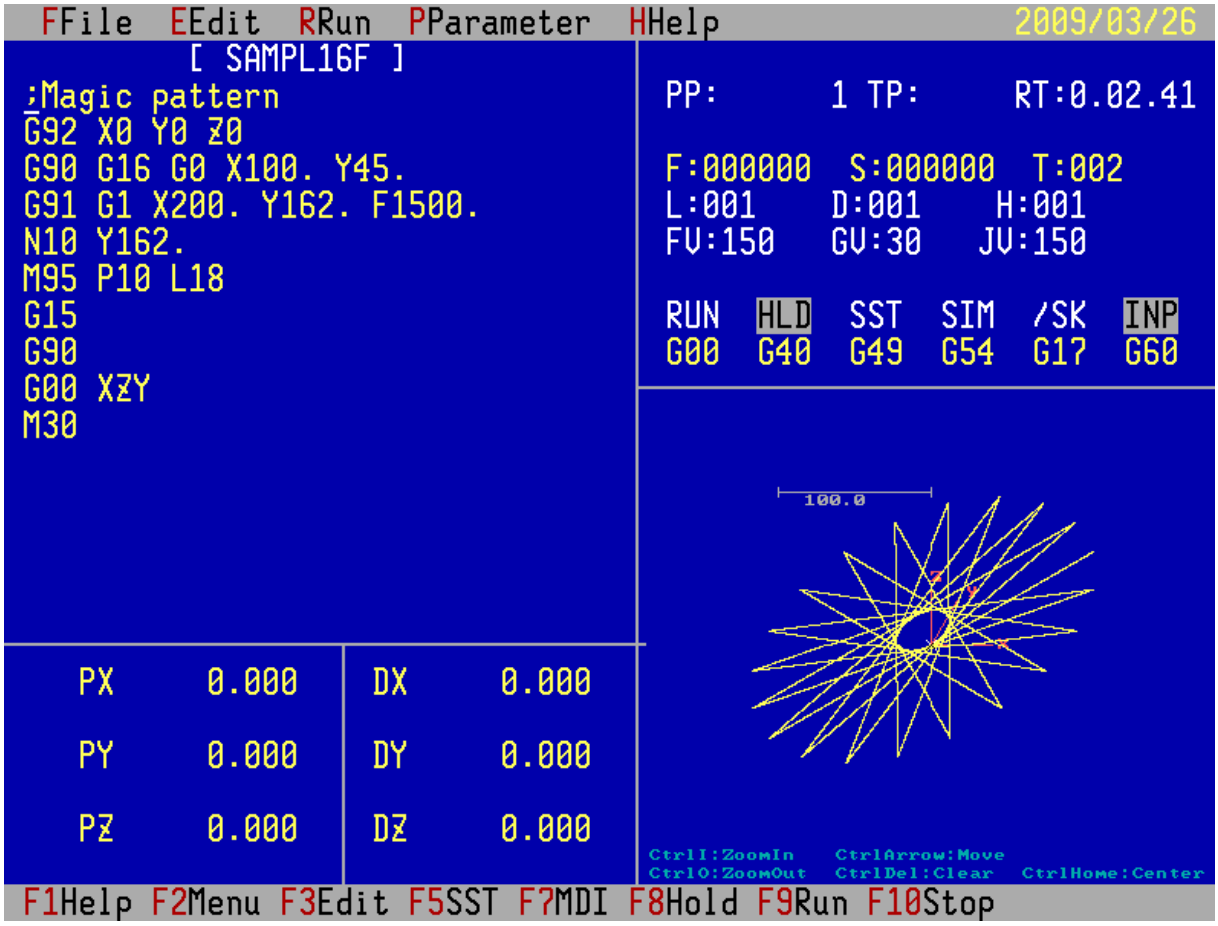


Figure 4-89 An example is illustrated

4.11.6 Miscellaneous Examples：

G01 corner chamfer and round : TS-G1-LR.CNC

```
; G1 with corner chamfer and round and their mirror paths
G92 X0 Y0 Z0
G91
G17; on X-Y plane
M97 P1217; Original path
G51 X Y Z I-1000 J1000 K1000
```

```
M97 P1217; Mirror to X axis
G50
G51 X Y Z I1000 J-1000 K1000
M97 P1217;Mirror to Y axis
G50
G51 X Y Z I-1000 J-1000 K1000
M97 P1217; Mirror to diagonal of X and Y axis
G50;
G90 G0 X0 Y0 Z0
M30
;
O1217
G0 X8. Y8. Z8.
G1 X20. F500.
Y10. L3.;chamfering corner
X-20. R5.;rounding corner
Y-10.
G0 X-8. Y-8. Z-8.
M99
;
```

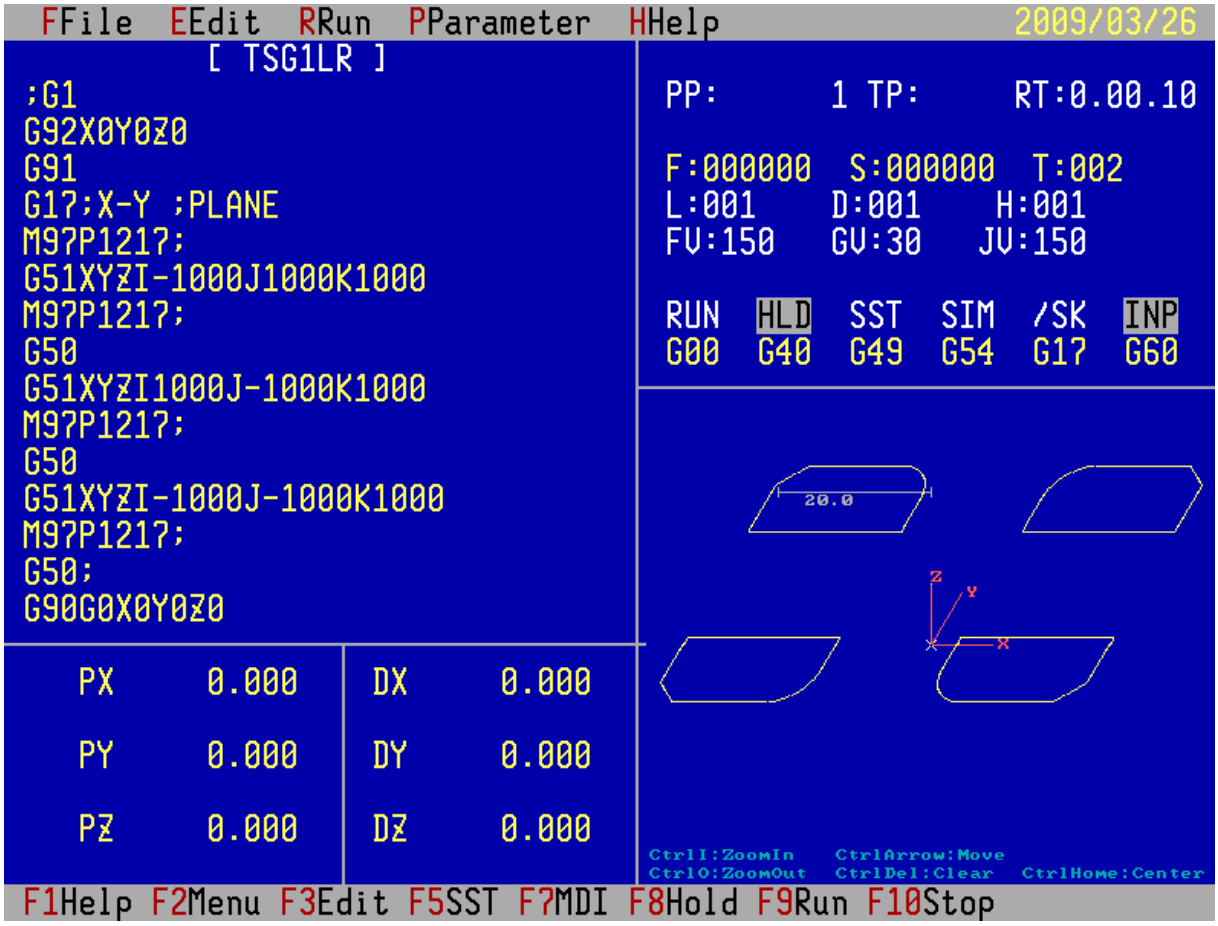


Figure 4-90 An example is illustrated

G16 mirror sample program: TS16-PLN1.CNC

```
G16 program the polar system on each plane
G92 X0 Y0 Z0
G91
G0 X-40.
G92 X0 Y0 Z0
G17 G16; Polar system on XY plane
N1 G1 X10. Y30. F1000.
Z10.
X10. Y180.
Z-10.
```

```
X0 Y180.
M95 P1L11 ; Loop from N1 11 times
G15;
G0 X40.
G92 X0 Y0 Z0
G18 G16    ; Polar system on XZ plane
N2 G1 Z10. X30. F1000.
Y10.
Z10. X180.
Y-10.
Z0 X180.
M95 P2L11 ;Loop from N2 11 times
G15;
G0 X40.
G92 X0 Y0 Z0
G19 G16    ;Polar system on YZ plane
N3 G1Y10. Z30. F1000.
X10.
Y10. Z180.
X-10.
Y0 Z180.
M95 P3L11 ;Loop from N3 11 times
G15;
G0 X-40.
G92 X0 Y0 Z0
M30.
```

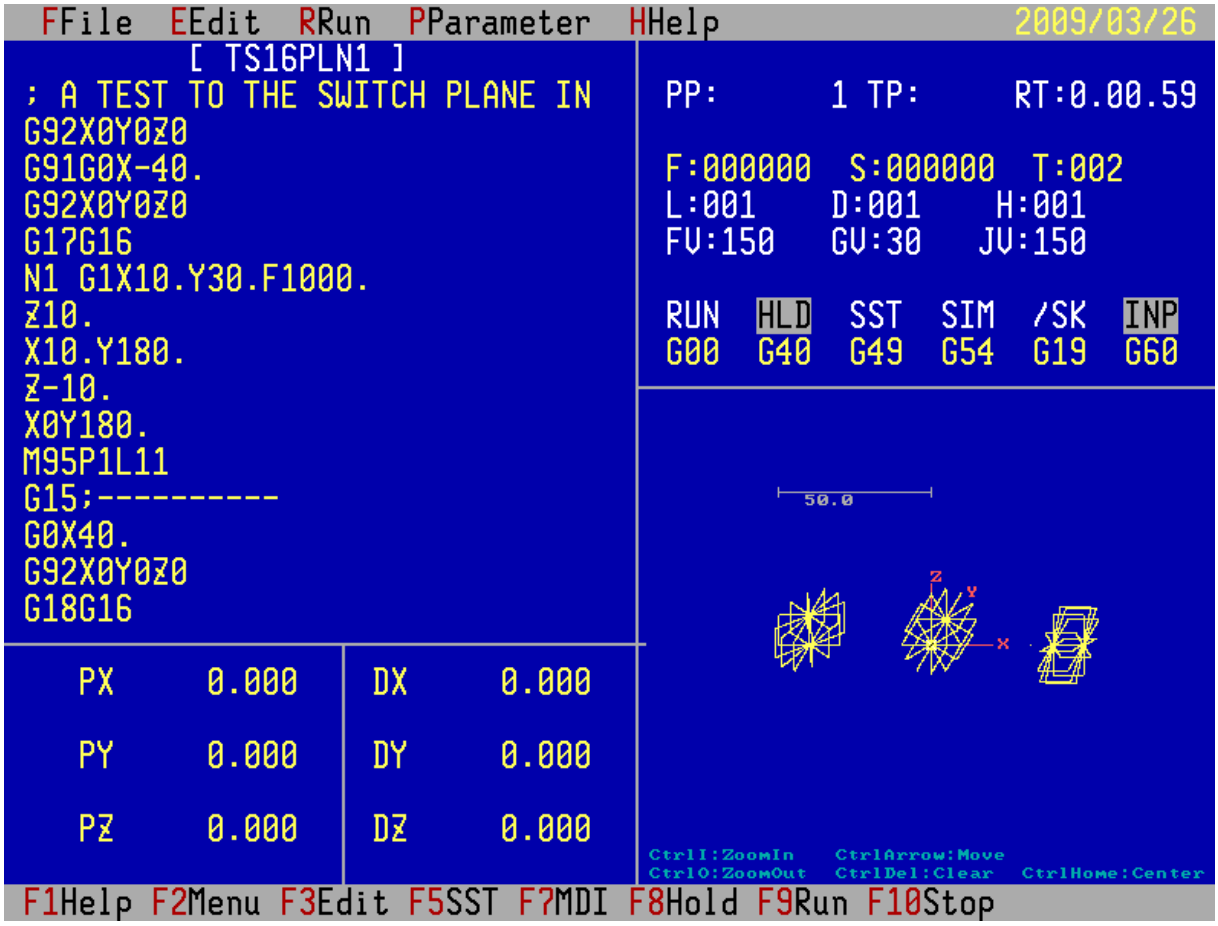


Figure 4-91 An example is illustrated

4.11.7 The example : G51/G68 Mirror, scaling and rotation

G51 mirror sample program : SAMPL51A.CNC

```
; Mirror function on each plane
G92 X0 Y0 Z0
G91
;
```

```
G17 G0 X-35. ; on X-Y plane
G92 X0 Y0 Z0
M97 P1234   ; Original path
G51 X Y Z I-1000 J1000 K1000
M97 P1234   ; Mirror to X axis
G50
G51 X Y Z I1000 J-1000 K1000
M97 P1234   ; Mirror to Y axis
G50
G51 X Y Z I-1000 J-1000 K1000
M97 P1234   ; Mirror to diagonal of X and Y axis
G50
;
G18 G0 X35.  ;on Z-X plane
G92 X0 Y0 Z0
M97 P1234   ; Original path
G51 X Y Z I-1000 J1000 K1000
M97 P1234   ; Mirror to X axis
G50
G51 X Y Z I1000 J1000 K-1000
M97 P1234   ; Mirror to Z axis
G50
G51 X Y Z I-1000 J1000 K-1000
M97 P1234   ; Mirror to diagonal of X and Z axis
G50
;
G19 G0 X30.  ; on Y-Z plane
G92 X0 Y0 Z0
M97 P1234   ; Original path
G51 X Y Z I1000 J1000 K-1000
M97 P1234   ; Mirror to Z axis
G50
G51 X Y Z I1000 J-1000 K1000
M97 P1234   ; Mirror to Y axis
```

```
G50
G51 X Y Z I1000 J-1000 K-1000
M97 P1234    ; Mirror to diagonal of Y and Z axis
G50
G0 X-30.
G17 G92 X0 Y0 Z0
M30
;
O1234
G0 X5. Y5. Z5.
G1 X10. F500.
Y10.
X-10.
Y-10.
X10. Y10.
X-10. Y-10. Z12.
Z-12.
G0 X-5. Y-5. Z-5.
M99
```

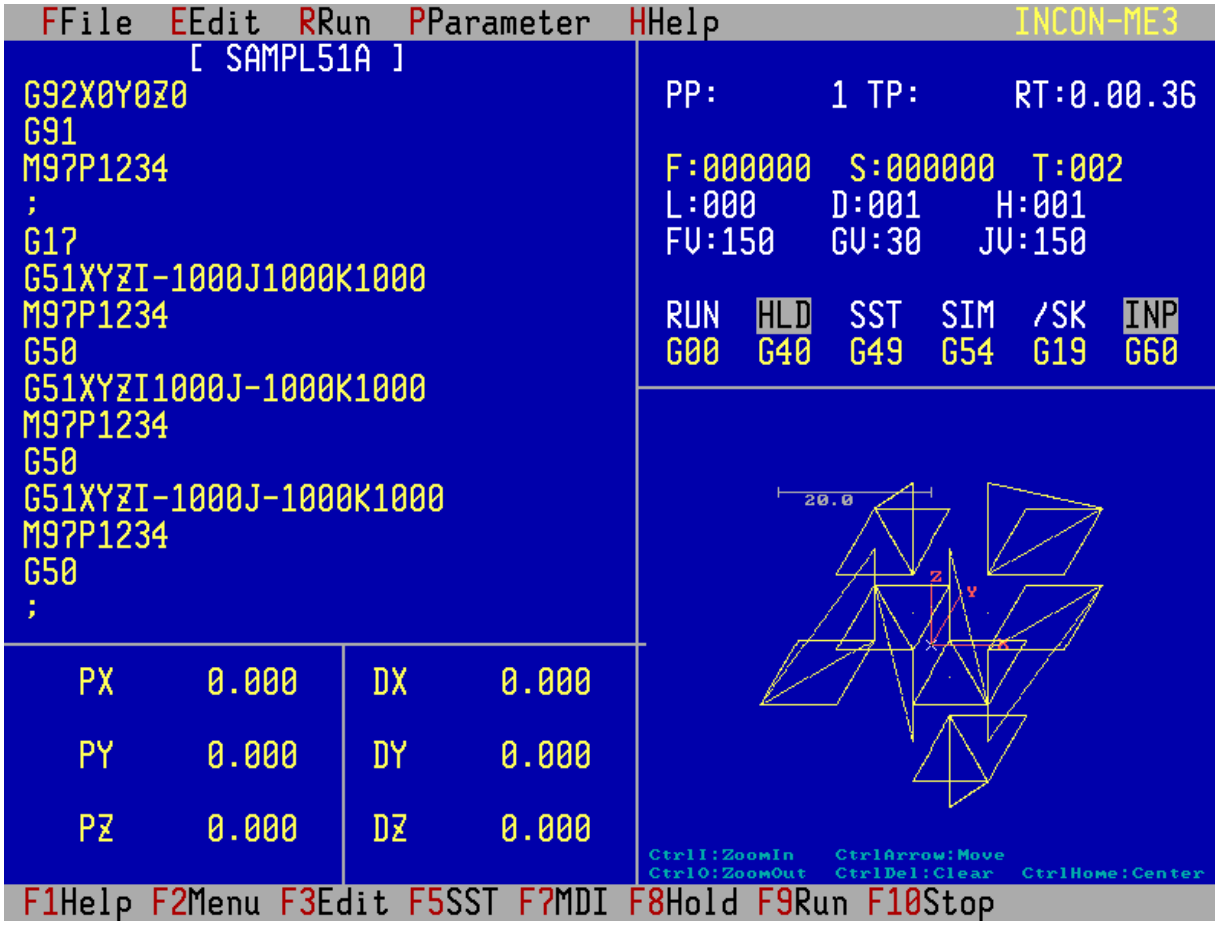


Figure 4-92 An example is illustrated

G68 rotation and G51 mirror sample: TS68-51A.CNC

```
; the combination of rotation and mirror on XY plane
G17 G91
G92 X0 Y0 Z0
M97 P0123    ; original path in first quadrant
G51 X Y Z I-1000 J1000 K1000
M97 P0123    ; Mirror to X axis
G50
G51 X Y Z I1000 J-1000 K1000
M97 P0123    ; Mirror to Y axis
G50
```

```
G51 X Y Z I-1000 J-1000 K1000
M97 P0123    ; Mirror to diagonal of X and Z axis in third quadrant
G50
M30
;
;
O0123        ;case 1
M97 P1234    ;original path
G68 X0 Y0 R90.; rotate 90° and duplicate a path
M97 P1234
G69
M99
;
;
O1234         ;case 2
G0 X10. Y5.
G1 X10. Y10. F1500.
X10.
X10. Y-10.
X-10.
X-5. Y5.
X-5. Y-5.
X-10.
G0 X-10. Y-5.
M99
```

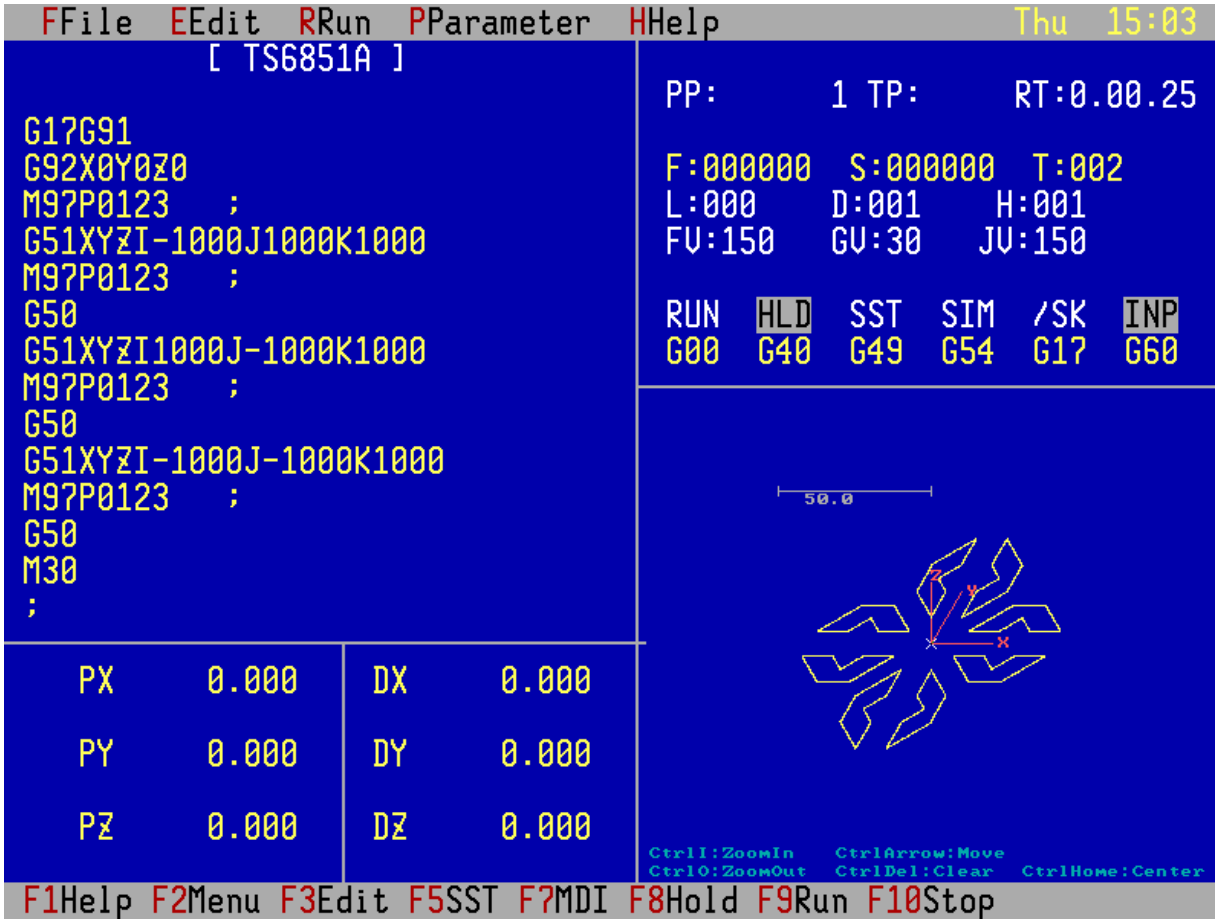


Figure 4-93 An example is illustrated

If we change the content in subprogram **O0123** as follows, then we'll get other tracing paths:

```
;
; 〔case 1〕rotate and duplicate
O0123
G68 X0 Y0 R45. ; rotate 45°
M97 P1234    ;duplicate a path
G69
G68 X0 Y0 R-45.; rotate -45°
M97 P1234    ;duplicate a path
G69
M99
```

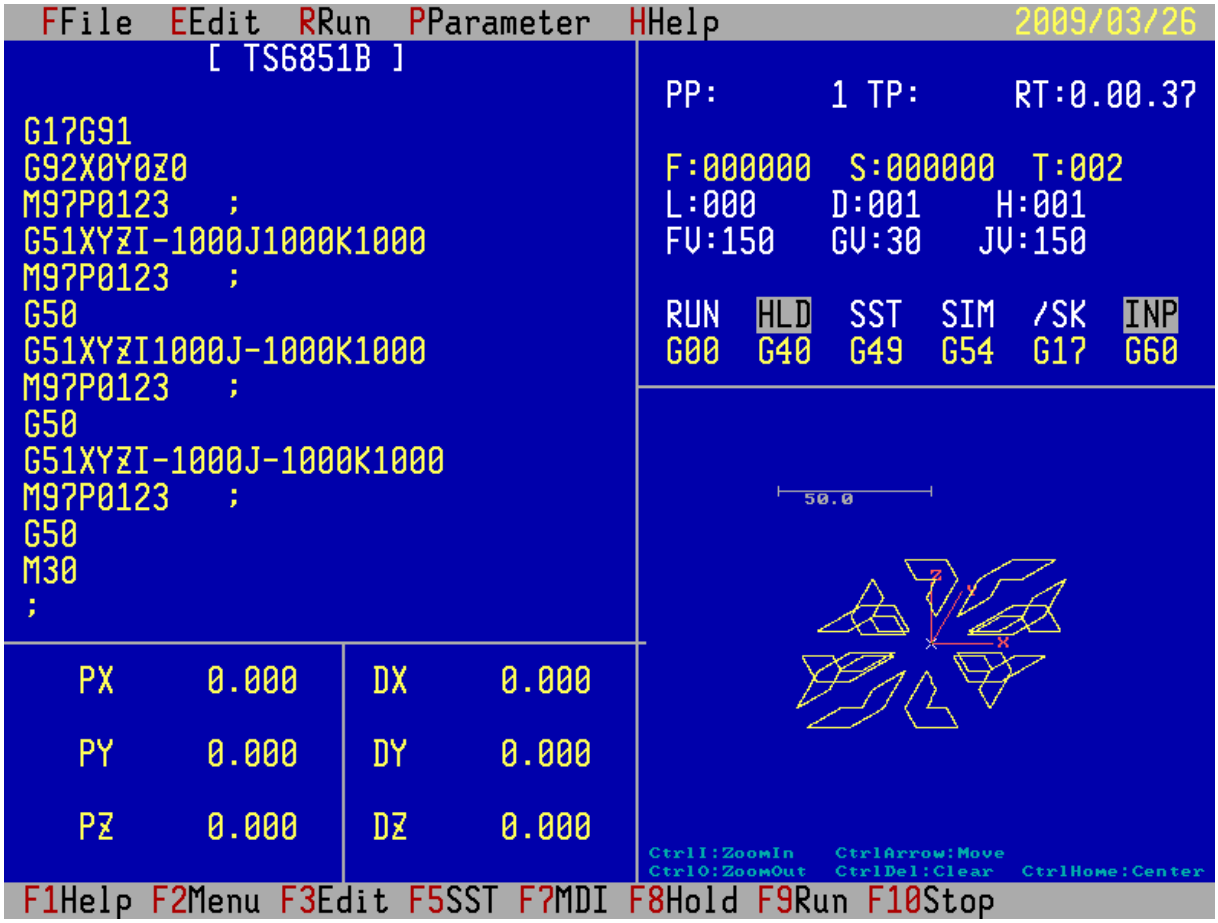


Figure 4-94 An example is illustrated

```
; 〔case 2〕original path
O0124
M97 P1234      ; original path
G68 X0 Y0 R45.; rotate 45°
M97 P1234      ; duplicate a path
G69
G68 X0 Y0 R90.; rotate 90°
M97 P1234      ; duplicate a path
G69
G68 X0 Y0 R135.; rotate 135°
M97 P1234     ; duplicate a path
G69
M99
```

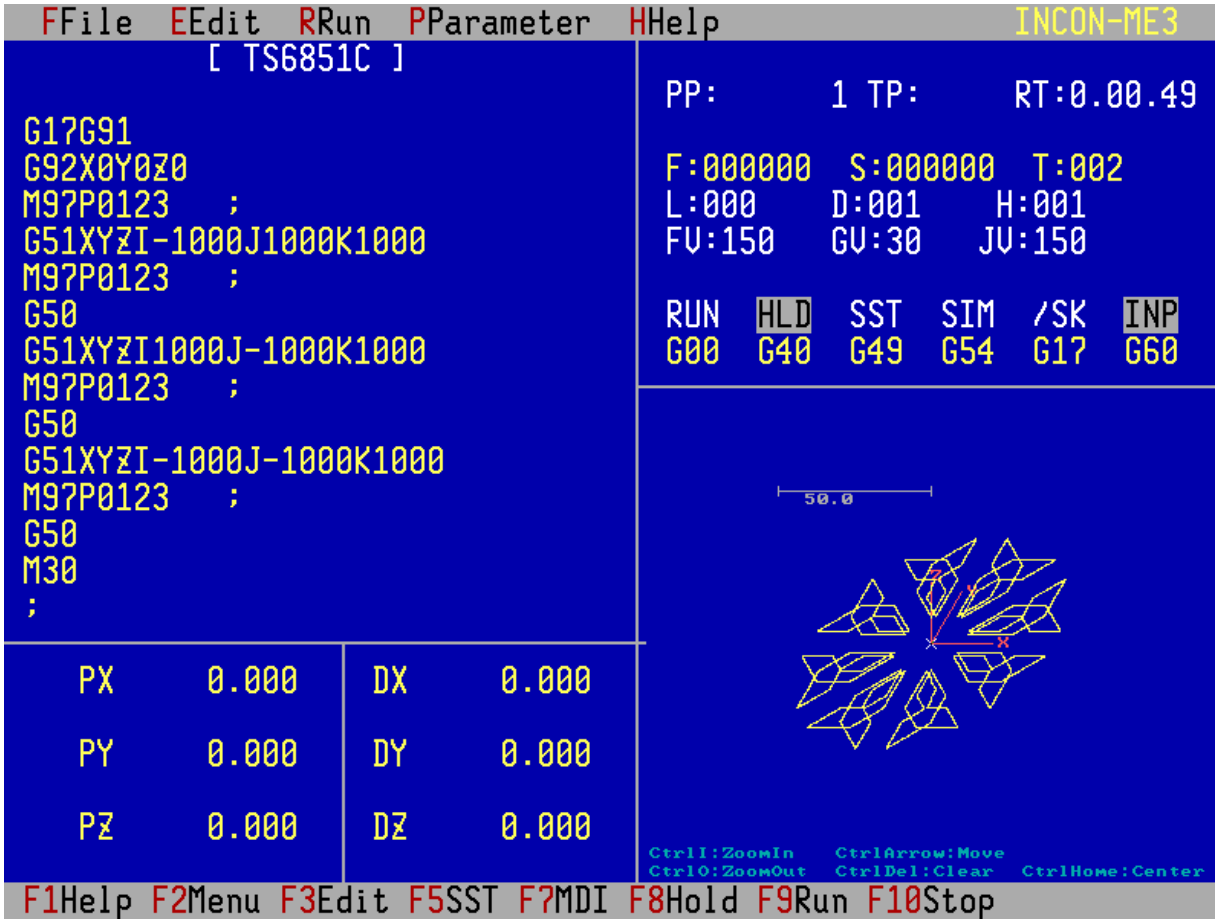


Figure 4-95 An example is illustrated

4.11.8 The Example : G73~G89 cancycle

G81/85 cancycle example：TS16-815.CNC

```
; G81/85 cancycles in polar system
G92 X0 Y0 Z0
G90 G0 X50.
G2 I-50. J0 F3000.
G0 X100.
```

```
G2 I-100. J0 F3000.
G0 X0 Y0
;
G16
G90 G99 G81 X50. Y0 Z-15. R5. F500.
N1 G91 G81 Y30. Z-10. R5. F500.
M95 P1L10
G15
G90 G0 X0 Y0 Z0
;
G16
G90 G98 G85 X100. Y0 Z-20. R5.F500.
N2 G91 G85 Y20. Z-15. R5. F500.
M95 P2 L16
G15
G90 G0 X0 Y0 Z0
M30
```

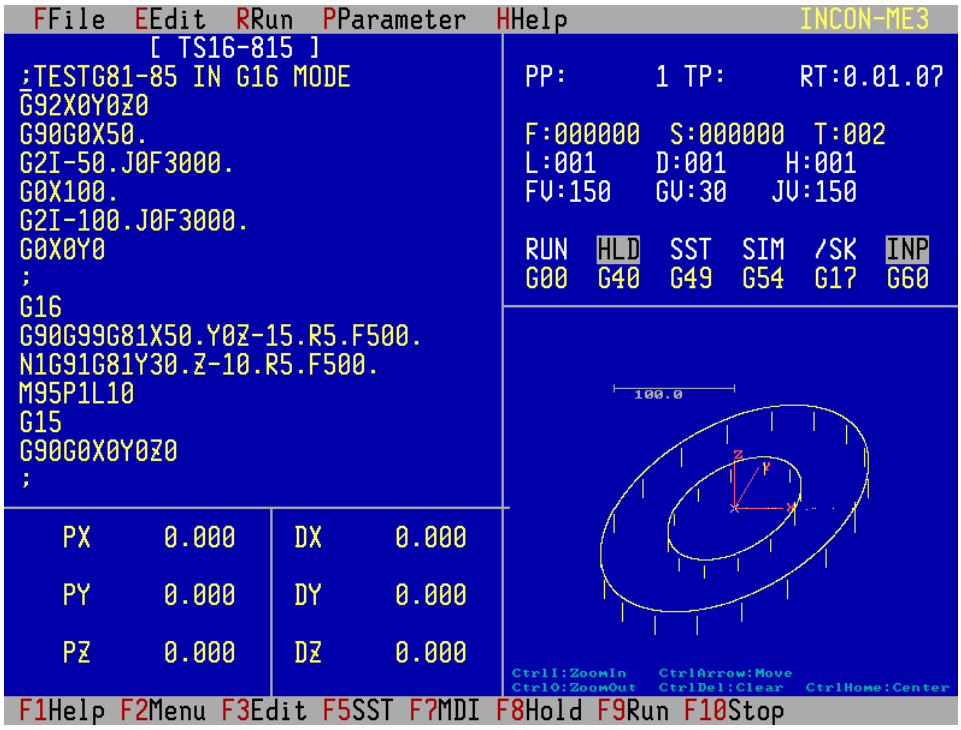


Figure 4-96 An example is illustrated

G73/G83 example：TS73-83.CNC

```
; G73 and G83 can cycles in Cartesian system
; Set retract D value=3.0 previously
G92 X0 Y0 Z0
G90 G0 X5. Y5.
G1 X70. F3000.
Y15.
X5.
Y5.
;
G90 G99 G73 X10. Y5. Z-25. Q5. R5. F500.
N1 G91 G73 X10. Z-20. R5. Q5. F500.
M95 P1L4
G90 G0 X70. Y15. Z0
;
G90 G98 G83 X62. Z-25. Q5. R5. F500.
N2 G91 G83 X-10. Z-20. R5. Q5. F500.
M95 P2L4
G90 G0 X0 Y0 Z0
M30
```

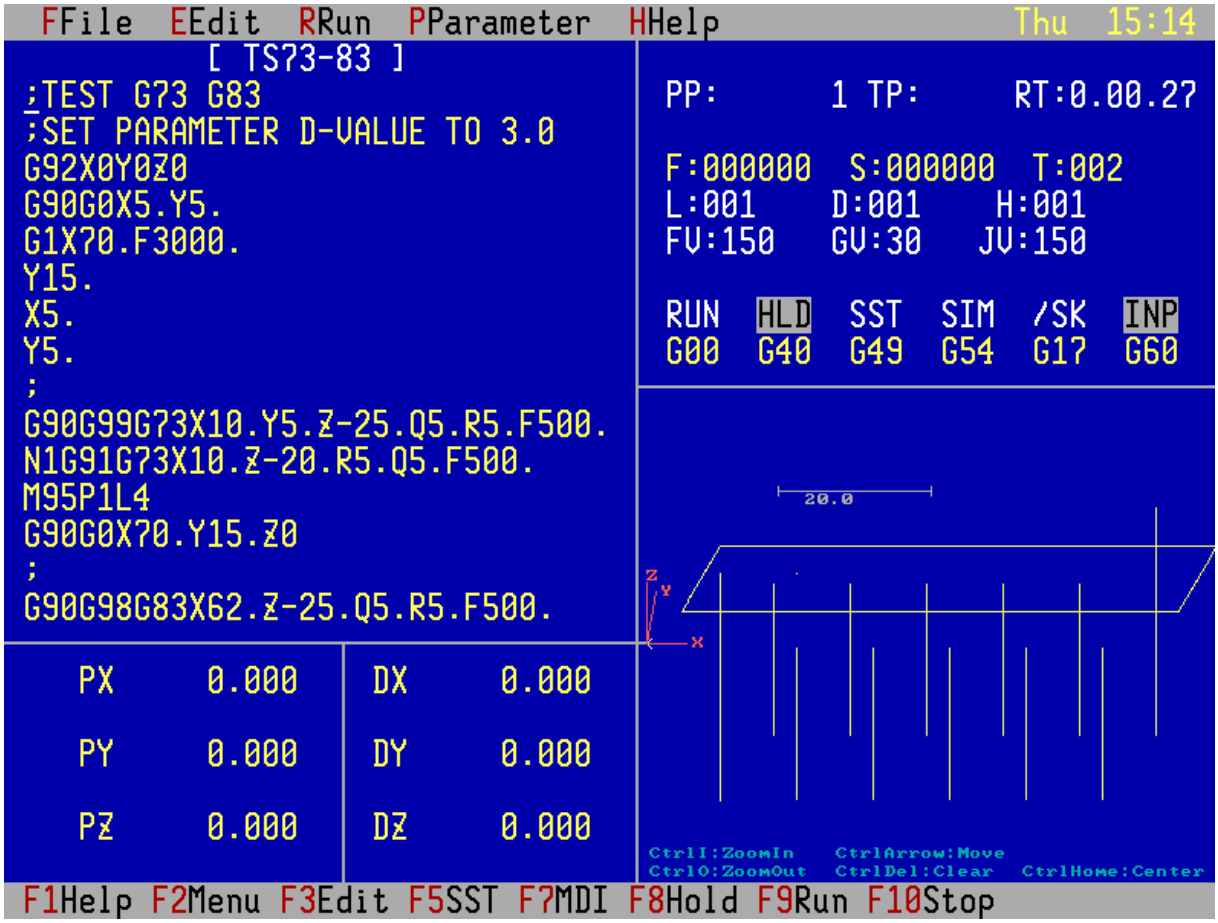


Figure 4-97 An example is illustrated

4.11.9 the example：G70.00/70.01 pocket cutting

SAPG70xx.CNC

```
;Example program of pocket cutting
;Set No.1 tool radius=6.0 in 〝Parameter→Tool table〞
;Set 〝Cut in retract length = 1.0〞 in 〝Parameter→Cancycle parameters〞 previously.
G92 X0 Y0 Z5.0
```

```
D1 ;select tool No.1
G91
/M03
G70.00 G2 X0 Y0 Z-5. U112. V88. W-13. C1. J0.5 K1. Q4. L10. I100 F1000 E650
G0 X130.
G70.01 G3 X0Y0Z-5. R56. W-13. C1. J0.5 K1. Q4. L10. I100 F1000 E650
G90 G0 X0 Y0 Z5.
/M05
M02
```

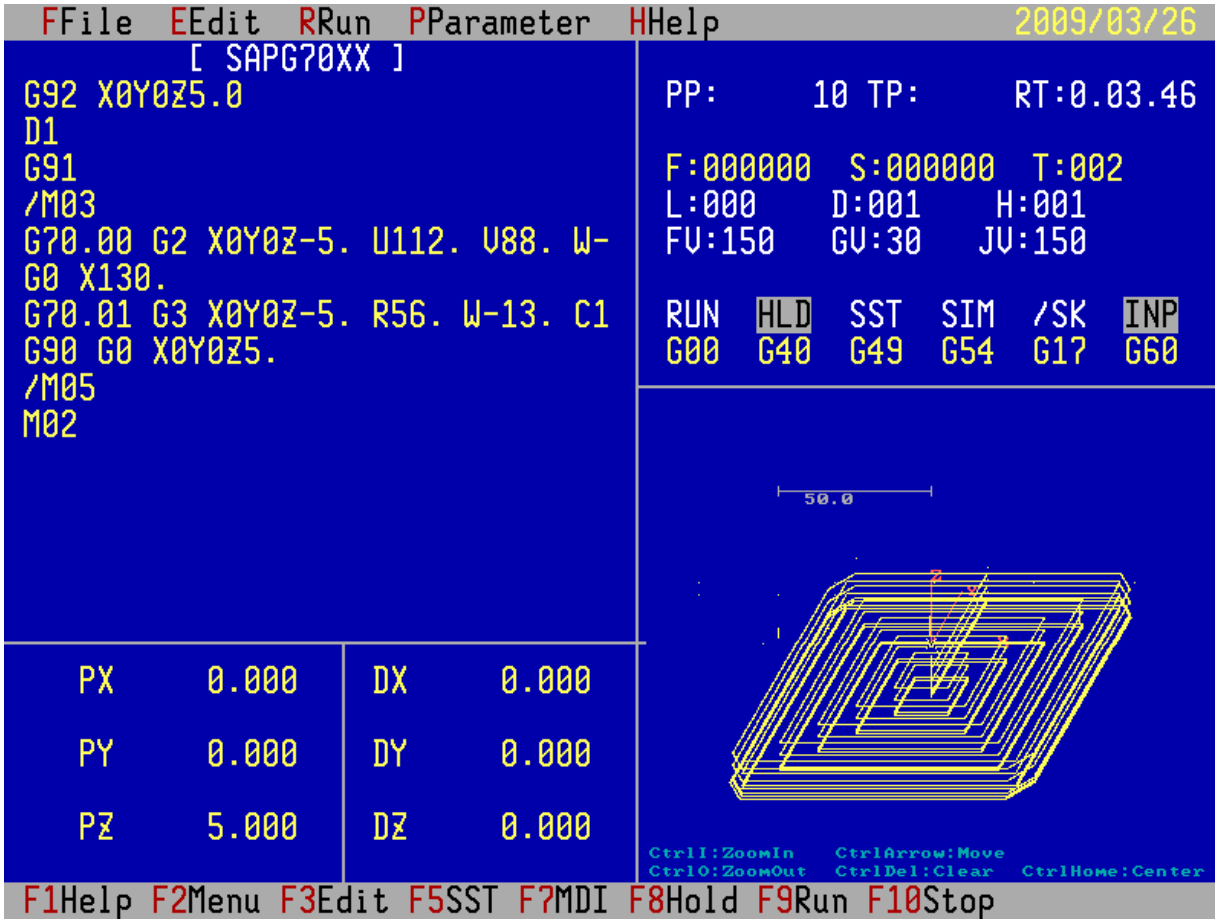


Figure 4-98 An example is illustrated

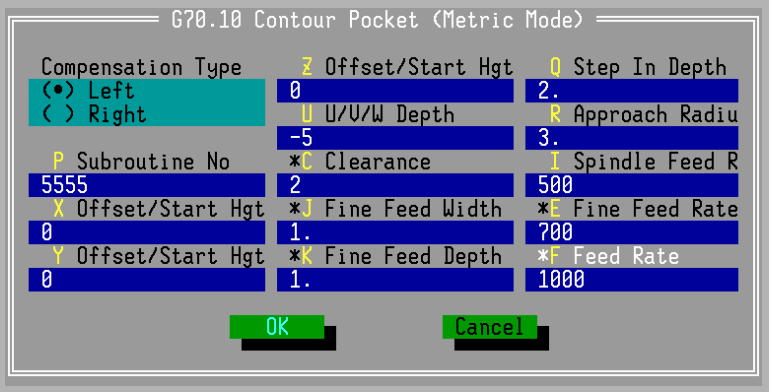
4.11.10 the example: G70.10 Contour Pocket

IR_PKT_1.CNC

```
;Example program of contour pocket cutting
;Set No.1 tool radius=1.0 in 〝Parameter→Tool table〞
;
;main program
G90 D1
G0 X0 Y0 Z5.
G41
M97 P5555
G40
G70.10 G41 P5555 X0 Y0 Z0 W-5. C2. J1. K1. Q2. R3. I500 E700 F1000
G0 Z10.

X0 Y0

M30
;
;Subprogram〔describe contour〕
O5555
G90 G0 X29.5 Y30.9
G1 X45.6
Y37.6
G2 X51.2 R-10.25
G1 Y30.9
X88.
Y42.7
X80.2 Y44.9
G2 Y49.6 R-8.25
G1 X88. Y52.1
Y65.3
```



```
X29.5
Y30.9
M99
```

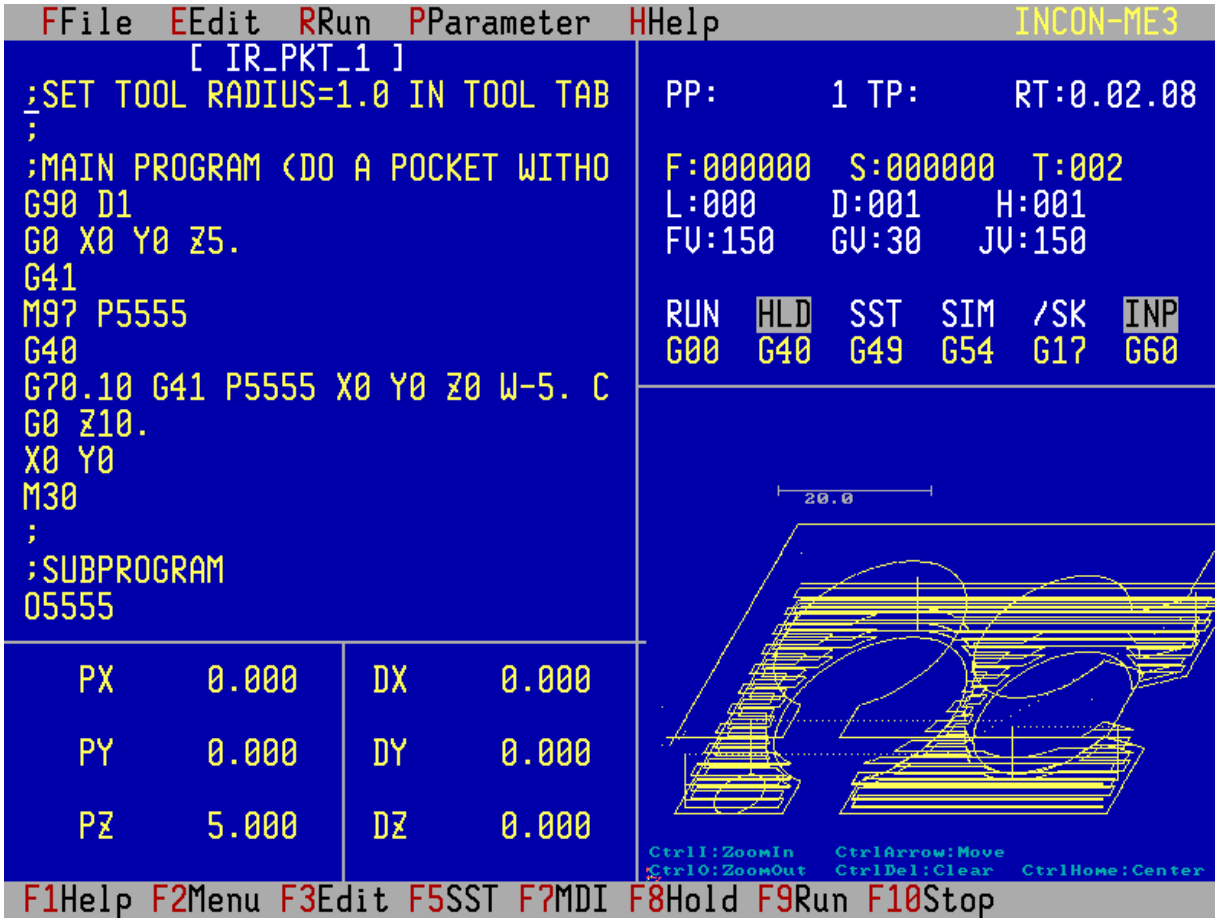


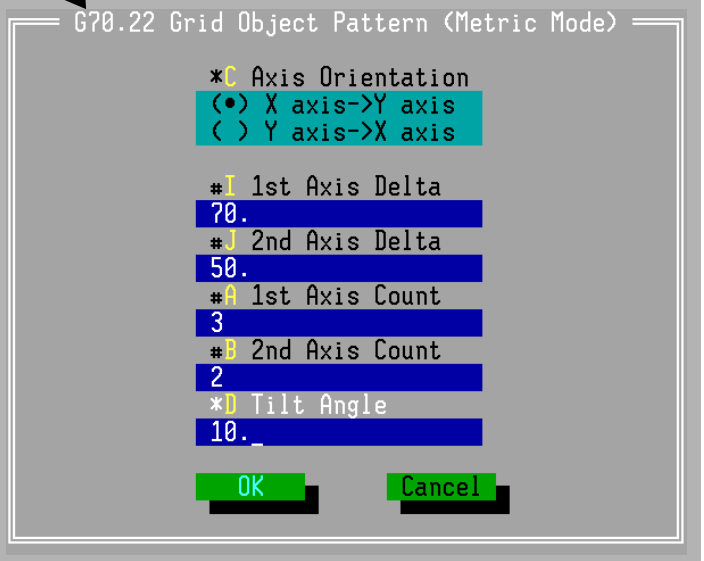
Figure 4-99 An example is illustrated

4.11.11 the example: G70.22 Grid Object pattern

OB_GR.CNC：

```
;Example of Grid Object pattern
;Set No.1 tool radius=1.0 in 〝Parameter→Tool table〞
;
;main program
G90 D1
G0 X0 Y0 Z10.
/M97 P5555
G70.22 C0 I70. J50. A3 B2 D10.
@ G70.10 G41 P5555 X70.0 Y-25.0 Z0 W-3. C2. J1. K0 Q3. R3. I500 E700 F1000
G0 Z10.

X0 Y0
M30
;
;Subprogram〔describe contour〕
O5555
G90 G0 X29.5 Y30.9
G1 X45.6
Y37.6
G2 X51.2 R-10.25
G1 Y30.9
X88.
Y42.7
X80.2 Y44.9
G2 Y49.6 R-8.25
G1 X88. Y52.1
Y65.3
X29.5
```



```
Y30.9
M99
```

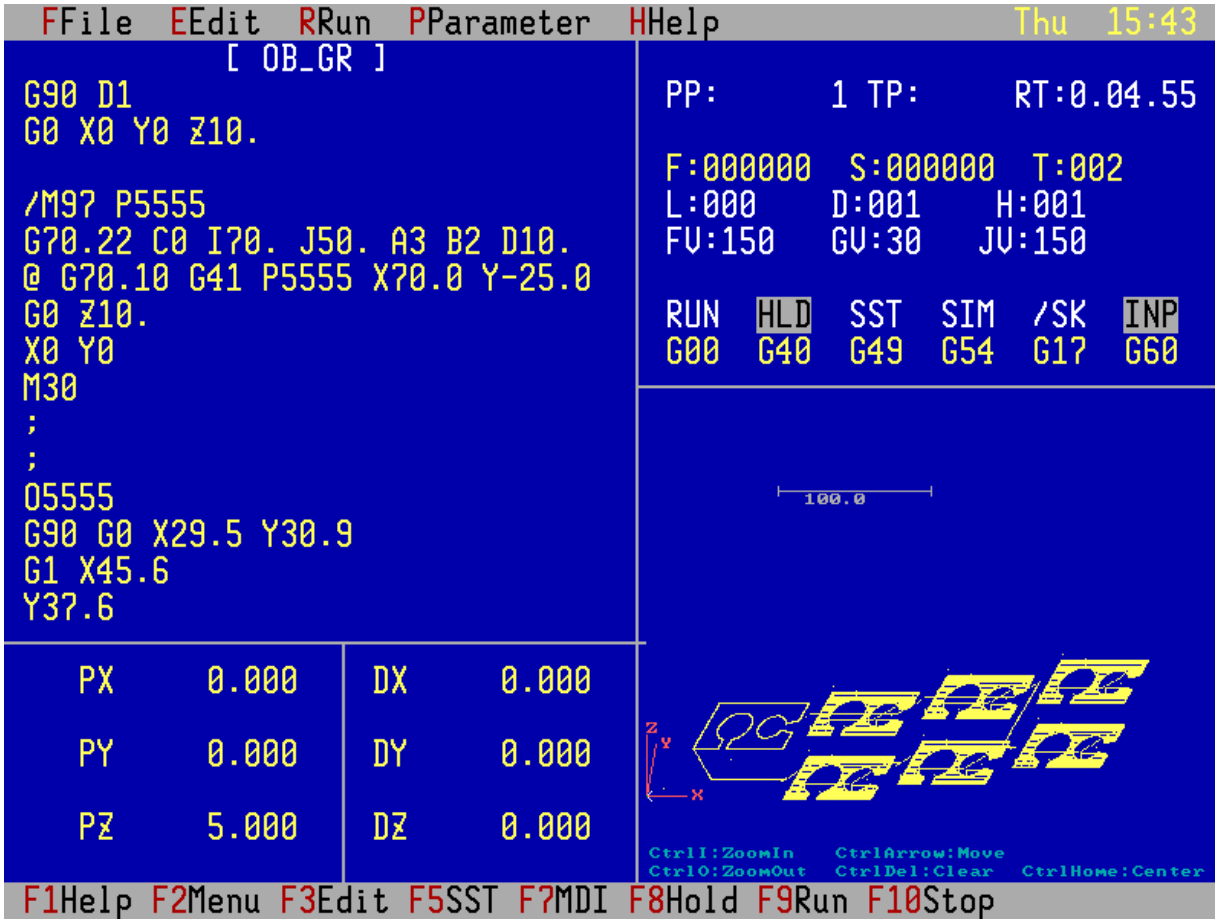


Figure 4-100 An example is illustrated

Chapter 5 Optional Facility

INCON-ME3 provides optional facilities for users. Users can get desired optional facility after finishing business process.

Users can pull down the menu item 〝**Help→About INCON-ME3**〞 and look at the code of optional function on the system information screen.

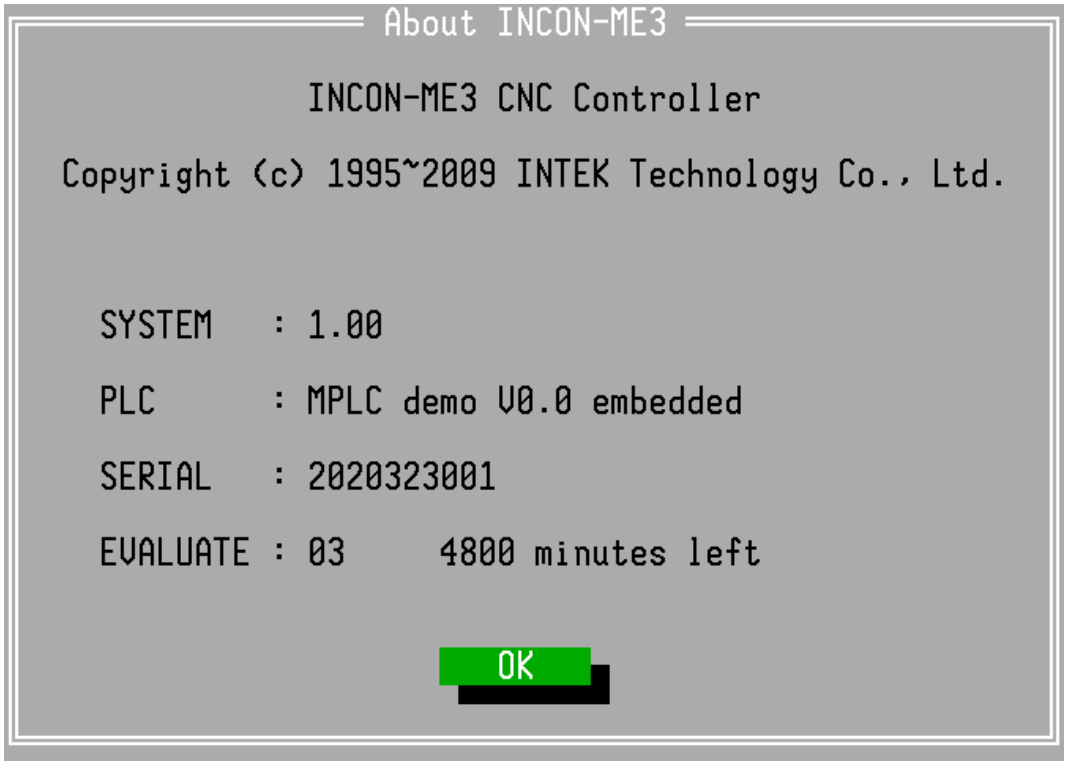


Figure 5-1 The system information

Each of codes of optional facilities enables by binary rotation 〔1: enable; 0: unable〕, but shows by binary hexadecimal, for example, 〝**FACILILY: 03**〞 means ME3 enables

Function1 and Function 2.

The optional facility modifies or adds with software maintained. ME3 provides 6 varied optional functions:

0x01：USB Device
0x02：Intek Macro
0x04：High Speed and Precision
0x08：the 4th Axis
0x10：Dual axis drive
0x20： Rigid Tapping of maximum speed up to 1500 rpm

5.1 Facility 0x01 : USB Device

CNC programs usually save as a solid flash memory built in INCON-ME3, and we can also transmit CNC programs to controller through FTP.

In addition, INCON- ME3 provides optional facility to transmit CNC programs from USB to controller and execute programs in USB.

Pay attention to that most of USB can be read by INCON- ME3 but few of USB can't be done this by its compatibility.

The following conditions are satisfied with using USB:

- The format of USB has to be FAT or FAT32.
- Power supply of USB slot built in INCON- ME3 has limit 〔≤ 0.15 A〕 so that few of USB has power supply in addition.
- The file name must be 〝**.CNC**〞 as its extension name. The file name is not over 8 letters which can be English alphabet, number, 〝**-**〞 or 〝**_**〞 ,for example, **TEST_123.CNC** is a usable file name.
- There are two USB slots in ME3, but the slot used first just can be read.

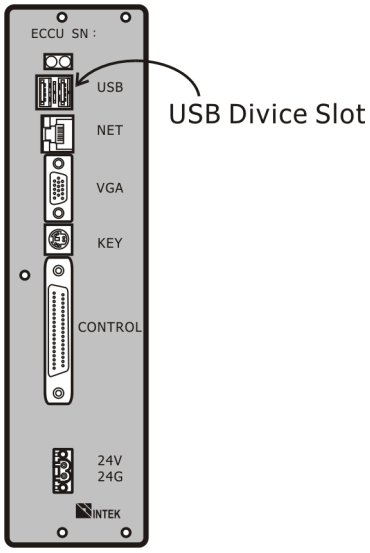


Figure 5-2 USB Device Slot

Insert USB to slot, and press 〝**F2→F→Open...**〞 to pop a 〝**Open a File**〞 dialog box , as below:

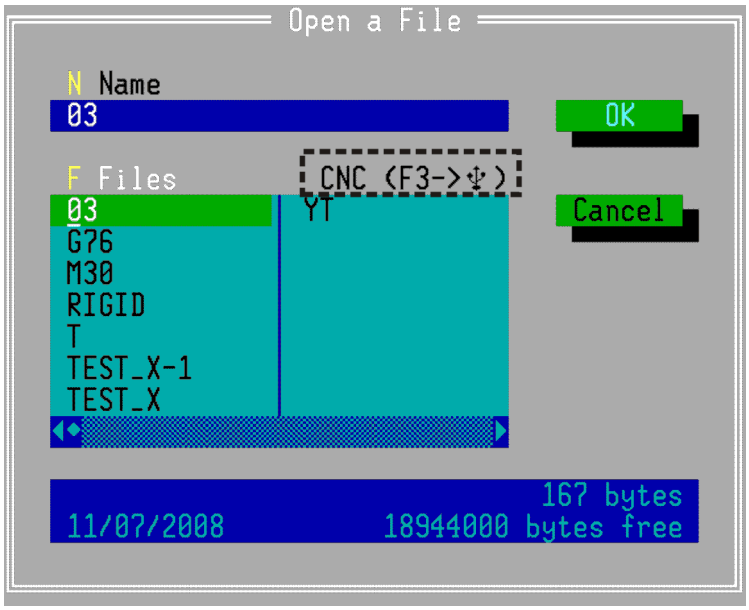


Figure 5-3 the 〞Open a File〞 dialog box shows there is USB to select〔press F3 to switch〕

Figure 5-3 shows the files in CNC controller. If the **USB Device** facility enables, the dialog box will appear 〝**CNC (F3->)**〞

When you press F3, it will open the CNC files in USB device.

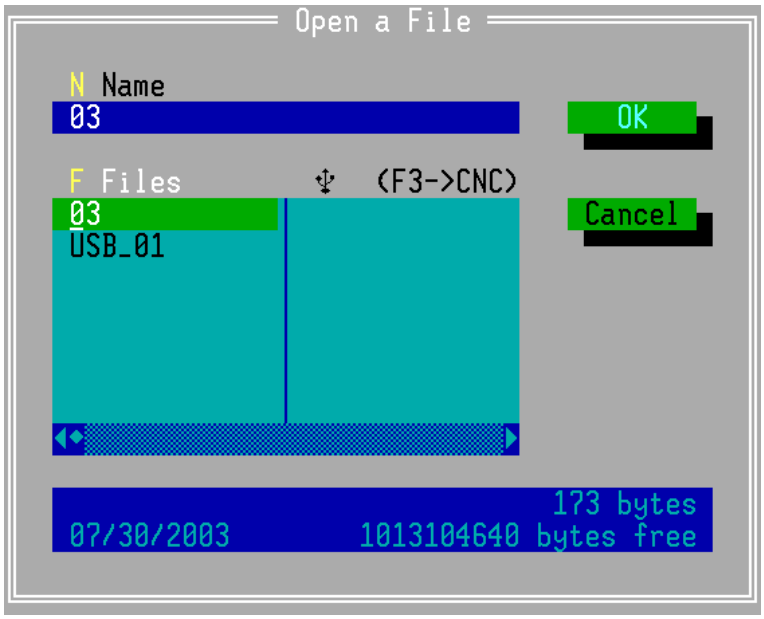


Figure 5-4 Save file in USB device

In the same way, 〝**F2→F→Save as...**〞, 〝**F2→F→Delete file...**〞 also use F3 to switch controller and USB device.

5.2 Facility 0x02 : Intek Macro

INTEK MACRO is designed for variable operation to make users program more flexibly.

The variable can be defined by program assign operation or user parameter. The variable also can be set or read by PLC .

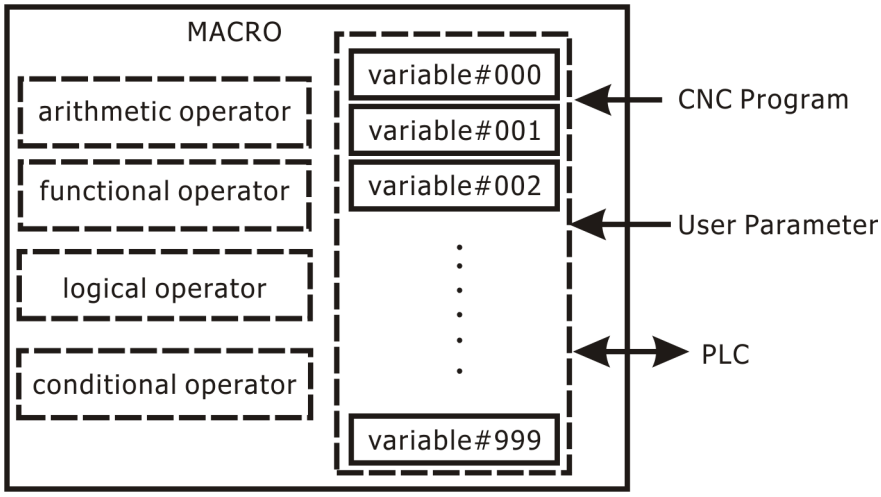


Figure 5-5 the relation between INTEK Macro and outside

There are 1000 variables, arithmetic operator, functional operator, logical operator, and conditional operator given on **INTEK MACRO**. The simplest expression is used under **INTEK MACRO**, so user does not need spend much time to learn program it.

5.2.1 Variable

Variable always starts with a leading code **#** and follows number. For example, **#210** is a No.210 variable ; **210** is not a variable.

INTEK MACRO provides 1000 variables from 0 to 999〔#0~#999〕. The numerical data of variable can be set by user parameter, program, or PLC.

- If variable is assigned by program, the expression shall be **#3=#4+#5**. Variable array pointer will transmit to PLC. PLC can set or read variables.
- When variables are set by user parameter, the dialog box will pop up on screen for setting.
- By using **Intek Macro** variables, The numerical data of variable in CNC program can be replaced by **Intek Macro** variables, for example:
  When **#1=500**, **#2=120**, **G01 X#2 F#1** is equal to **G01 X120 F500**

5.2.2 Arithmetic operator

INTEK MACRO fits for the four fundamental operations of arithmetic. Basic arithmetic operators, left and right parenthesis are also used on **INTEK MACRO**.

+ ; add
- ; subtract
***** ; multiply
/ ; divide
(; left parenthesis
) ; right parenthesis
= ; assign

Arithmetic operation example:

```
#3=4                    ;#3 to assign as 4
#5=#3+2                 ;#5 to assign as 4+2=6
#6=#5+(#3 - 1) ;#6 to assign as 6+(4-1) =9
```

Parenthesis must be a pair on expression otherwise error occurs.

```
#6=#5/(#7-#8    ;wrong expression
#6=#5/(#7-#8)   ;right expression
```

5.2.3 Functional Operator of Intek Macro

NTEK MACRO provides built-in functions, such as **SIN, SIND, COS, COSD, TAN, TAND, ATAN, SQRT**, and so on. Furthermore, the one related to trigonometric function with "**D**" means its unit is angle; for example, the angle of circle is 360°. The one without "**D**" means its unit is rad.; for example, a circle is about 6.2832 rad.

Format		Definition
SIN	SIND	sine
COS	COSD	cosine
TAN	TAND	tangent
ASIN	ASIND	arcsine
ACOS	ACOSD	arccosine
ATAN	ATAND	arctangent
ABS		absolute value
ROUND		integer
BROUND		round off
SQRT		mean square root

Example：

```
#4=SIN（#3）；
#5=COS（#2+1）；
#7=SQRT（#2+#1）；
#1=TAN（#4）；
#8=ATAN（0.9）；
```

5.2.4 Logical Operator

INTEK MACRO has logical operation facility; the result of logic operation is〝**Zero**〞or〝**None Zero**〞. The logical operation usually combines with conditional expression or loop. Expressions of logical operation are listed as below:

== ；equal
!= ；not equal
>= ；large equal
<= ；small equal
> ；large
< ；small
& ；intersection, and

| ;union、or

! ;not

Example 1：

```
#3=8;
#4=9;
#7=(#3==#4);
```

The value of **#7** is〝**Zero**〞.

```
#3=7;
#4=7;
#7=(#3==#4);
```

The value of **#7** is〝**None Zero**〞

Example 2：

```
#251=20;
#252=21;
#100=(#251!=#252);
```

The value of **#100** is〝**None Zero**〞

Example 3：

```
#250=10;
#253=4;
#252=8;
#100=9;
#121=0;
#120=1;
#40=((#250-#253)>#252)&(#100>=(#121+#120));
```

The value of **#40** is〝**Zero**〞.

5.2.5 Condition and Program Flow

INTEK MACRO has three loops for condition and program flow: **IF**, **IF... ELSE** and **WHILE**.

5.2.5.1 IF

Format：

IF variable P end line number

Example：

```
IF #100 P100;
G00 X#3 Y100;
G01 X100 Y#2 F#8;
N100 G53 X0 Z0;
G01 X300 F0.1;
M30;
```

The value of **#100** is〝**None Zero**〞, program will execute as below:

```
G00 X#3 Y100;
G01 X100 Y#2 F#8;
N100 G53 X0 Y0;
G01 X300 F0.1;
M30;
```

If variable **#100** is〝**Zero**〞, program will be executed as below:

```
G01 X300 F0.1;
M30;
```

5.2.5.2 IF … ELSE

Format:

IF variable P end line number A
N end line number A
ELSE P end line number B
N end line number B

Example:

```
IF #1 P100
G00 X800 Y1000;
G01 X1000 Y200 Z-10;
N100 G00 X0 Y0 Z0
ELSE P200
G00 X1600 Y1000;
G01 X1800 Y200 Z-10;
N200 G00 X0 Y0 Z0;
M30;
```

The value of **#1** is〝**None Zero**〞, program will execute as below:

```
G00 X800 Y1000;
G01 X1000 Y200 Z-10;
G00 X0 Y0 Z0;
M30;
```

The value of **#1** is〝**Zero**〞, program will execute as below:

```
G00 X1600 Y1000
G01 X1800 Y200 Z-10;
G00 X0 Y0 Z0;
M30;
```

5.2.5.3 WHILE

Format**:**

WHILE variable P end line number

Example:

```
#7=4;
WHILE #7 P80
G90 G01 Z-10 F0.8;
G91 G01 X100 F1.0;
#7=#7-1;
N80 G90 G01 Z0 F0.8;
```

Program will execute as below:

```
G90 G01 Z-10 F0.8;
G91 G01 X100 F1.0;
G90 G01 Z0 F0.8;
G90 G01 Z-10 F0.8;
G91 G01 X100 F1.0;
G90 G01 Z0 F0.8;
G90 G01 Z-10 F0.8;
G91 G01 X100 F1.0;
G90 G01 Z0 F0.8;
G90 G01 Z-10 F0.8;
G91 G01 X100 F1.0;
G90 G01 Z0 F0.8;
```

5.2.6 High Speed Macro

G10.60：High Speed Macro enable
G11.60：High Speed Macro disable

5.2.7 Macro application example

Pocket Circular :

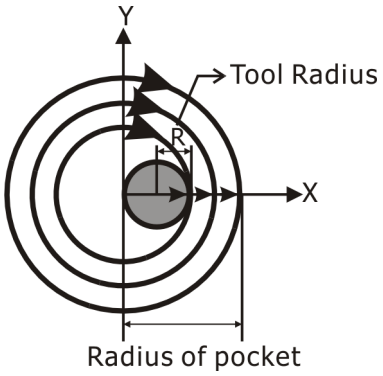


Figure 5-6 Pocket Circular

Radius of pocket is assigned by users on account of variable. Radius of tool is also assigned by users on account of variable. Flowchart is as below:

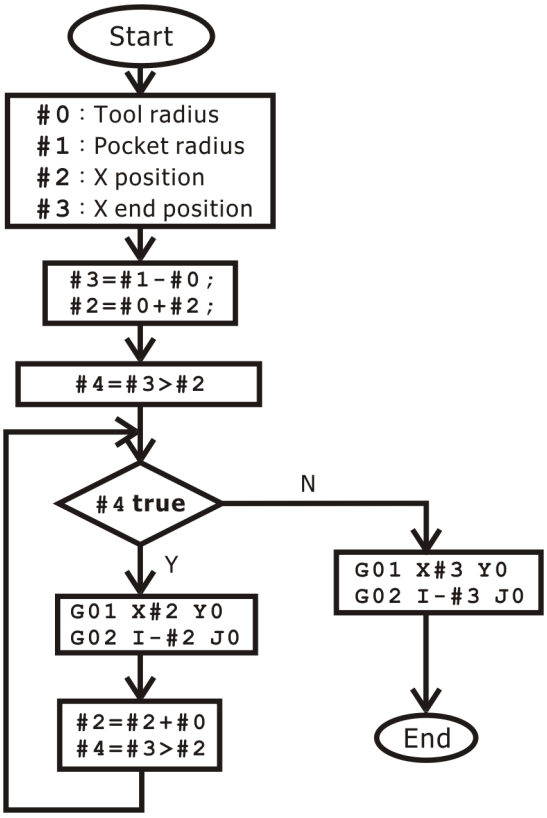


Figure 5-7 Flowchart of pocket circular

Pocket Circular Program:

```
#2=0;
#3=#1-#0;
#2=#0+#2;
#4=#3>#2;
WHILE #4 P10;
G01 X#2 Y0 F400;
G02 I-#2 J0 F400;
#2=#2+#0;
N10 #4=#3>#2;
G01 X#3 Y0 F400;
G02 I-#3 J0 F400;
M30;
```

5.3 Facility 0x04 : High Speed and High Precision

When INCON-ME3 provides optional facility: high speed and high precision cutting, the parameters about high speed and high precision cutting enable.

5.4 Facility 0x08 : the 4th Axis

When this optional facility is selected, INCON-ME3 has the 4th Axis.

5.5 Facility 0x10 : Dual Axis drive

When this optional facility is selected, the 4th axis is treated as a teamwork follow axis and move with the followed axis at the same time.

5.6 Facility 0x20 : Rigid Tapping

When this optional facility is selected, ME3 can execute rigid tapping of maximum speed up to 1500 rpm.