SYNTEC Incorporation

LATHE (900T)
User Guide

By：syntec
Ver：8.00
Date：2001/07/01

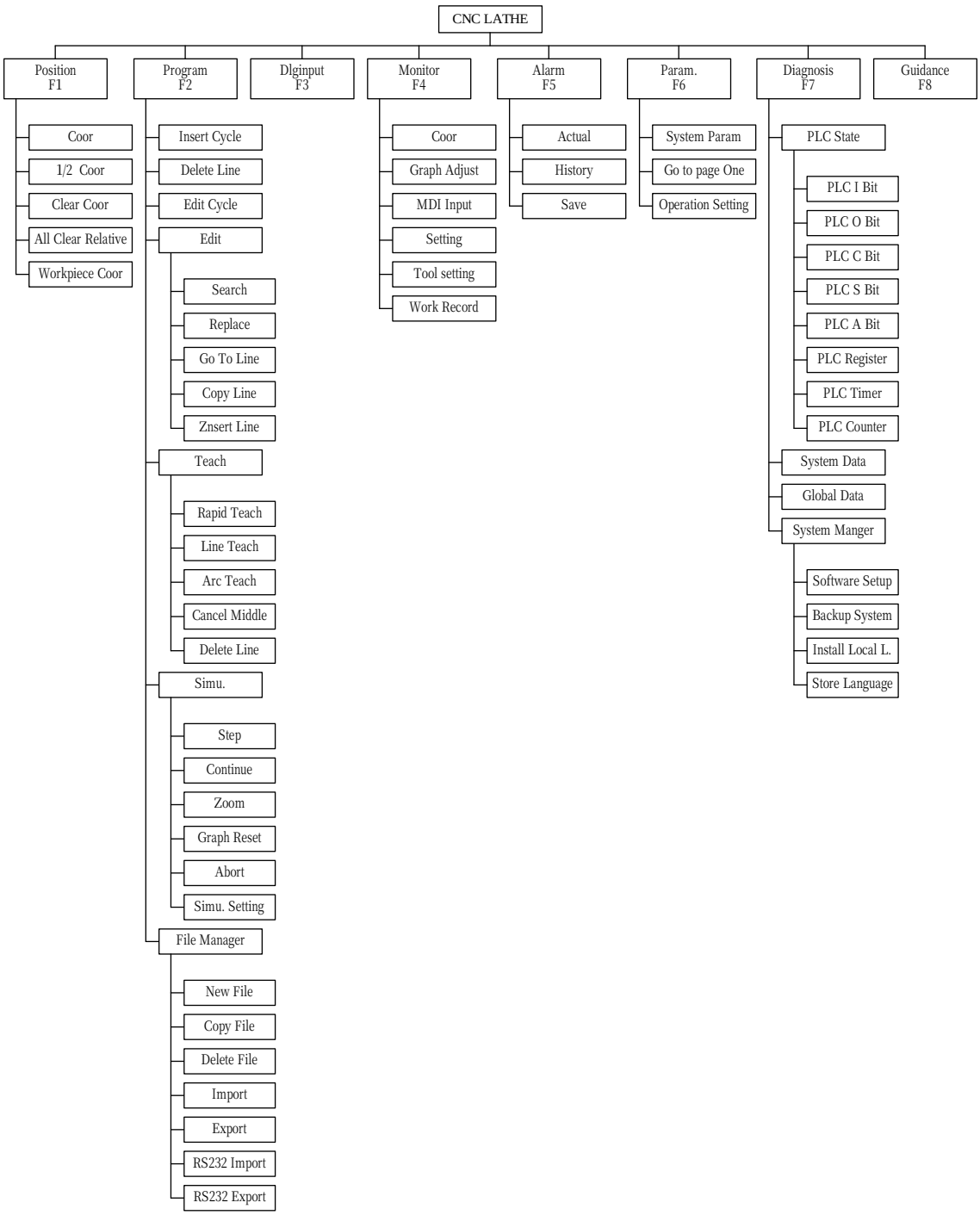
版本更新記錄

項次	更改內容紀錄	更改日期	作者	更改後版本
01	初版定稿	2001/07/01		V8.0

Table Of Contents

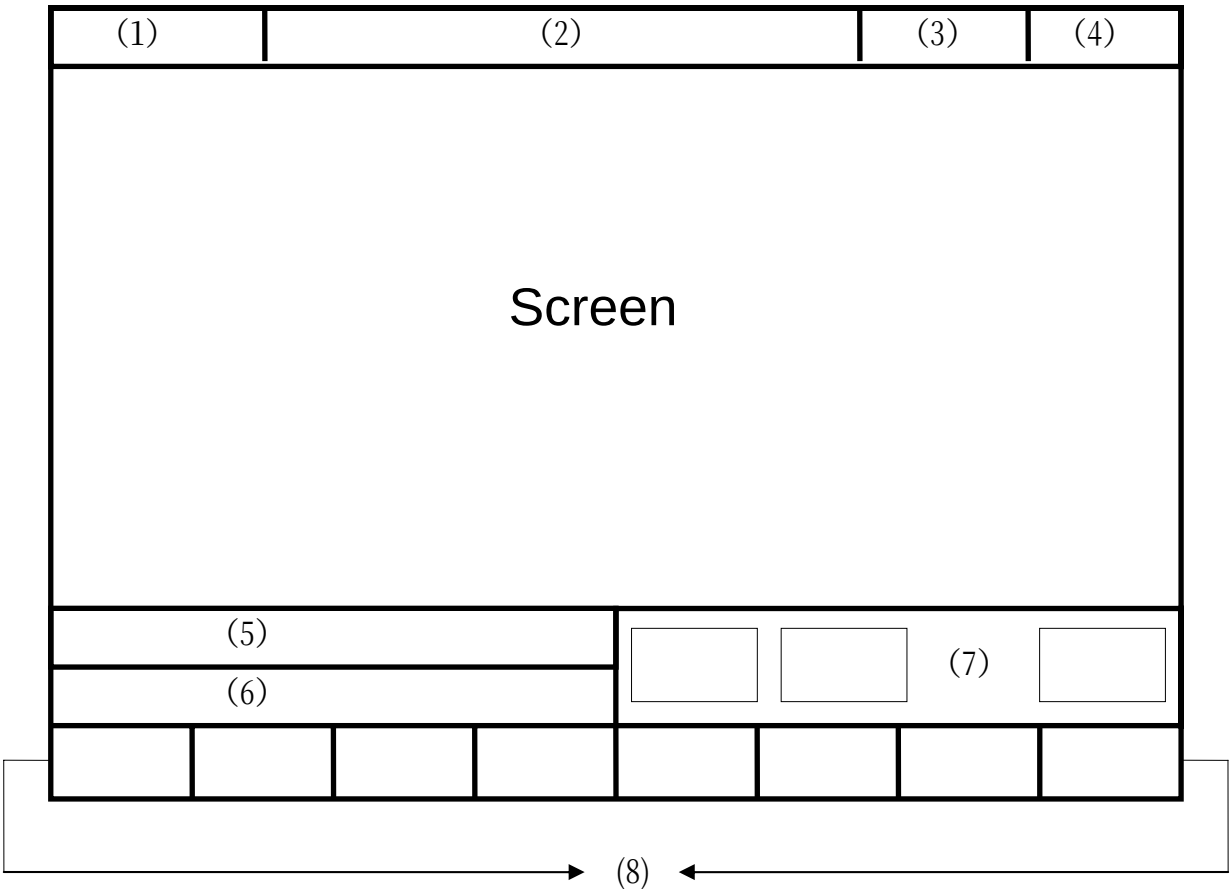
Chapter 1 Lathe Controller Interface

1.1 SNC System Configuration



1.2 Screen Sections

The are shown as followings:

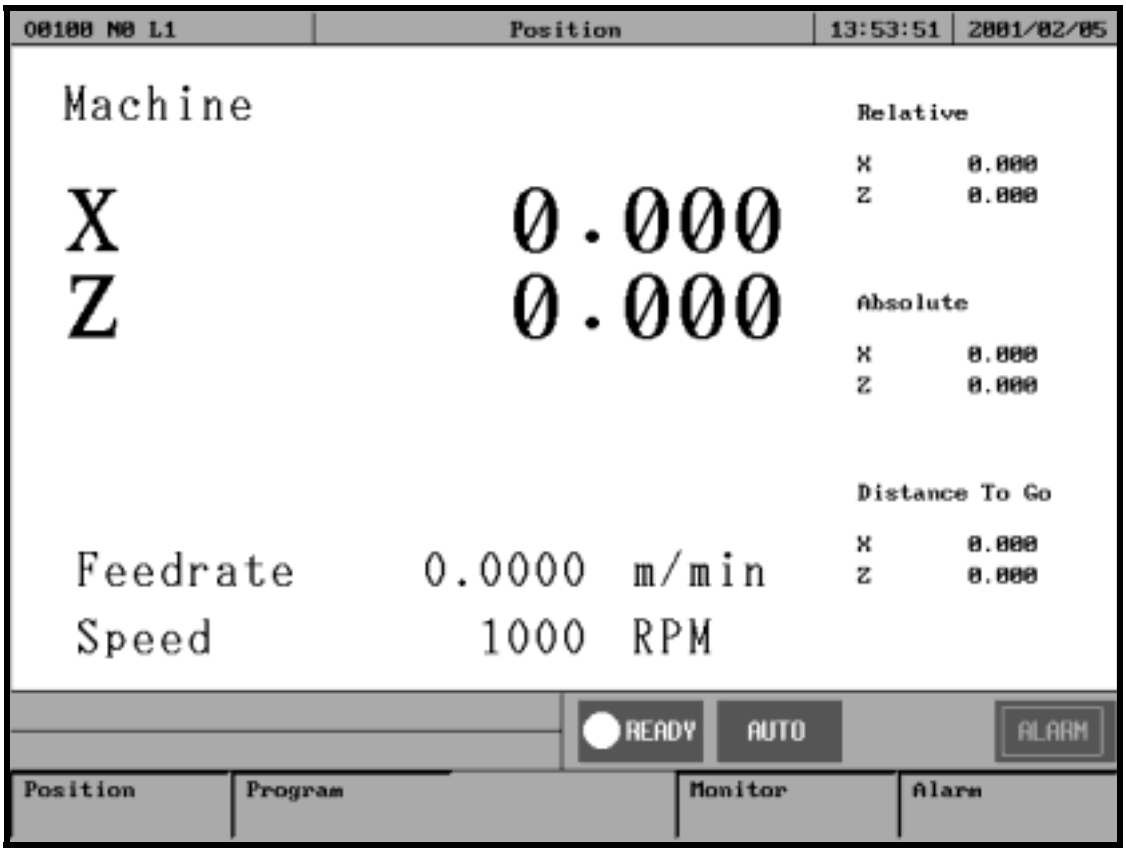


Meanings For Fields on the Display:

（1）Program Number
（2）Title
（3）Time
（4）Date
（5）Data Input
（6）Hint
（7）Status
（8）Function Key Switch

1.3 Main Menu Selections

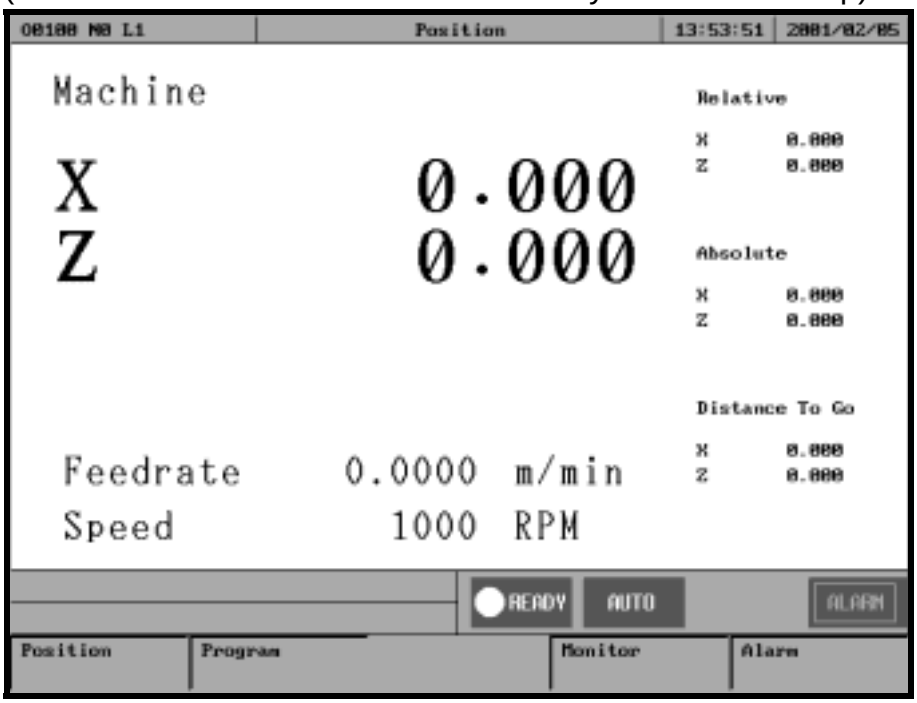
The following diagram is the main menu selections for SNC Lathe controller. To operate SNC Lathe controller, users simply make the selections by pressing function keys, F1 F5 located on the bottom of the screen.



1.3.1 F1：Position

This selection displays coordinate settings of current position. It can also be used to reset the position of relative coordinate. Pressing function key, F1, under the main menu to enter this selection

(Note：This is the first screen when the system is booted up)



Meaning of fields on the display------

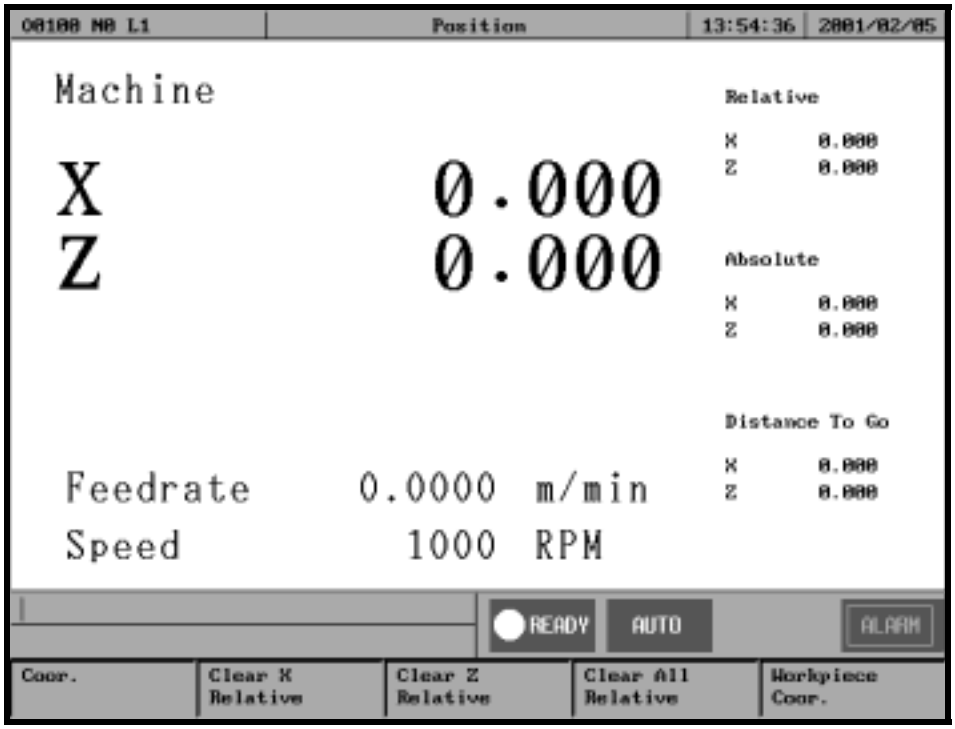
- X：X axis coordinate.
- Y：Y axis coordinate.
- Z：Z axis coordinate.
- Feedrate：Feedrate of cutting tool at each machining, mm per minute (mm/min).
- Spindle：RPM of spindle speed.
- Machine(Relative coordinate of working platform)
  The current position of cutting tool relative to working platform is shown as machine coordinate on the display.
- Relative
  The current position of cutting tool relative to the previous location.
- Absolute(Programming Coordinate)
  The current position of the origin of user defined coordinate is shown as an absolute position on the display.
- Distance To Go：The distance for the cutting tool to move to the next position in both position(+) and negative(-) direction.

Function key selections：

1.3.1.1 F1：Coor.

Function：Switch Coordinate Display。

Operation：Under the Position submenu, whenever users press F1 key, the values and coordinate on the left corner of the display will toggle among the four different coordinates with bigger fonts as shown in the following figure:



1.3.1.2 F2：1/2 Coor.

Function: Set the center point of work piece as coordinate origin.

Operation: Under the Position submenu, when the message line shows “X Input”(or “Y Input” or “Z Input”), press “F2 1/2 Coordinating” and the origin of the coordinate will move to the center point of work piece.

1.3.1.3 F3：Clear Coor.

Function：Reset the value of X(or Y or Z) axis relative coordinate to zero.
(No effect on other axis)

Operation：Under the Position submenu, when the message line shows “X Input”(or “Y Input” or “Z Input”), press F3 to reset the value of X(or Y or Z) axis relative coordinate to zero.

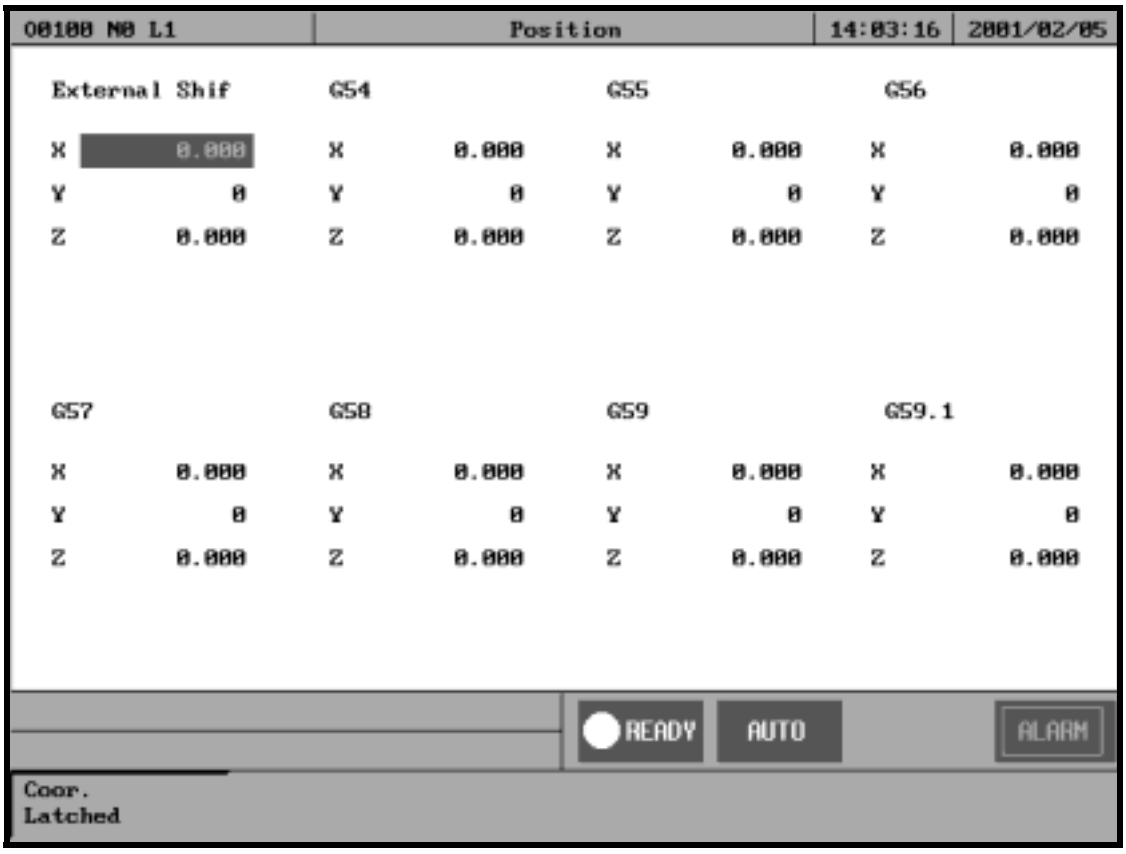
1.3.1.4 F4：All Clear Relative

Function：Reset XYZ relative coordinate to zero.(No effect on other coordinates)

Operation ：Under the Position submenu, pressing “F4” will reset XYZ relative coordinate to zero.

1.3.1.5 F5：Worlpiece Coor.

Function：Relative to machine coordinate setting for G54~G59.

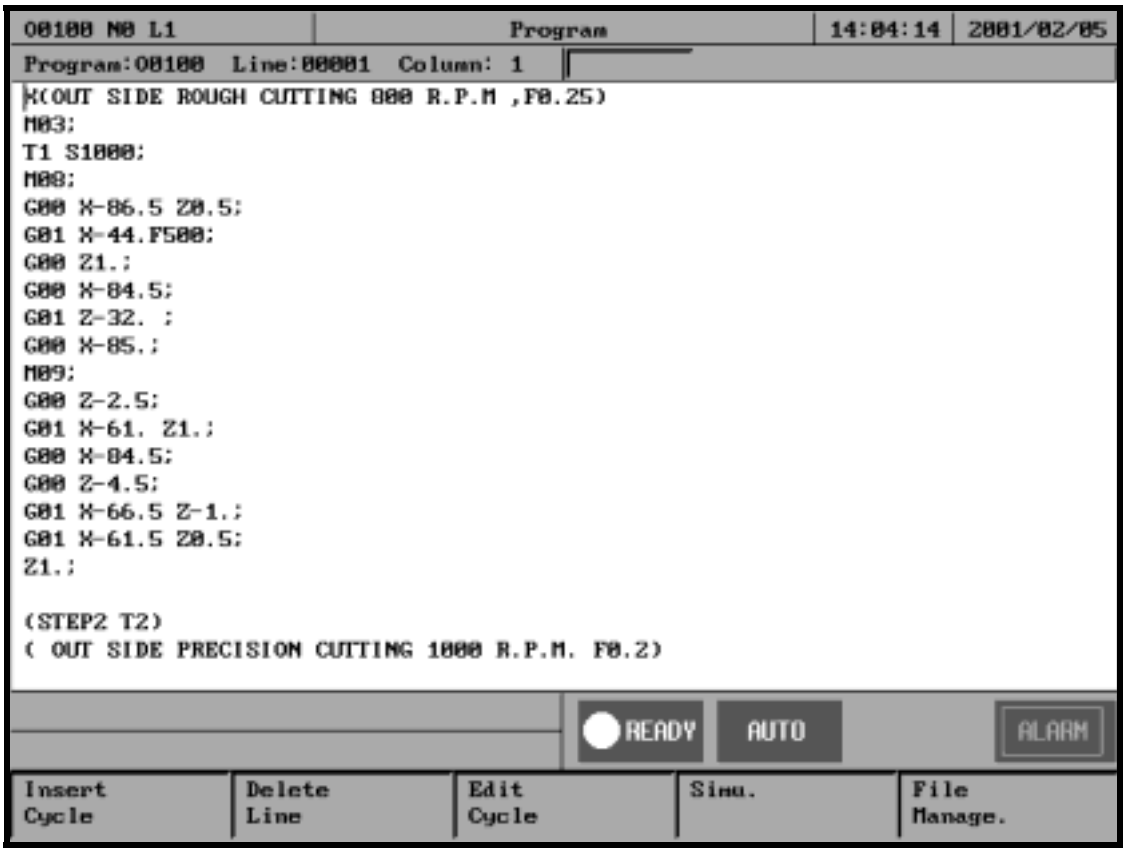


Operation：Under the Position submenu, press "F5 " key and the following screen will show up. By pressing "F1 " key, then users can begin to set the auto machine coordinate settings of G54~G59 one by one. (The system needs to be in manual date input (MDI) mode.)

1. “External Shift” : operator can set the all coordinate G54..G59 at the same time .
2. CNC default G54 ,if user don’t set any G54..G59 in the NC file

1.3.2 F2：Program

This selection provides users with program file management and editing. With a full screen editor, users can use arrow keys（↑、↓、←、→）to move the cursor to anywhere on the screen for editing purpose. Pressing F2 under the main menu to enter this selection. The full screen editor is shown as follows:

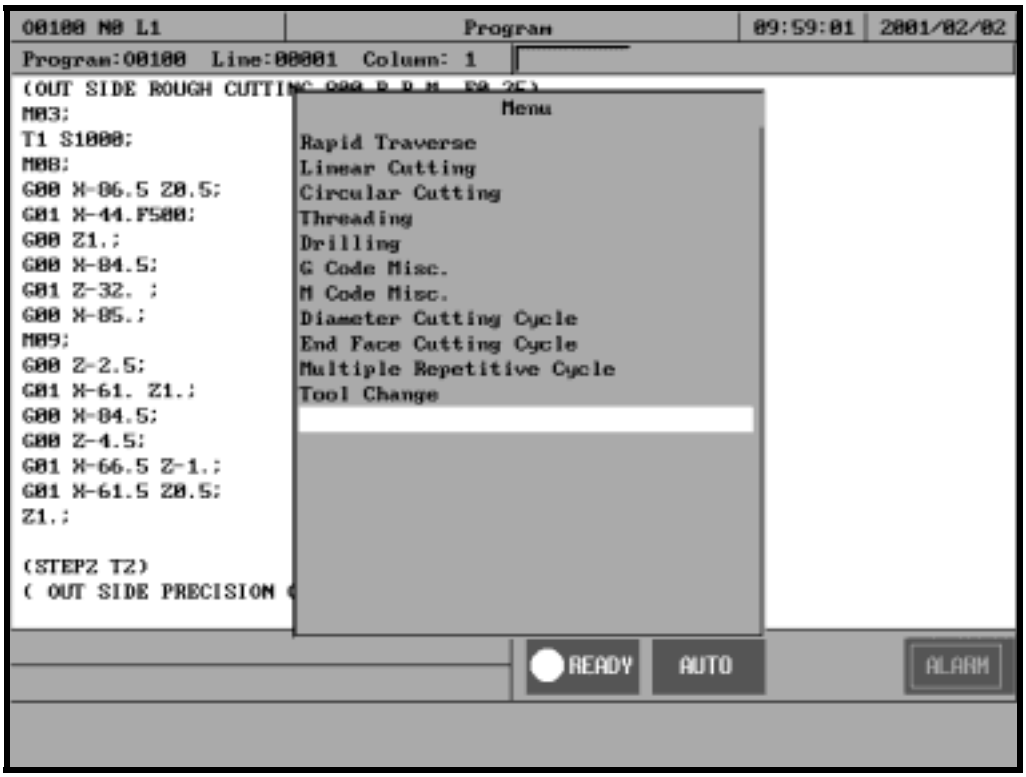


Program Sub menu Key Selections:

1.3.2.1 F1：Insert Cycle

Function：Insert a block or cycle by conversation

Operation：Under Program submenu, press F1 to insert a line or cycle before cursor position.(See Graphic Input Interface(900TE) User Guide)



1.3.2.2 F2：Delete Line

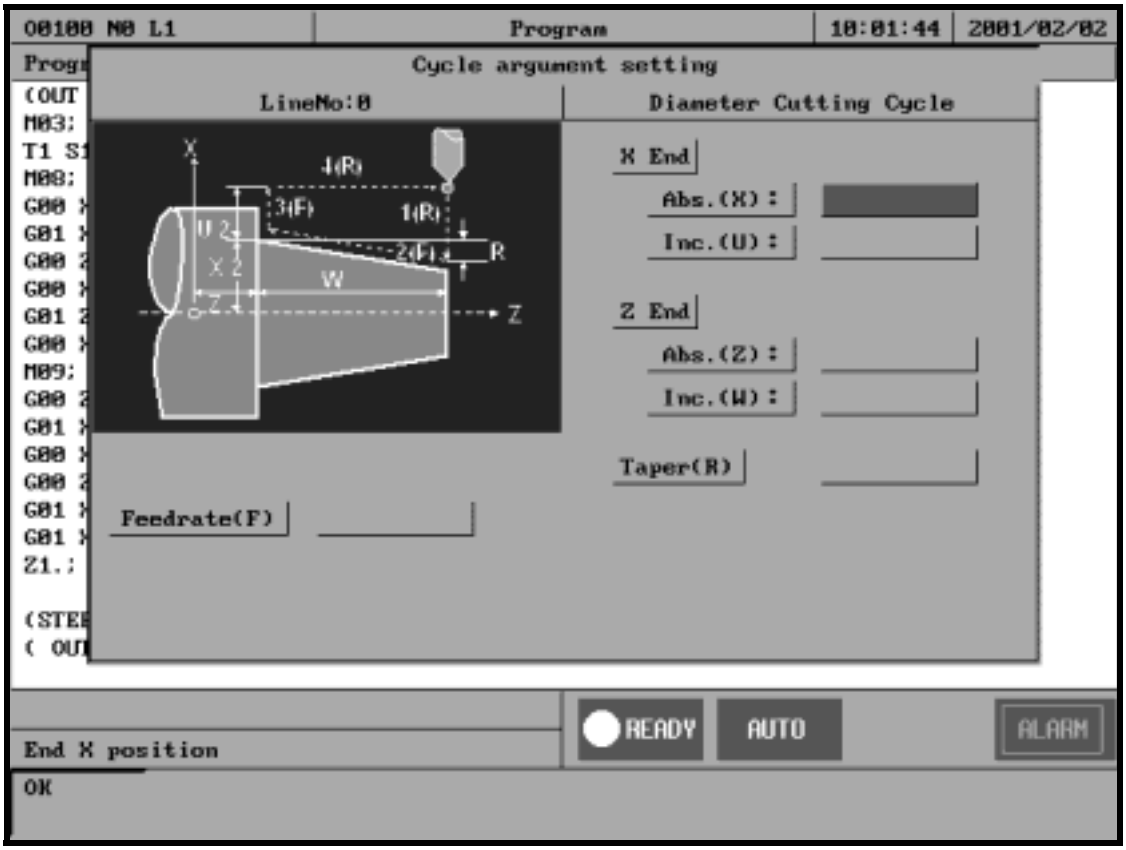
Function：Delete a line at cursor position.

operation: Under Program submenu, press F2 to delete a line where the cursor is located.

1.3.2.3 F3：Edit Cycle

Function：Edit an old block or cycle by conversation input

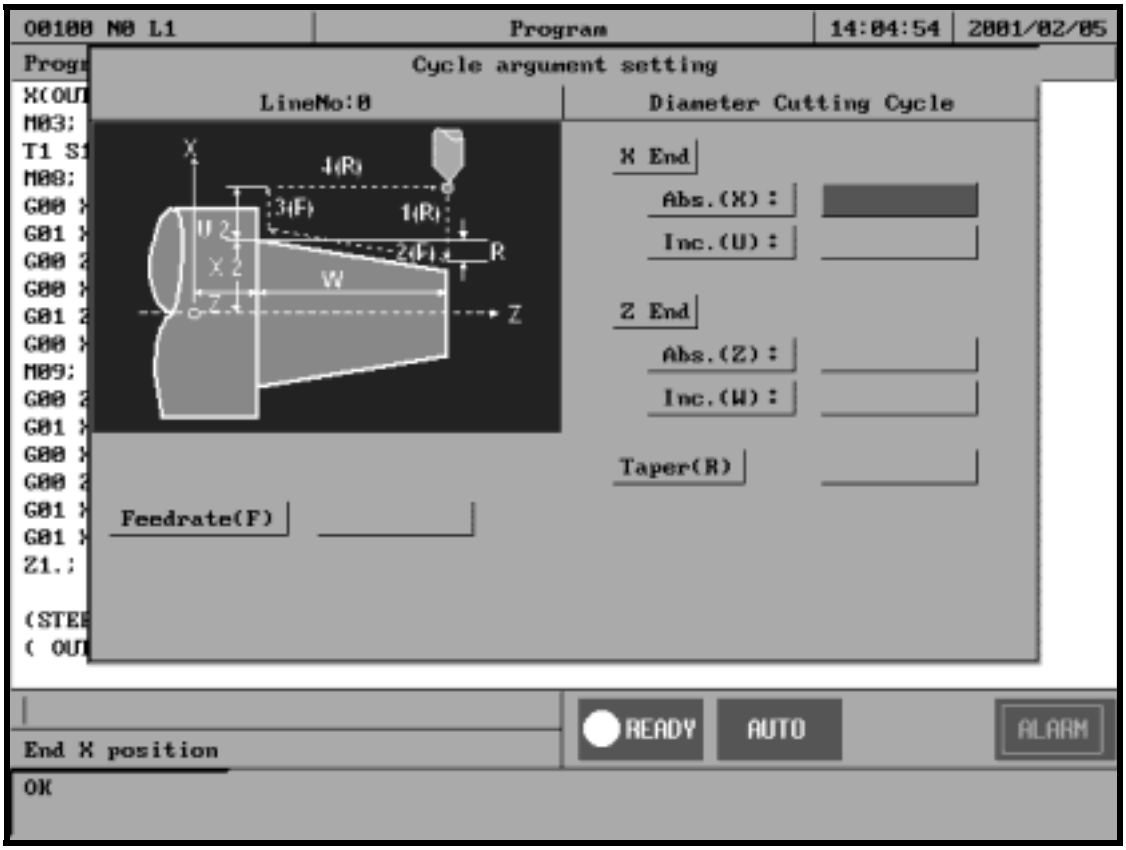
operation: Under Program submenu, press F3 to edit a line where the cursor is located. .(See Graphic Input Interface(900TE) User Guide)



1.3.2.4 F6：EDIT

Function : Edit sub function “ Search “ “Replace” “Goto line” “Copy line””Insert line”

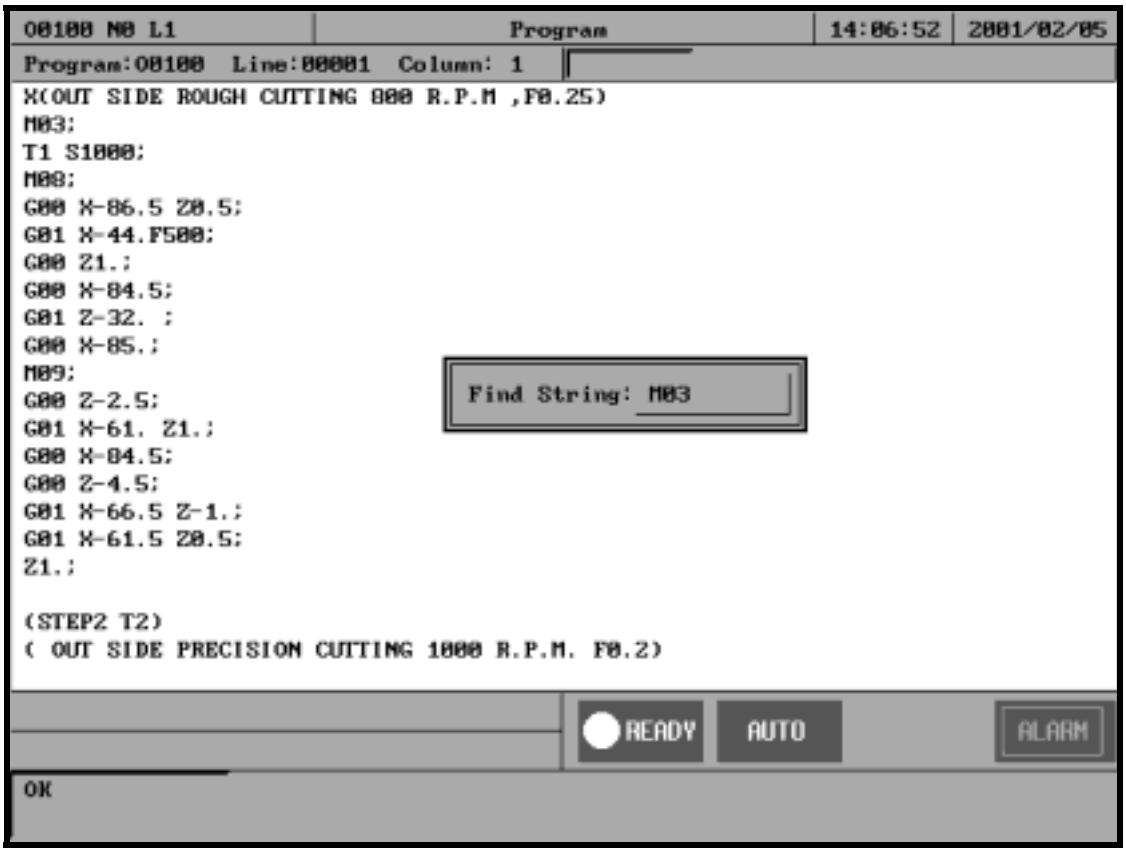
Operation : user can use this menu for more edit sub function



1.3.2.4.1 EDIT sub function “F1 : Search”

Function：Search String。

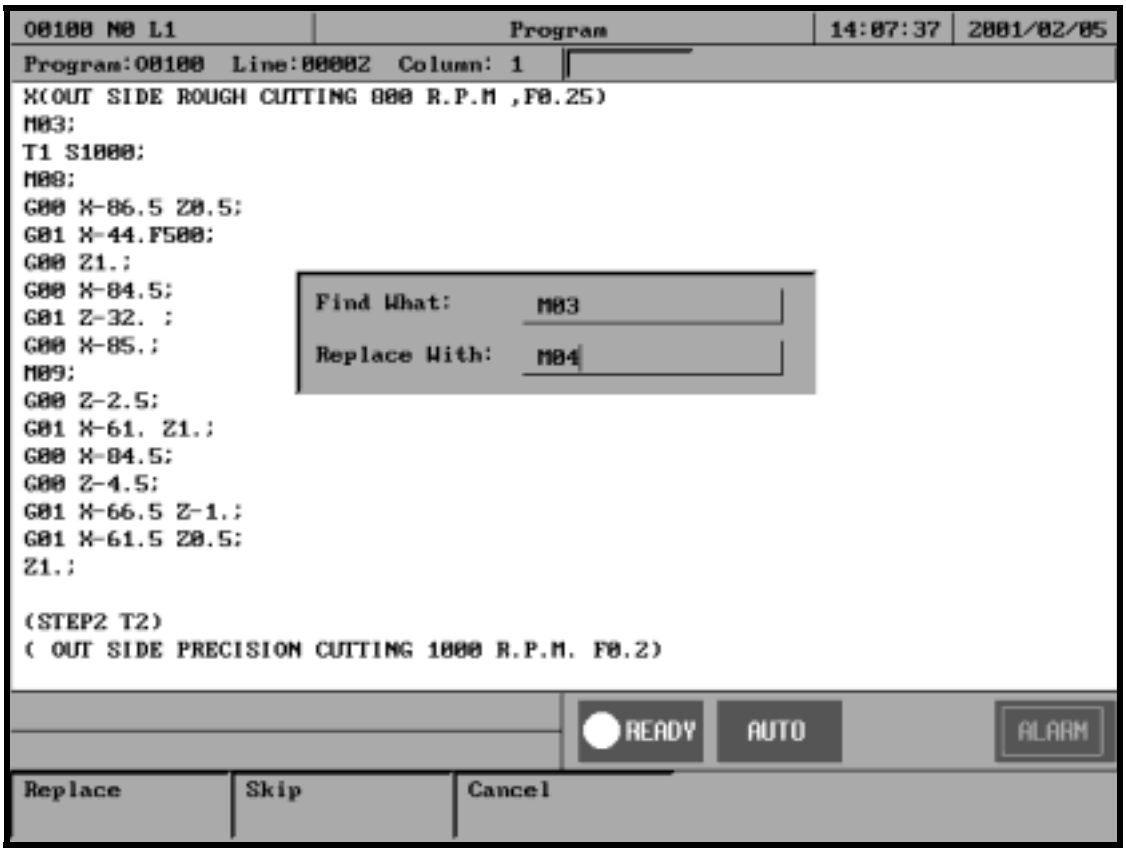
Operation：Under Program submenu, press F6”EDIT” and then F1 “Search” to search string. An dialog box will pop up asking users to input a string as shown in the following figure. After keying in a string, press F1 to start searching.



1.3.2.4.2 EDIT sub function “F2 : Replace”

Function：Replace String.

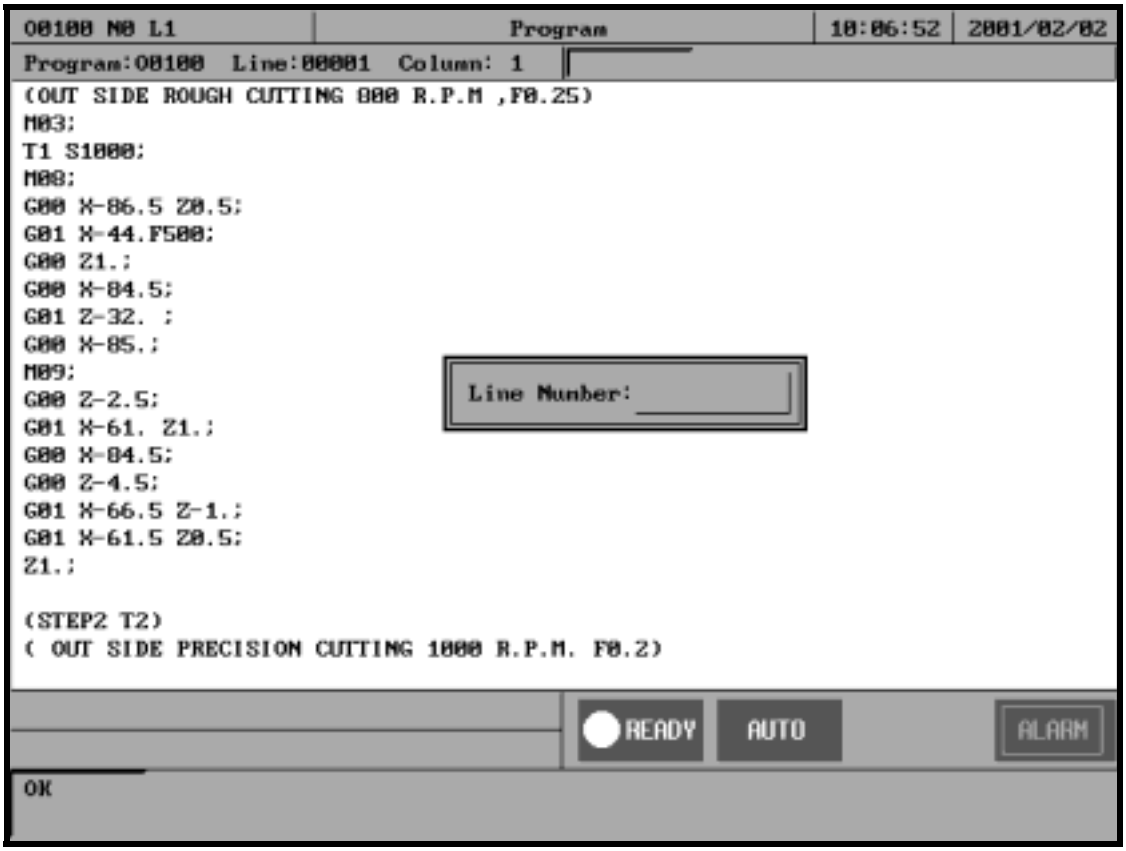
Operation：Under Program submenu, press F6 “EDIT” and then F2”Replace” to replace string. An dialog box will pop up asking users to input the replacing string and the new string as shown in the following figure. After keying in a string, press F1 to start replacing.



1.3.2.4.3 EDIT sub function “F3 : Go To Line”

Function：Go to a line number

Operation：Under Program submenu, press F6”EDIT” and then F3”GOTO line” to go to the line number. An dialog box will pop up asking users to input a line number. After keying in a number, press F1 to go to the desired line.



1.3.2.4.4 EDIT sub function “F4 : copy line”

Function：copy a line from current cursor to next line

Operation：Under Program submenu, press F5 “EDIT” and then F4”Copy line” to go to the next line.

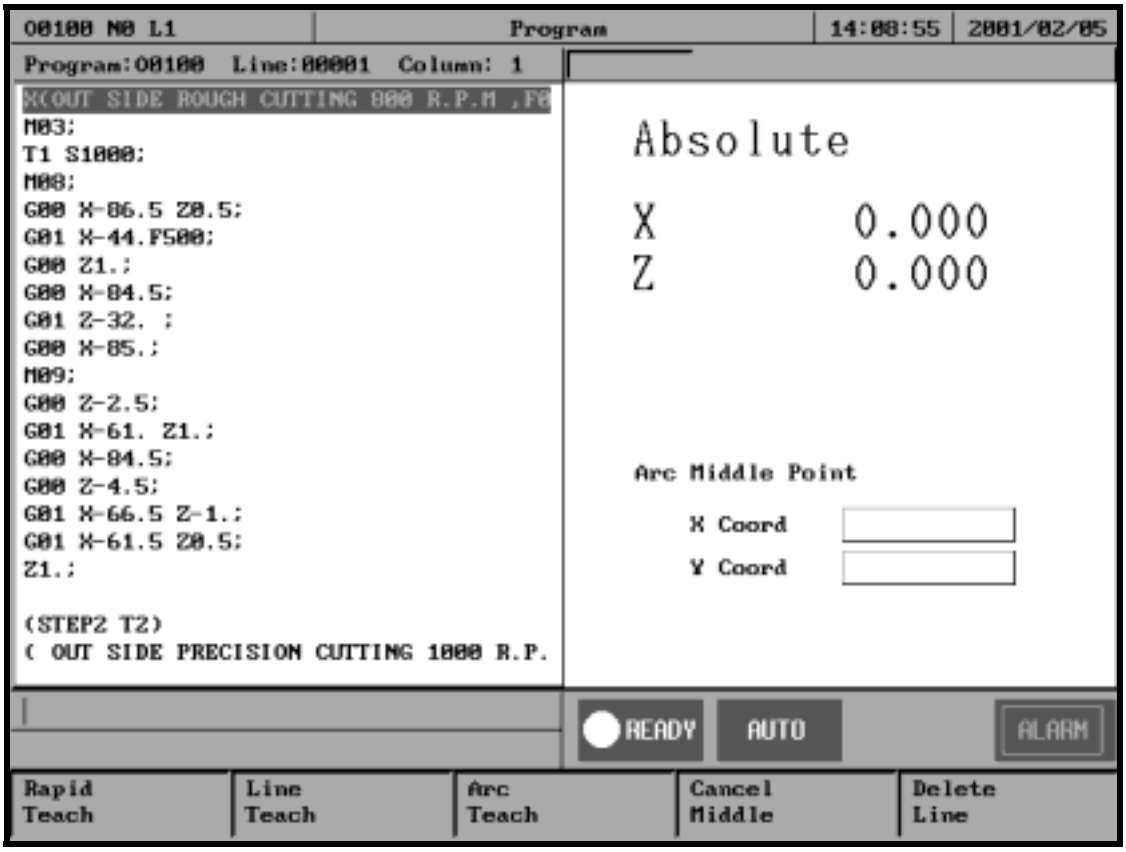
1.3.2.4.5 EDIT sub function “F5 : Insert line”

Function：Insert a space line above current cursor line

Operation：Under Program submenu, press F5 “EDIT” and then F5”Insert line” to Insert a new space line

1.3.2.5 F7：Teach

Function: Teach now absolute coordinate to NC files
Operation：Under Program submenu, press F6”Teach”



Teach sub Key Selections：

1.3.2.5.1 F1：rapid Teach

Function: Add “G00” code to NC files ,G00 to current absolute coordinate,

1.3.2.5.2 F2：linear Teach

Function: Add “G01” code to NC files ,G01 to current absolute coordinate,

1.3.2.5.3 F3：Arc Teach

Function: Add “G02” or “G03” code to NC files
1st time press this key “arc teach “ ,CNC auto put current value to Arc middle point
2nd time press this key “arc teach “ ,CNC auto calculate G02 or G03 ,and filled the complete code to NC files

1.3.2.5.4 F4：Cancel Middle

Function: When arc teach ,user can use this key to abort middle point teach .

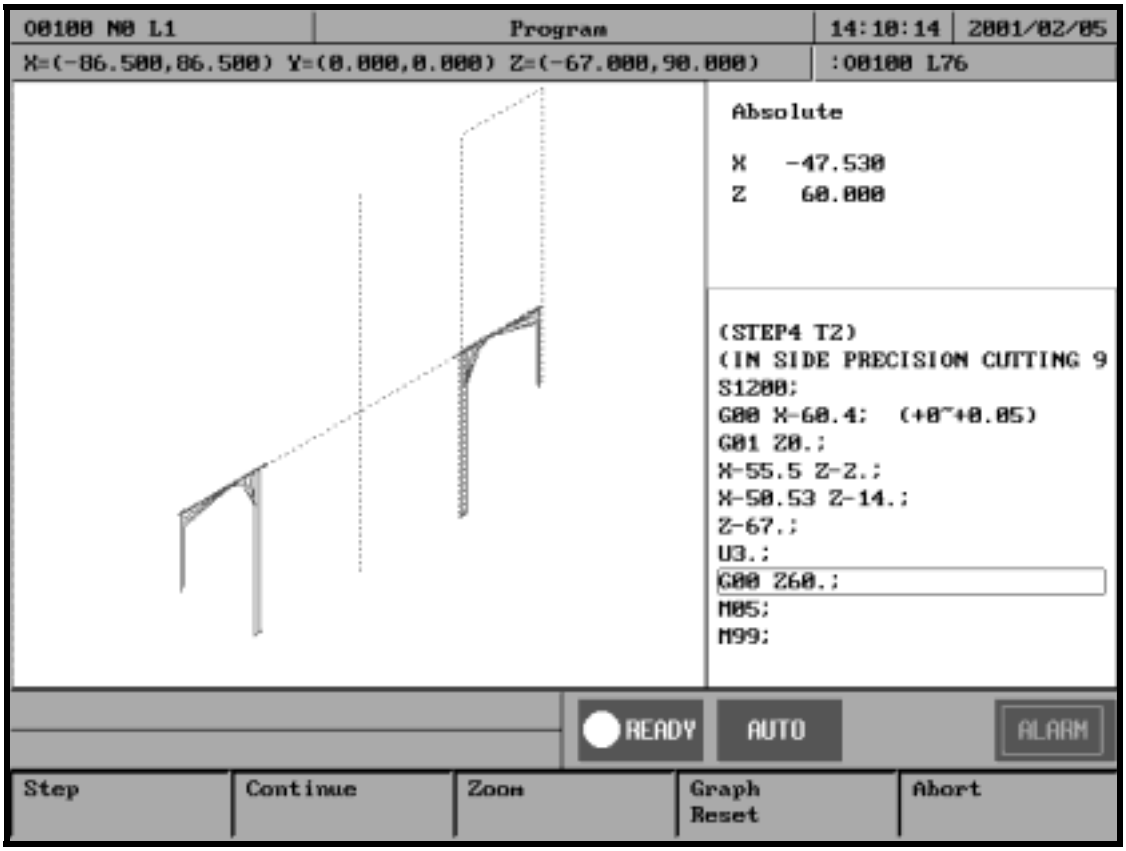
1.3.2.5.5 F5：Delete Line

Function: When user use Teach function , user can use this key “Delete line “ to delete line

1.3.2.6 F4：Simulation

Function: Simulating the workpiece program can prove the accuracy of the editing program.

Operation：Under Program submenu, press F4



F7 “simulation” sub Key Selections：

1.3.2.6.1 F1： STEP

Function: To simulate NC files STEP by STEP

Operation: Under Program submenu, press F4”Simulation” and then F1”Step” .The operator can use this function ,to check NC file Step by STEP

1.3.2.6.2 F2：Continue

Function: To simulation NC file one time .

Operation: Under Program submenu, press F4 “Simulation” and then F2”Continue” .The operator can use this function to check NC file whole picture ,when push button.

1.3.2.6.3 F3：Zoom

Function: To enlarge the workpiece graph.

Operation: Under Program submenu, press F4 and then F3.The operator can use the "←", "↑","→", "↓" cursor to move the frame to the determined area. And use "PageUp" "PageDn" to Enlarge this area .

1.3.2.6.4 F5：Graph reset

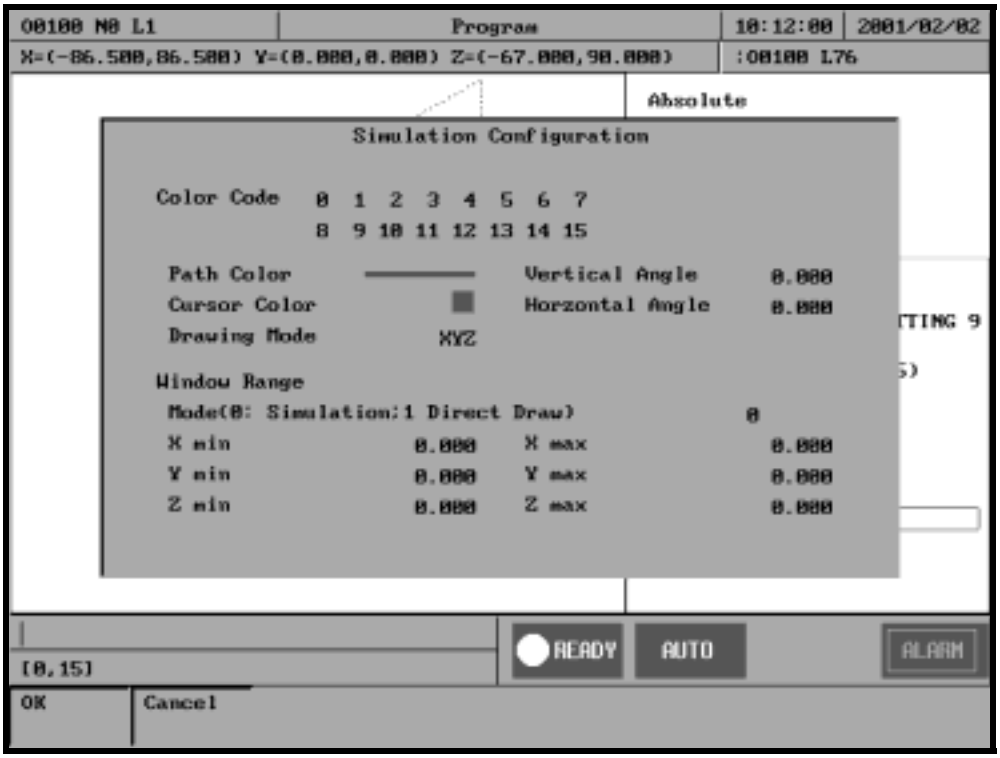
Function: To recover the zoomed workpiece graph.

1.3.2.6.5 F6：Abort

Function: To abort simulation action

1.3.2.6.6 F8：Simu. Setting

Function: To set simulation parameter



Simulation Parameter discription :

Path color : user can select cutting path color by this parameter

Cursor color : user can select cutting point color by this parameter

Drawing mode : user can select simulation plane by this parameter

Vertical / Horizontal angle : when XYZ drawing mode ,user can select 3D View angle by these 2 parameter

Window range :

Mode (0: simulation ,1: direct draw)

0 : When operator change his operation main screen to F4”Monitor” ,CNC would automatically simulation at that screen

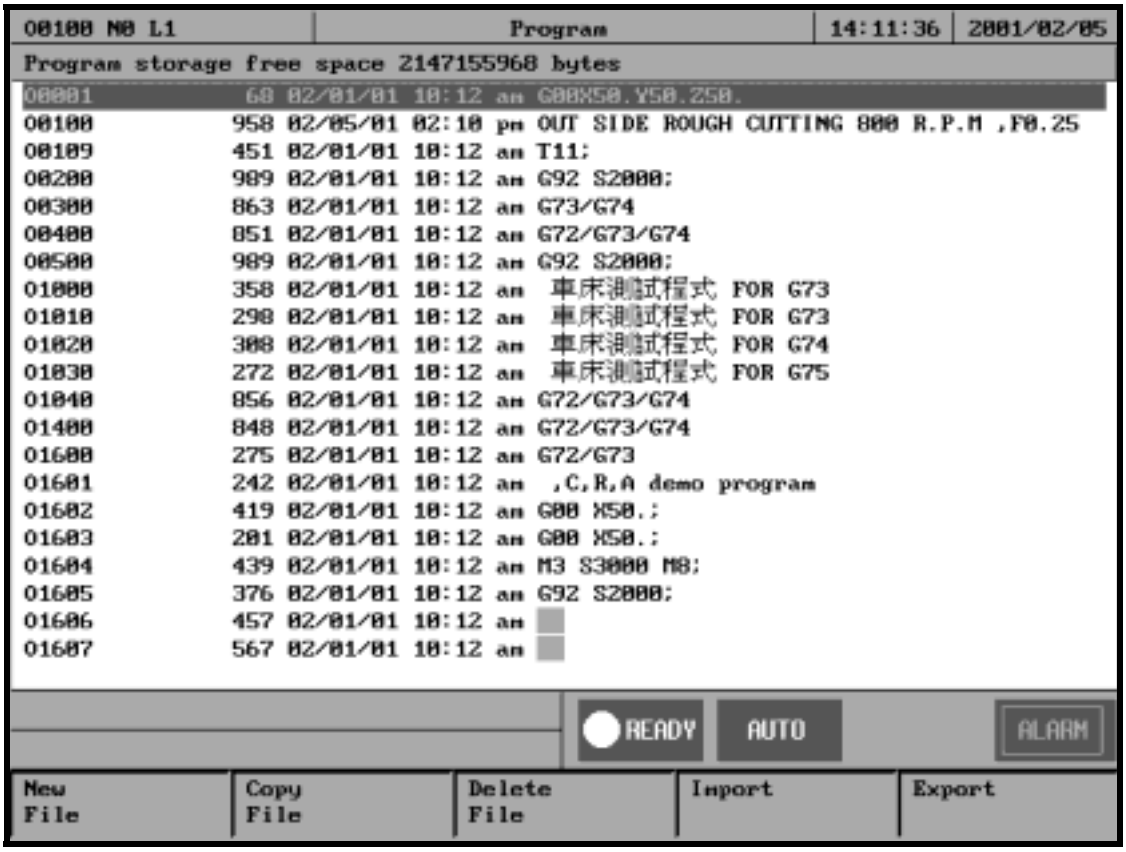
1 : When operator change his operation main screen to F4”Monitor” ,CNC would not simulation at that screen but direct drawing the cutting cursor .

Xmin/Xmax ,Ymin/Ymax ,Zmin/Zmax :

When “direct draw” mode ,operator must set draw window by these parameter ,the best way : after simulation use simulation result which is located at the top of this screen to X,Y,Z range .

1.3.2.7 F5：File Manager

Under Program submenu, press F5 and the following diagram will show up. Users can use arrow keys（↑、↓）to select file to be edited. After pressing 【ENTER】, content of the file will show up on the screen



Key Selections：

1.3.2.7.1 F1：New File

Operation：Step 1: A dialog box will prompt users with “New File ”. Type in the new file name and press 【ENTER】.

Step 2: An empty screen shows up waiting users to type in a new program.

I.3.2.7.2 F2：Copy File

Operation: After pressing F2, a dialog box will prompt users to type in a file name and press 【ENTER】. The current file is then copy to the hard disk with a different file name.

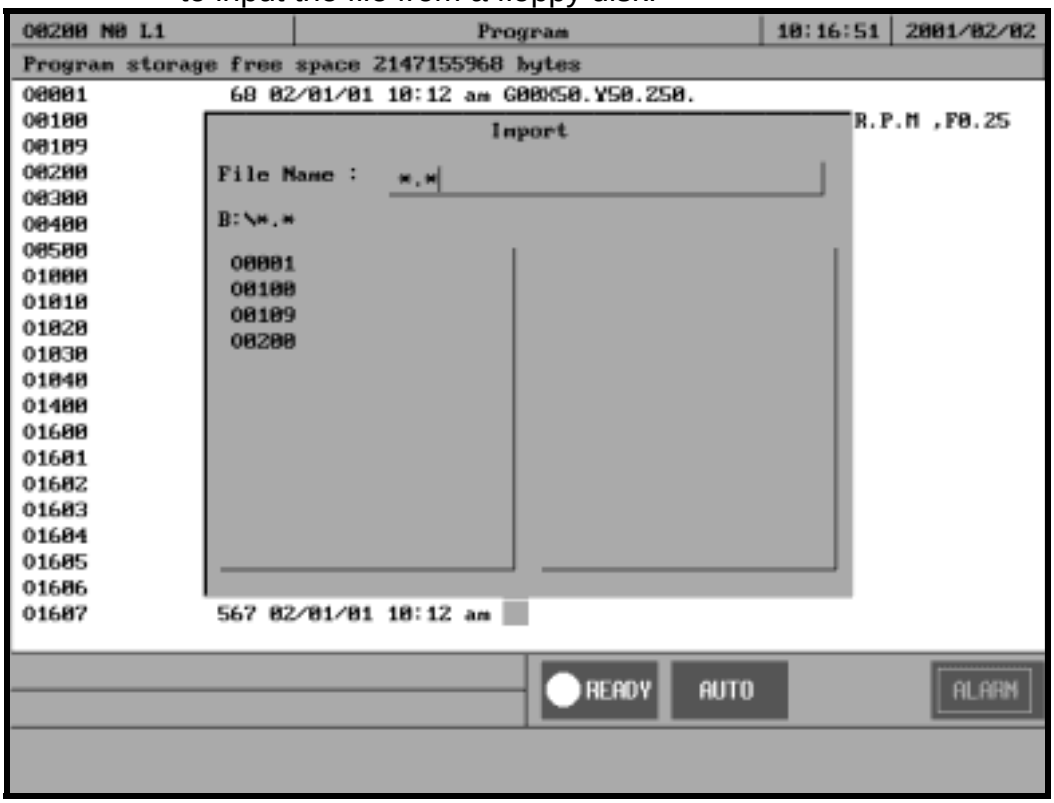
1.3.2.7.3 F3：Delete File

Operation: Select a file to be delete by pressing （↑、↓）. A dialog box will pop up to confirm this operation.

1.3.2.7.4 F4：Import

Function：Input file from floppy.

Operation：Insert a disk to the floppy drive and then press F4. Select a file name by pressing（↑、↓、←、→）. Press【ENTER】to input the file from a floppy disk.



P.S.

1. **FLOPPY DISK FILE FORMAT IS ASCI CODE**
2. **SYNTEC CNC ALSO CAN ACCEPT *.ZIP FORMAT ,WHEN IMPORT FROM FLOPPY DISK ,CNC WOULD UNZIP AUTOMATICALLY**
3. **IF NC FILE TOO BIG ,OPERATOR CAN USE MORE THAN ONE FLOPPY DISK TO INPORT NC FILE , SEPARATES A BIG FILE TO SOME FLOPPY DISKS ,USE THE SAME FILE NAME ,THEN IMPORT THIS FILE DISK BY DISK ,OPERATOR CHOOSE APPEND BUT NOT OVERWRITE ,IT IS VERY EASY TO INSATLL A BIG FILE**
4. **IF OPERATOR INSTALL EHTERNET ,IT IS MORE EASY TO INSTALL A BIG FILE FROM NET .**

1.3.2.7.5 F5：Export

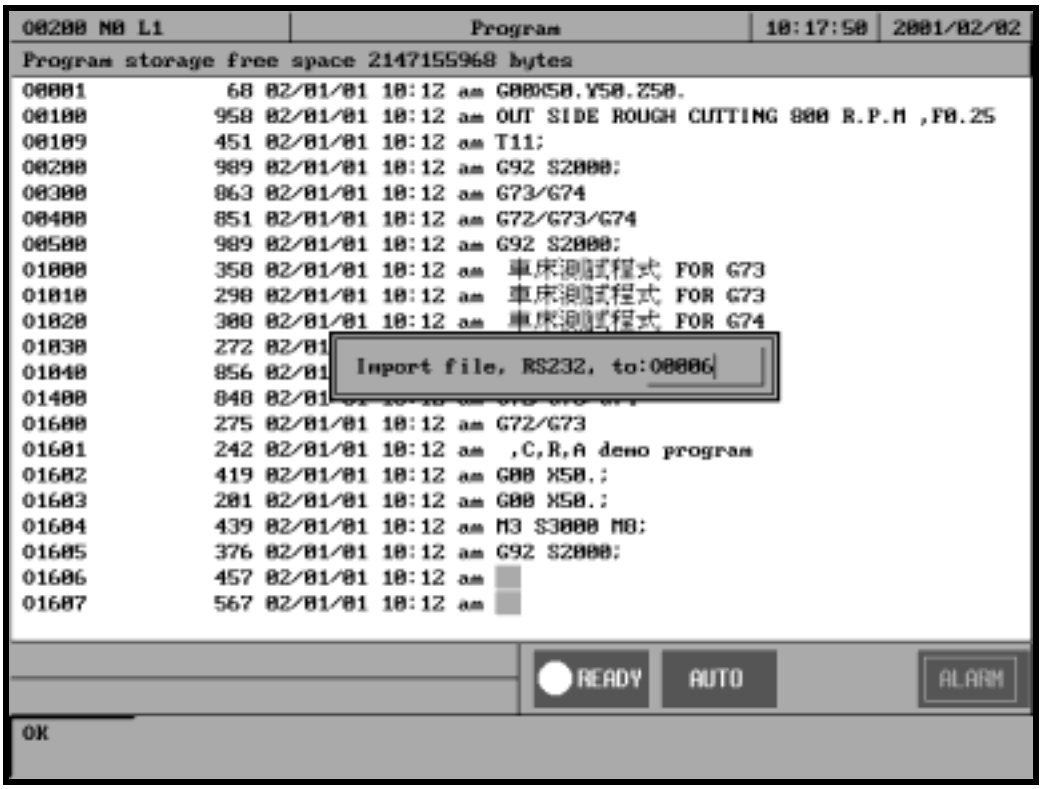
Function: Output file to floppy disk

Operation: Select a file by pressing（↑、↓、←、→）and then press F5. After following the prompt in the dialog box, confirm this operation by pressing 【ENTER】.

1.3.2.7.6 F6：RS232 Import /F7 RS232 Export

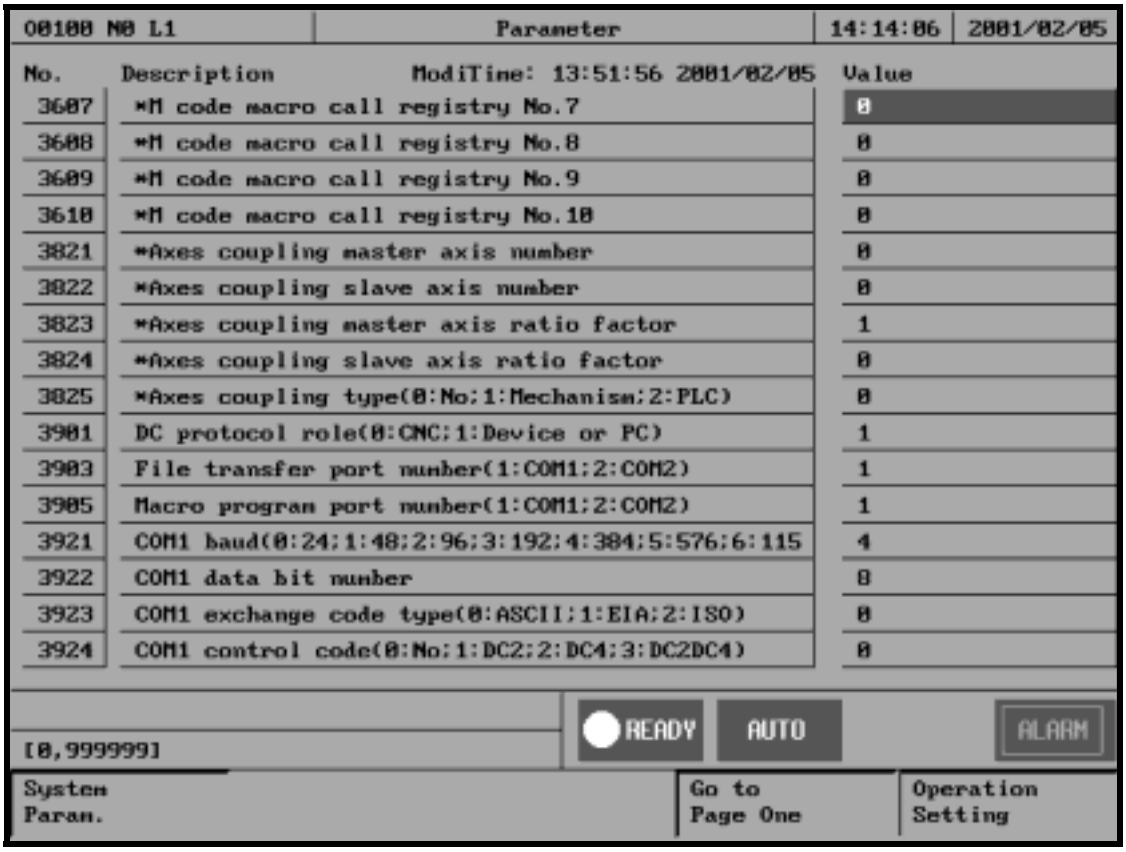
Function: use RS232 communication function to import /Export NC files

Operation: Follow the prompts in the dialog box and type in needed data.



1.3.2.7.7 RS232 communication parameter setting

CNC parameter settings from : 3901 to 3929



00100 N0 L1	Parameter	14:14:06	2001/02/05

No.	Description ModiTime: 13:51:56 2001/02/05	Value
3607	*M code macro call registry No.7	0
3608	*M code macro call registry No.8	0
3609	*M code macro call registry No.9	0
3610	*M code macro call registry No.10	0
3821	*Axes coupling master axis number	0
3822	*Axes coupling slave axis number	0
3823	*Axes coupling master axis ratio factor	1
3824	*Axes coupling slave axis ratio factor	0
3825	*Axes coupling type(0:No;1:Mechanism;2:PLC)	0
3901	DC protocol role(0:CNC;1:Device or PC)	1
3903	File transfer port number(1:COM1;2:COM2)	1
3905	Macro program port number(1:COM1;2:COM2)	1
3921	COM1 baud(0:24;1:48;2:96;3:192;4:384;5:576;6:115	4
3922	COM1 data bit number	8
3923	COM1 exchange code type(0:ASCII;1:EIA;2:ISO)	0
3924	COM1 control code(0:No;1:DC2;2:DC4;3:DC2DC4)	0

[0,999999]

READY AUTO ALARM

System Param. | Go to Page One | Operation Setting

1.3.3 F3：DlgInput (ONLY for 920T)

Pressing "F3" under the main menu to begin dialog box input as shown in the following figure. After users follow the prompts shown on the dialog box to key in every needed parameter, the SNC lathe controller can start lathe a work piece immediately.

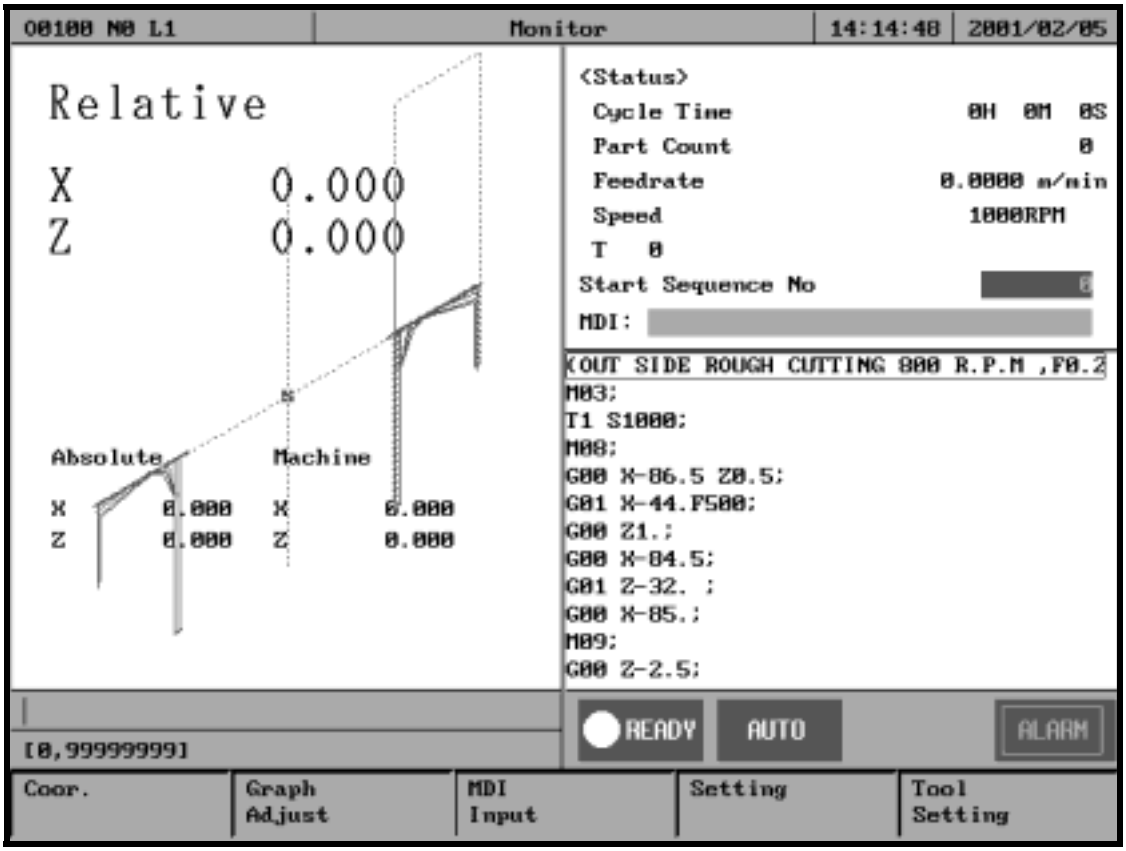
1.3.4 F4：Monitor

This selection displays machining speed, time, manual data input (MDI) and some machine information such as coordinate, range or program at the run time. Press F4 under the main menu to select this function.
Key Selections：

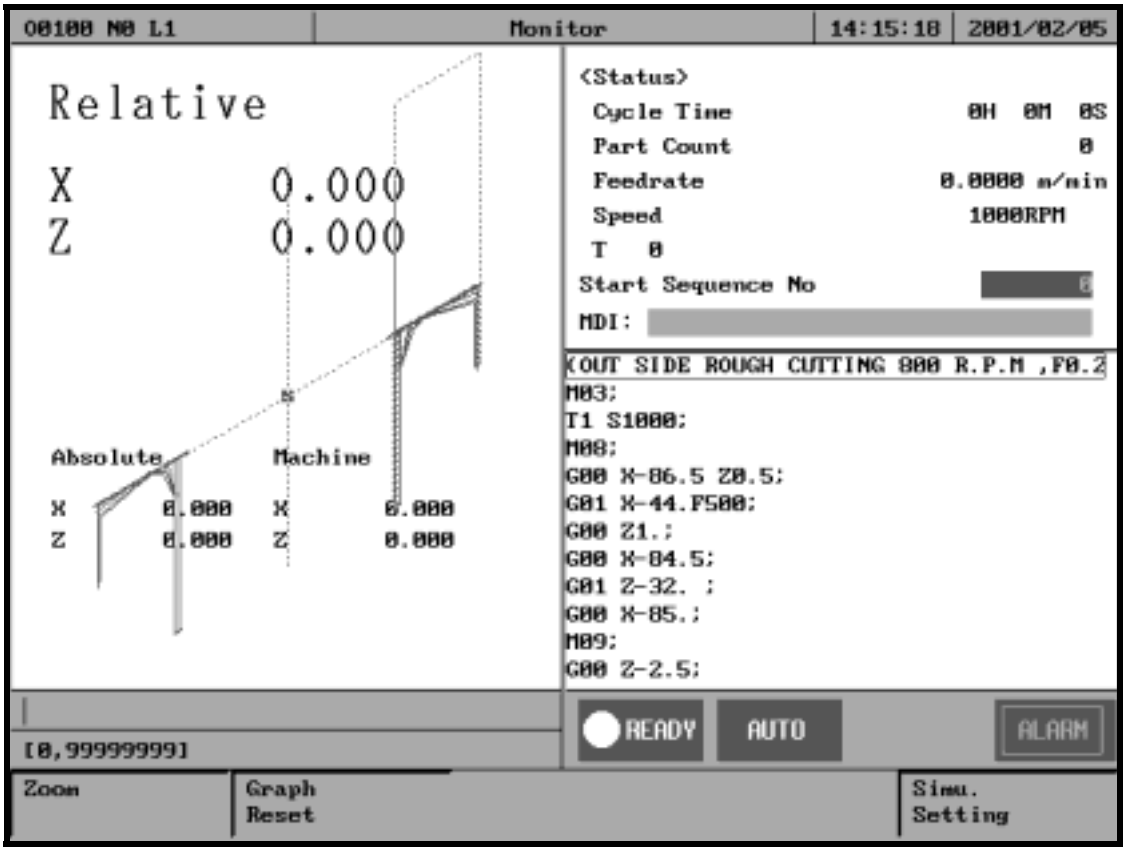
1.3.4.1 F1：Coor

Function: Toggle way of display among the four different coordinate systems, graphical working paths display and the absolute coordinate. (Absolute coordinate is display at the upper right corner of the left half screen.

Operation: Under Monitor submenu, press F1 to toggle coordinate display between the four coordinates as shown as the following figure:



1.3.4.2 F2：Graph Adjust



Key Selections：

1.3.4.2.1 F1：Zoom

Function: To enlarge the workpiece graph.

Operation: Under Monitor submenu, press F2 and then F1.The operator can use the cursor to move the frame to the determined area.

1.3.4.2.2 F2：Graph Reset

Function: To recover the zoomed workpiece graph.

Operation: Under Monitor submenu, press F2”Graph Adjust” and then F2”Graph Reset”

1.3.4.2.3 F5：Simu Setting

Function: To set simulation parameter

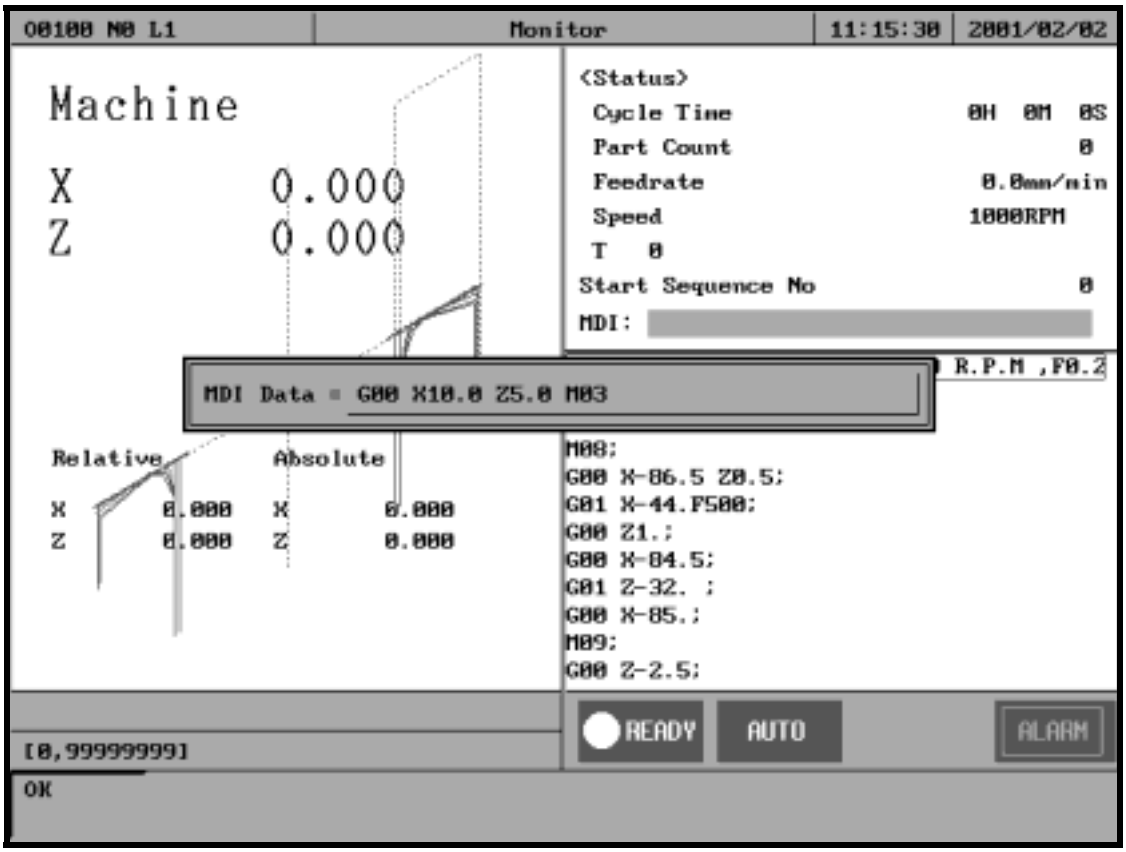
Operation: Under Monitor submenu, press F2”Graph Adjust” and then F5”Simu Setting” .

1.3.4.3 F3：MDI Input

Function：Manual Data Input

Operation：Users can operate SYNTEC Lathe Controller manually in MDI mode.

1. Press F3 under Monitor submenu and type in single-line G or M code.
2. Press F1 (OK) to confirm the input command.
3. The typed-in command line will show on right upper corner of the screen.
4. Press 【CYCLE START】on the machine panel to execute the single-line command.
5. The following figure shows an example of this function.

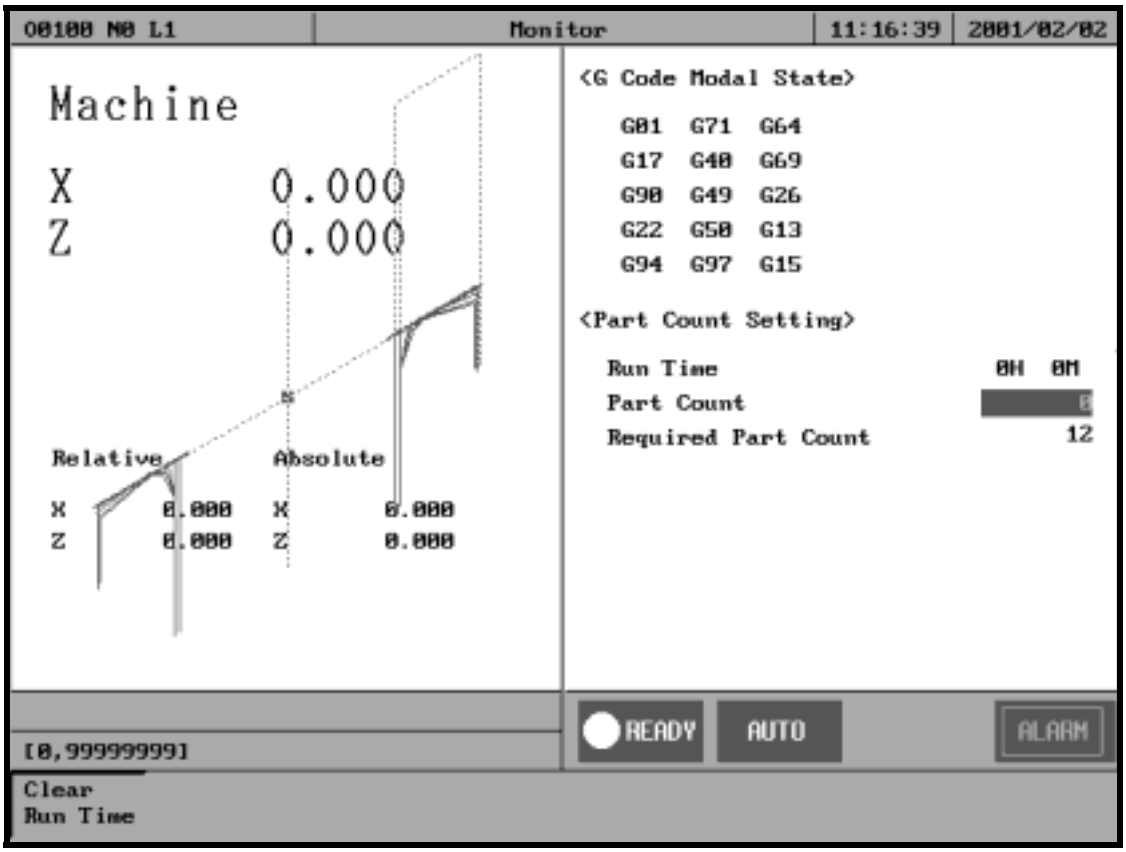


1.3.4.4 F4：SETTING

Function：To set the part count and also set required current

Operation：From this screen users can set the part count what he need

1. when CNC execute M02 ,M30 ,M99 ,part count would add 1 automatically ,
2. When part count reach required part count ,CNC would stop executing.



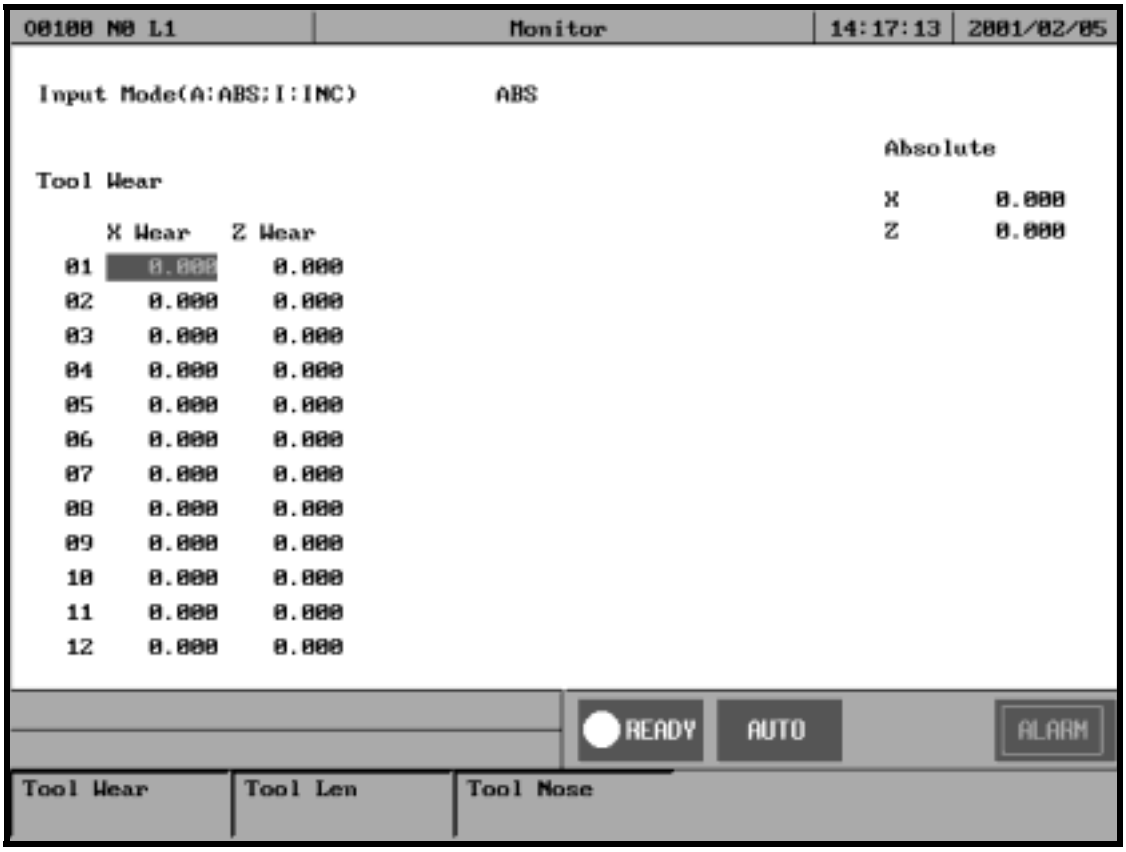
1.3.4.5 F5：Tool Setting

Function：To set the tool compensation value ,including tool length ,tool wear ,tool norse ,describe as below :

Tool length : G43/G44 tool length Hn compensation

Tool Length wear : for small length dimension adjust

Tool norse : G41/G42 tool radius Dn compensation (not diameter)

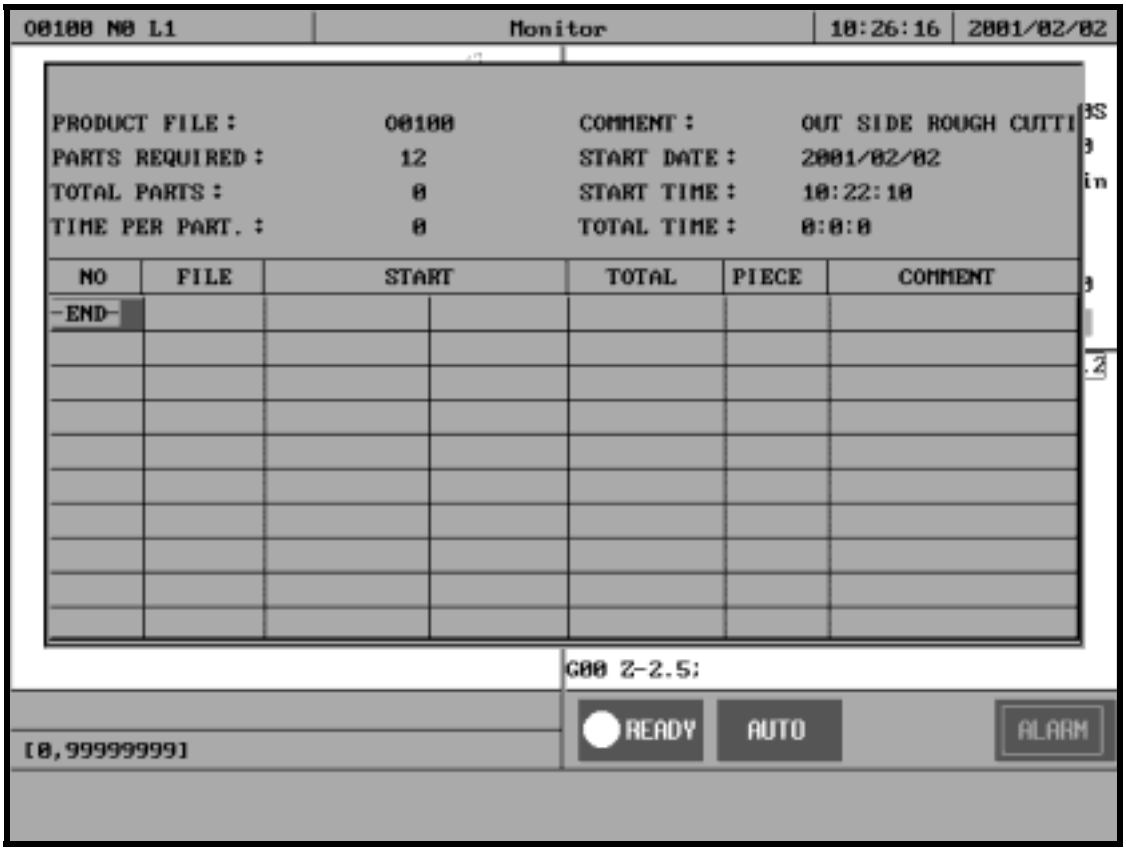


Operation procedure :

1. Main screen press F4 “monitor “
2. Press F5 “Tool Setting”
3. F1 ” Tool Wear ”
4. F2 ” Tool Len ”
5. F3 ” Tool Nose ”
6. Use ” Page Up ”,” Page Down ”,”↑”, ”↓ ”, ”← ”, ”→ ”select tool No ,and X,Z item

1.3.4.6 F8：Work Record

Function：This table can record 300 sets executed NC file ,this is very helpful to know end user working history .

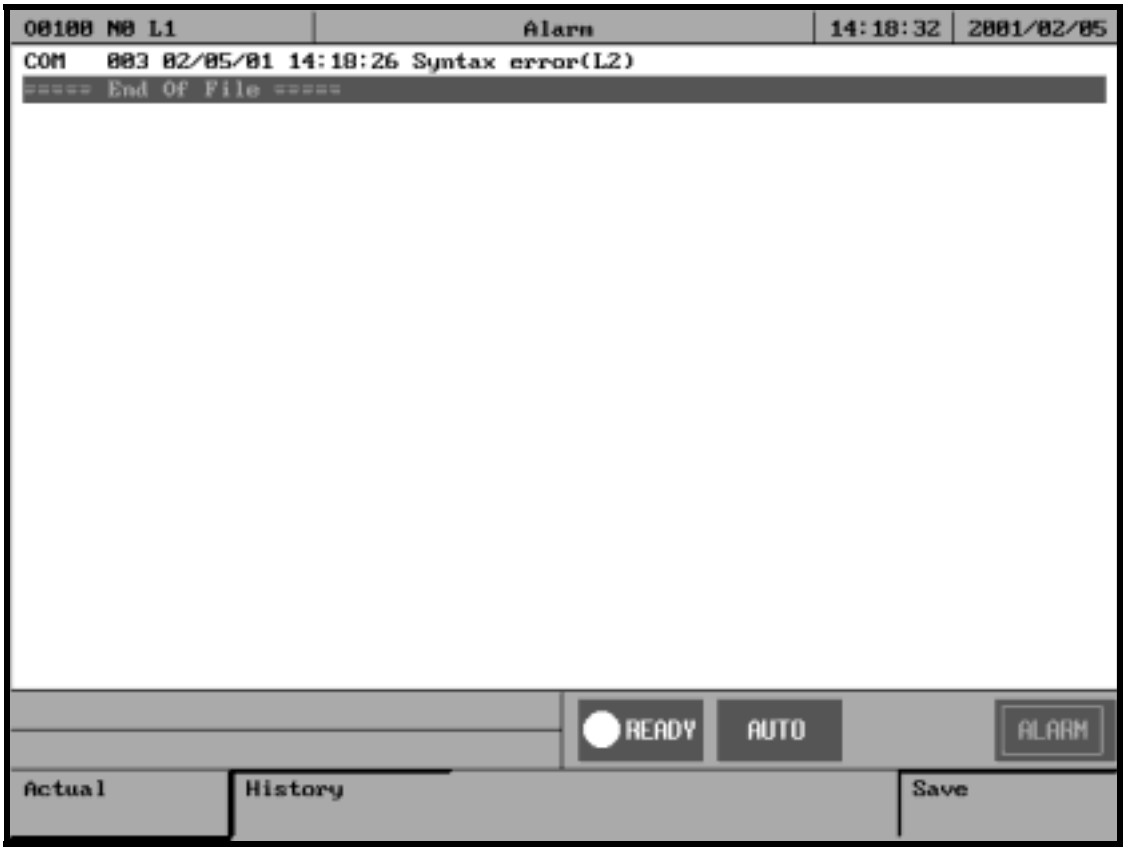


Operation Procedure :

1. main screen ,press F4 “monitor”
2. at main screen ,Press right arrow “◥ “
3. Press F5 “Work record”

1.3.5 F5：Alarm

Whenever the system or the program stops running due to some errors, there will be an alarm message shown on the screen. In order to clear the errors, users can press F5 in the main menu for Alarm submenu as shown in the following figure.



Key Selections:

1.3.5.1 F1：Actual

Operation: Under the Alarm submenu, press F1 to show current alarm situation.

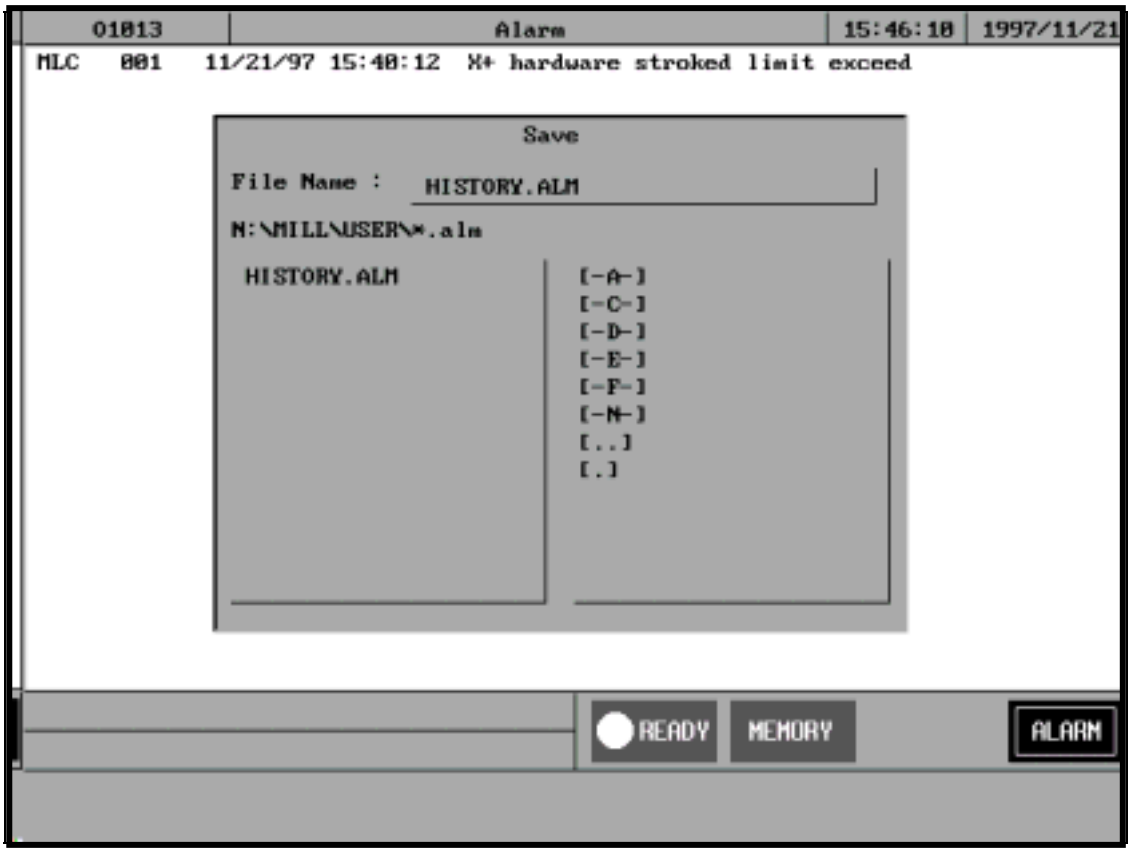
1.3.5.2 F2：History

Operation: Under the Alarm submenu, press F2 to show the alarm history of the system.

1.3.5.3 F5: Save

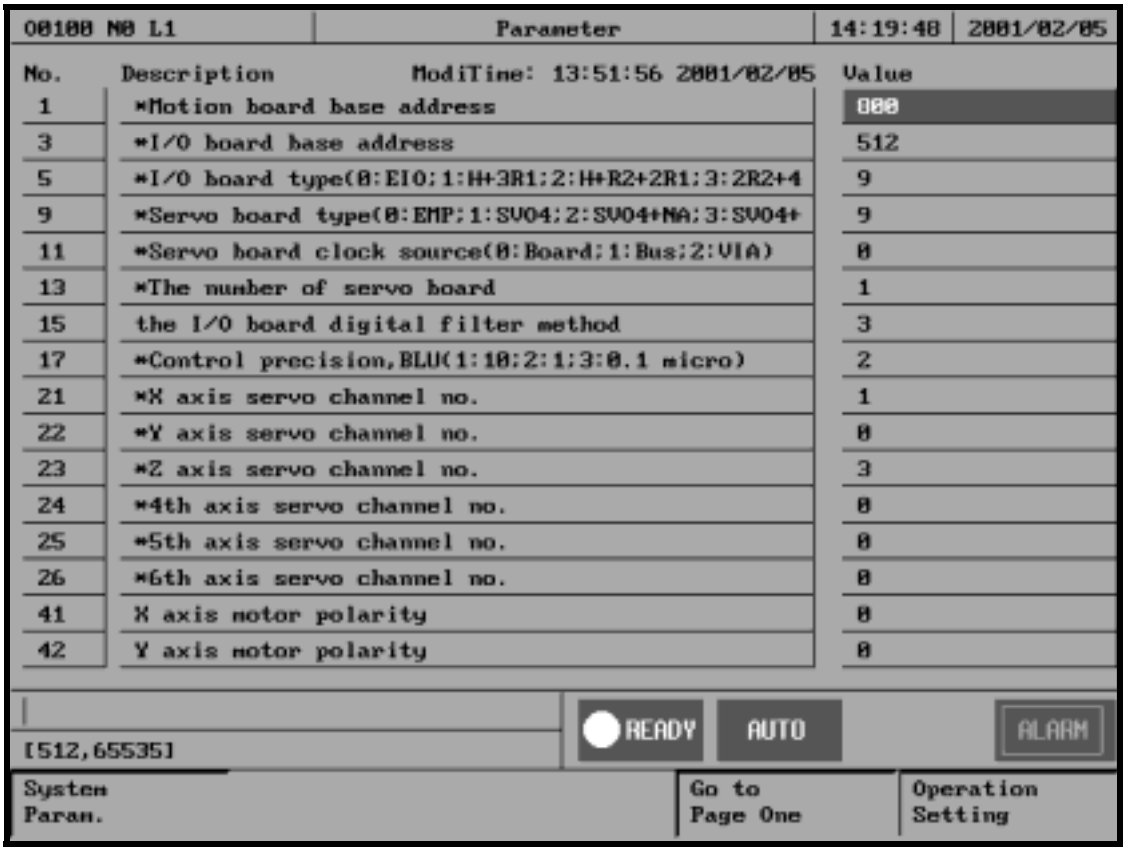
Function：Save Alarm History To File.

Operation：Under the Alarm submenu, press F5 to save alarm history to a file as shown in the following figure. A dialog box will prompt users to type in file name to be saved. After selecting a disk drive by using (↑、↓、←、→), users press【ENTER】to confirm this operation.



1.3.6 F6：Parameter

Under the main menu, press f6 and then f1 to enter this function as shown in the following figure.



Key Selections：

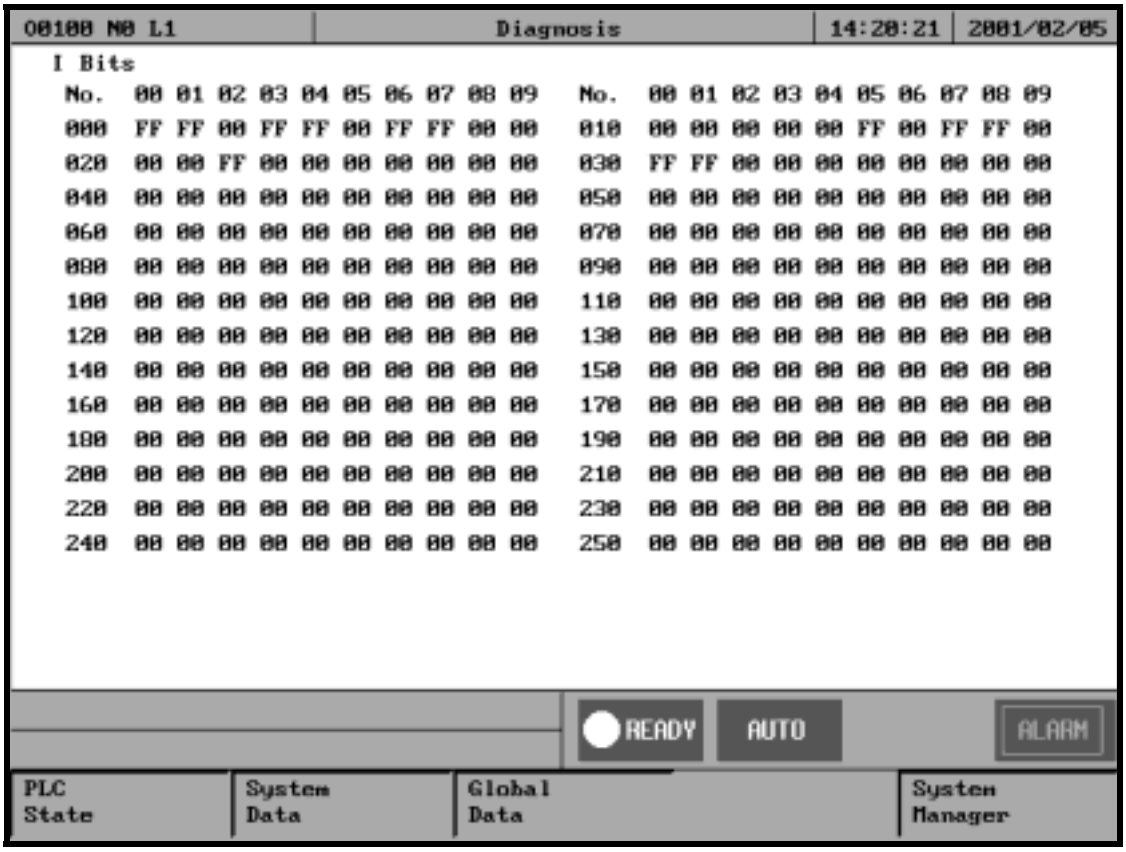
1.3.6.1 F1：GoTo

Function：Parameter setup for servo system such as servo parameters, mechanical parameters, programmable travel range, maximum machining speed and so on.

Operation：While the system is in manual data input(MDI) mode , users needs to press E-Stop before setting the parameters and “Not Ready” will show up on the status line. After key in new values for the parameters, users need to reboot the system before any change takes effect.

1.3.7 F7：Diagnosis

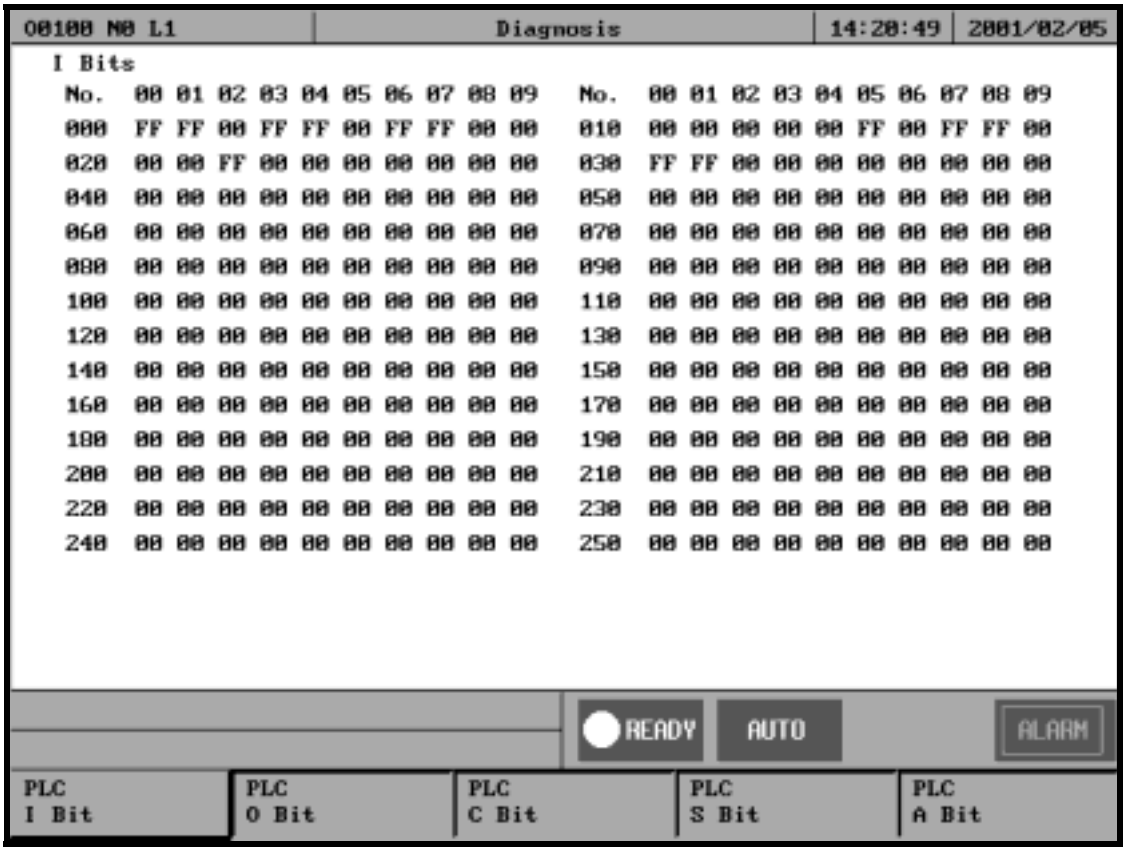
This selection provides users with direct access to the memory area for parameter checking, parameter settings and NC diagnosis function. It can also be used to maintain and debug the control devices. Under the main menu, press F6 and then F2 to access this function as shown in the following figure.



Key Selections:

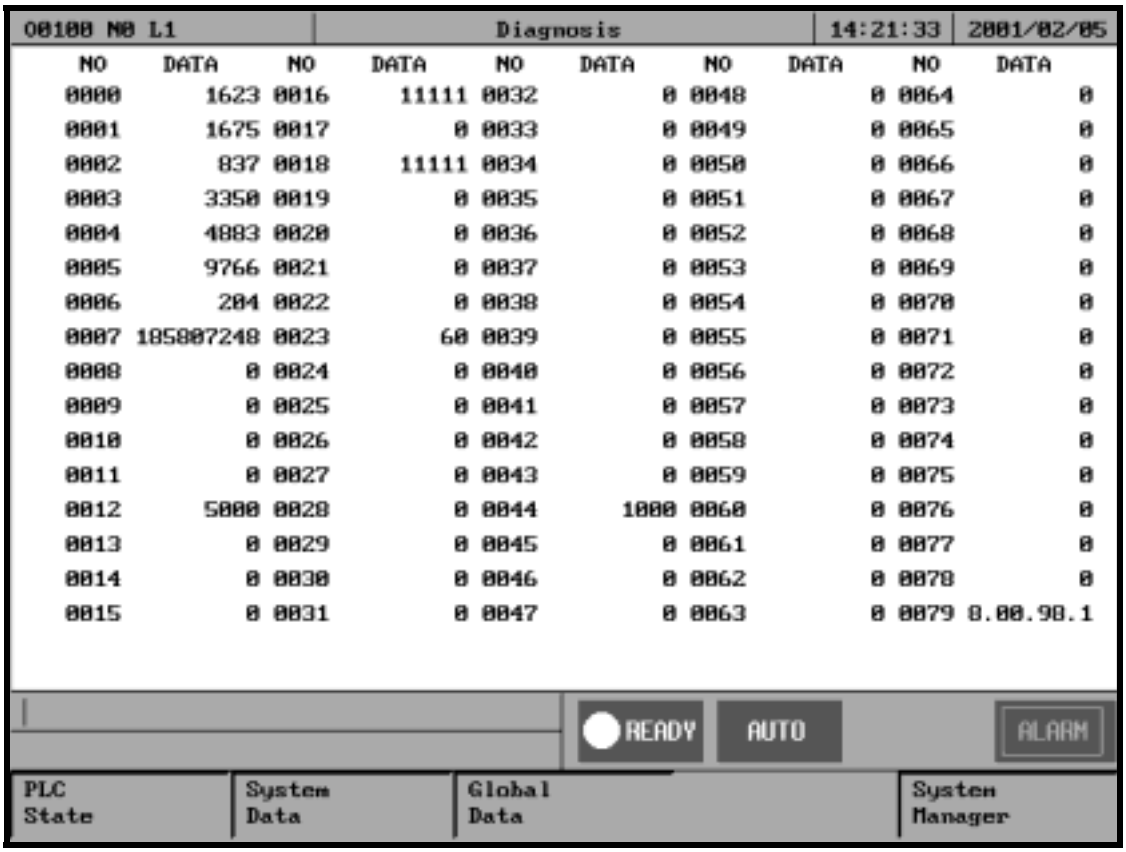
1.3.7.1 F1：PLC State

Function：for upgrade system software ,or Ladder ,system parameter …



1.3.7.2 F2：System Data

Function：



1.3.7.3 F3：Global Data

1.3.7.4 F5：System Manager

1.3.8 F8：Guidence

If users have any problem about SNC Lathe controller, in addition to user manual, users can also use this function for on-line help. Under the main menu, press “F8“ for on-line help.

Chapter 2 Machine operation panel

2.1 2nd machine operation panel

2.1.1 POWER ON

Turn on main power

2.1.2 POWER OFF

Turn OFF power

2.1.3 Emergency STOP

For safety reason ,Press this button ,CNC would stop all movement ,and also stop all main power ,so ,people and machine safety is guarantee .

2.1.4 Home mode and Home function

Discription : when CNC power on ,please do the home function

Operation :

1. mode select to HOME mode
2. Press axis manual key X+,X-,Y+,Y-,Z+,Z-:
3. CNC would start the machine home function

2.1.5 Continus JOG (Rapid JOG)

Discription : User can use this function to move the machine by press JOG key

Operation :

1. mode select to CON JOG mode
2. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move
3. operator can use JOG% or G01% adjust Jog federate
4. When operator press manual key and rapid Key "〰" at the same time CNC would move the work table "RAPID speed"
5. operator can use G00% Rapid Jog federate

G00 % : Adjust G00 % (F0 .25% .50% .100%)

G01 % : Adjust G01/G02 /G03 feedrate override %:

2.1.6 Incremental JOG

Discription : User can use this function to move the machine by press JOG key

Operation :

1. mode select to INC JOG mode
2. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move a fixed distance
3. operator can set the incremental distance by G00 rotary switch , *1 : 1um ,*10 : 10um ,*100 : 100um

2.1.7 MPG JOG

Discription : User can use this function to move the machine by MPG(Manual Pulse Generator)

Operation :

1. mode select to MPG mode
2. Select axis by hand box
3. Select incremental distance
4. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move a fixed distance *1 : 1um , *10 : 10um , *100:100um , *1000 : 1000um

2.1.8 AUTO mode NC file execute

Discription : User use this function to execute NC file

Operation :

1. mode select to AUTO mode
2. After Home function .AUTO mode is available
3. Set workpiece coor.(G54..G59) ,CNC default G54 ,if user don't set any G94..G59 in the NC file
4. Set Tool setting ,to select tool radius and tool length .
5. Press "START" key to start the NC program .
6. Press "Feedhold" key to feedhold the NC program ,if necessary

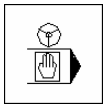
2.1.9 MDI mode single block execute

Discription : User use this function to execute a block without NC file

Operation :

1. mode select to MDI mode
2. After Home function .MDI mode is available
3. Main function select F4”Monitor”
4. Press F3 “MDI Input” ,screen would pop up a window.
5. After key in date ,Press “ENTER” key to input the data
6. Press “START” key to start the MDI block.
7. If MDI block SYNTAX is correct ,data in MDI menu would disappear

2.1.10 MPG Simulation



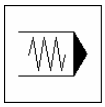
Discription : User can use this function to check NC file

Operation :

1. mode select to AUTO mode
2. Press this button ,and button led light“ON”
3. Press “START” key to start the NC file.
4. CNC would change machine status from “READY” to “BUSY”
5. Machine is still not moving
6. Operator can use rotate MPG to start the NC file
7. MPG rotate more fast ,machining speed is more fast
8. MPG stop ,CNC stop too .
9. this function can “Enable” “ Disable” immediately

P.S. this function is very friendly to user to check his program

2.1.11 Dry Run

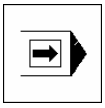


Discription : User can use this function to check NC file

Operation :

1. mode select to AUTO mode
2. Press this button ,and button led light“ON”
3. Press “START” key to start the NC file.
4. CNC would change machine status from “READY” to “BUSY”
5. this function can “Enable” “ Disable” immediately

2.1.12 Single block

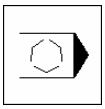


Discription : User can use this function to check NC file

Operation :

1. mode select to AUTO mode
2. Press this button ,and button led light“ON”
3. Press “START” key to start the NC file.
4. CNC would execute NC file only one block and STOP
5. CNC would change machine status from “BUSY ” to “B_STOP”
6. Press “START” again ,then CNC execute next block
7. This function is for user to check his NC file Block by Block

2.1.13 Option Stop

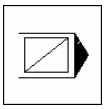


Discription : User can use this function to decide NC file M01 is STOP or not

Operation :

1. mode select to AUTO mode
2. Press this button ,and button led light“ON”
3. Press “START” key to start the NC file.
4. When CNC execute “M01” ,CNC would STOP
5. CNC would change machine status from “BUSY ” to “Feedhold”
6. This function use to change tool or check workpiece

2.1.14 Option Skip



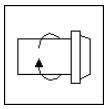
Discription : User can use this function to decide NC file ‘/” is skip or not

Operation :

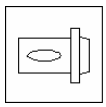
1. mode select to AUTO mode
2. Press this button ,and button led light“ON”
3. Press “START” key to start the NC file.
4. When CNC execute “/” ,CNC would Skip this block
5. If this key is not pressed ,CNC would execute this block

2.1.15 Spindle control

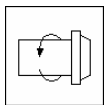
Spindle CW rotate



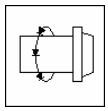
Spindle stop



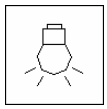
Spindle CCW rotate



Spindle low speed : When spindle is rotate ,
press this key ,spindle would rotate with low speed



2.1.16 Working led

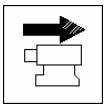


ON/OFF working led

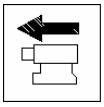
2.1.17 Working Liquid

Flush working liquid

2.1.18 Aux table forward



2.1.19 Aux table backward



2.2 Text key description :

A Z : English 26 character key 。

0 9 : numerical key。

DEL : delete a character

INS : Insert /replace mode switch

SHIFT : for select keyboard the other textkey

SPACE : add a space key

BACK SPACE : backword delete a character

RES ET : ”RESET “ abort the CNC status ,so please be careful to use this key

ENTER : To input current data to input box

HELP : press this key ,user can get help message about this screen

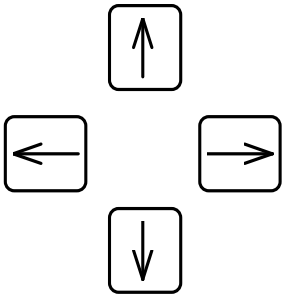
/ : for optional skip key input

; : End of block

. : decimal fraction

(、)、[、]、|、!、&、$、#、<、>、=、%、@、*、 :、 , 、+、-

↑ ↓ : edit cursor Page Up /Page Down



：edit cursor control key

Chapter 3 、 How to operate SYNTEC 900TE

This chapter is written for user task by task ,when user operates this controller ,operator can follow task description as below STEP by STEP ,So very easy to use this controller ,TASK description as below :

3.1 Manul function(JOG ,INC_JOG ,MPG)

When power on SYNTEC CNC ,there are 3 mode to manual machine

1st CON_JOG :

1. Release emergency stop button ,CNC status “NOT READY”change to “READY ”
2. Mode select switch rotate to JOG mode
3. Press axis direction key(X+,X-,Y+,Y-,Z+…) ,table would move
4. operator can use JOG% adjust JOG speed
5. operator can press axis direction key and rapid key “ “at the sametime ,machine will move by rapid speed
6. rapid JOG speed can be adjusted by G00%

Incremental JOG :

1. Release emergency stop button ,CNC status “NOT READY”change to “READY ”
2. Mode select switch rotate to INC JOG mode
3. Press axis direction key(X+,X-,Y+,Y-,Z+…) ,table will move a fixed distance once
4. operator can select incremental distance by G0% (*1,*10,*100)

MPG incremental jog (MPG):

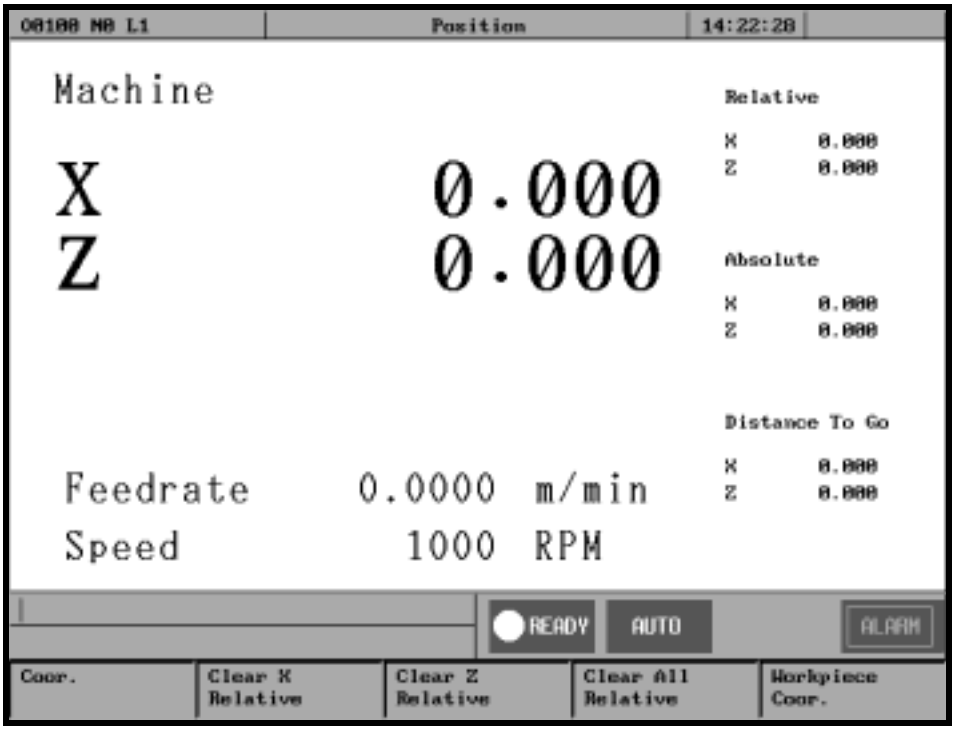
5. Release emergency stop button ,CNC status “NOT READY”change to “READY ”
6. Mode select switch rotate to MPG INC JOG mode
7. Select movement axis
8. Select movement distance (*1,*10,*100)
9. Rotate MPG ,table would move .

3.2 HOME

Because tool setting ,workpiece coordinate setting is based on Machine zero point ,so ,it is necessary to make sure where is machine zero (HOME) ,so ,when CNC bootup ,execute HOME function is very important ,otherwise SYNTEC CNC controller would not be allowed to start AUTO NC files

Procedure :

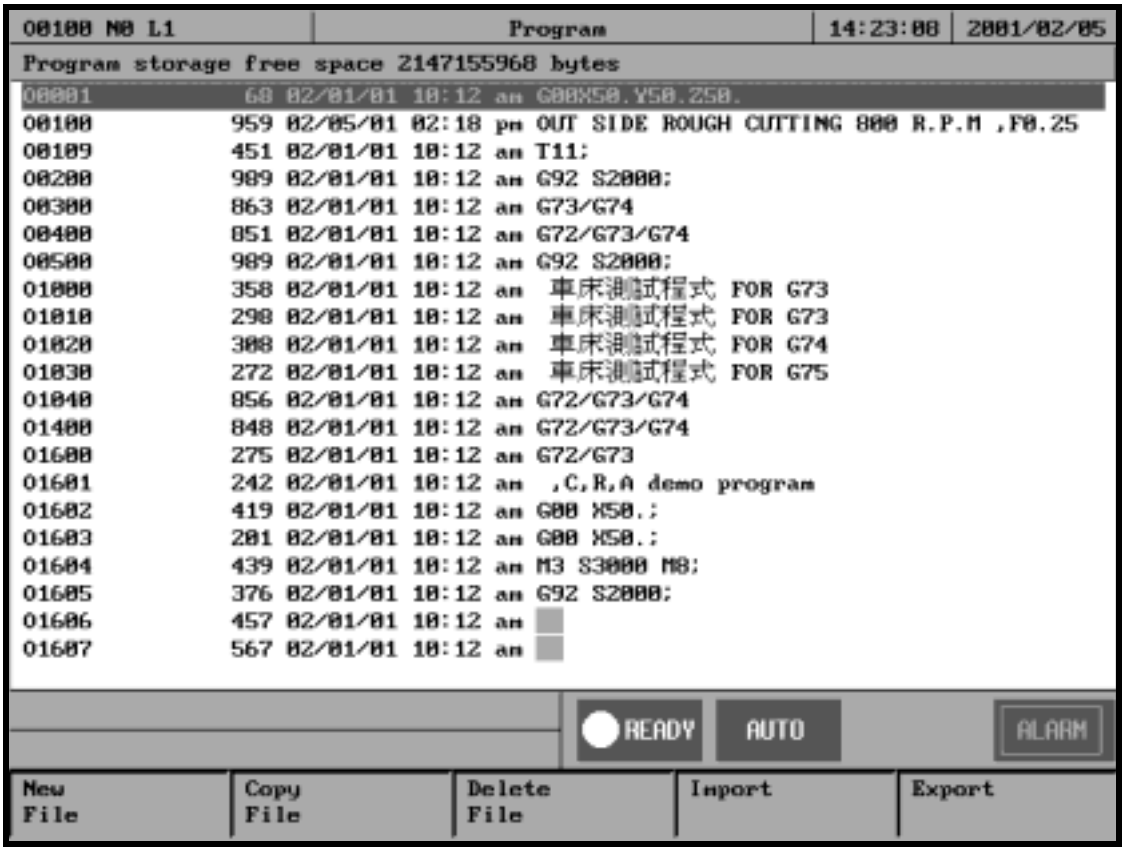
1. Release emergency stop button ,CNC status “NOT READY”change to “ READY ”
2. Mode select switch rotate to HOME mode
3. Press axis direction key(X+,X-,Y+,Y-,Z+…) ,axis would start HOMING
4. Home direction is defaulted in the CNC parameter
5. Home function can run 3 axis at the same time
6. After home function ,machine coordinate would be zero .
7. After home function completed ,software limit protectin is available ,so ,before HOME function ,please don’t run machine too fast.



3.3 Open a file (EDIT / FLOPPY /RS232)

Procedure :

1. Press Group function key “Program”
2. Press submenu function key “File manage”
3. screen display file system screen

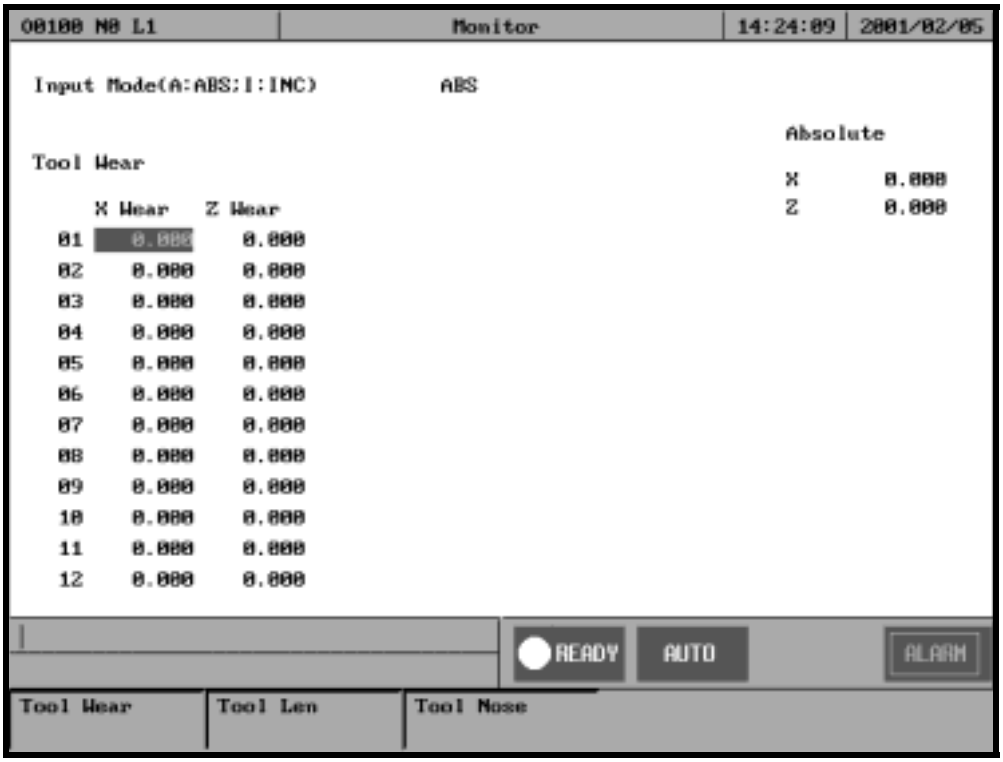


4. Press F1 “New file” to open a new file
5. Press F2”copy file”, to copy current hilight file to target file
6. Press F3 “delete file “ ,to delete current hilight file
7. Press F4 “Inport” ,to inport a new file from floppy disk
8. Press F5 “Export “ ,to export hilight file to floppy disk
9. Press F6 “RS232 Inport” , ,to inport a new file from RS232
10. Press F7 “RS232 Export “ ,to export hilight file to RS232

3.4 Tool setting (G40/G41/G42 ,G43/G44/G49)

Procedure for setting Tool offset value :

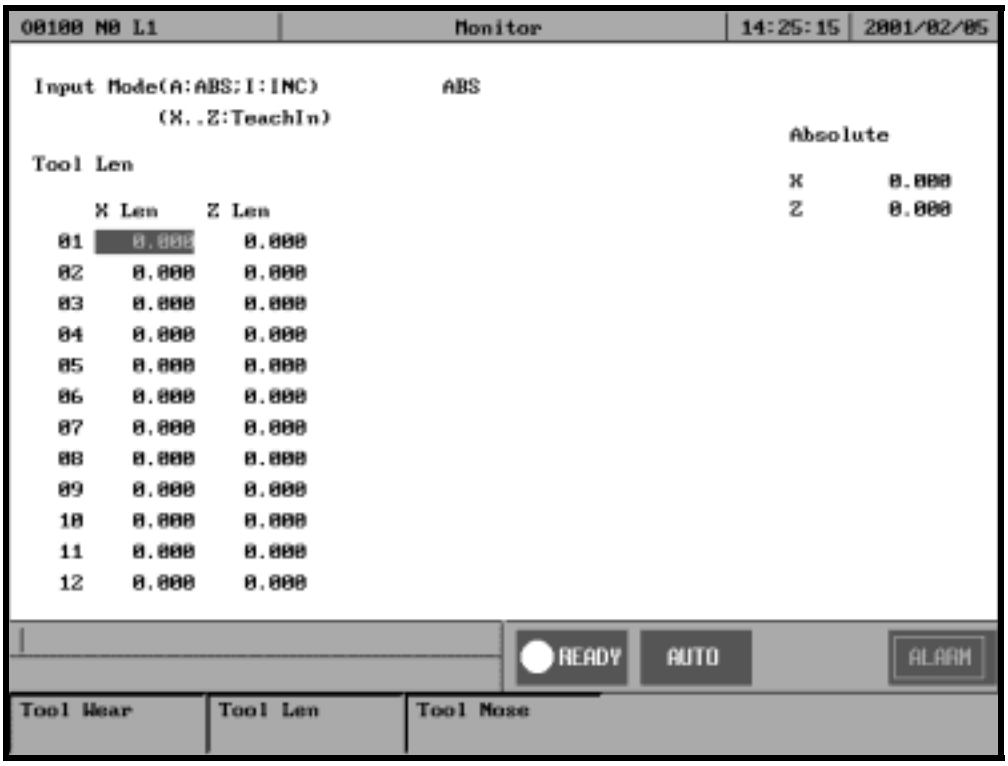
1. Press Group function key “Monitor”
2. Press function key F5 “tool setting “
3. Move the cursor to the compensation value to be set or change using page keys and cursor keys
4. Type “A” or “I” key to selected input type is “Absolute” or “Incremental”
5. Generally use Absolute type to input Tool radius and Tool length
6. Use Incremental type to input radius wear and Length wear for small value adjust
7. (Tool radius + radius wear) is real G41/G42 compensation value
8. (Tool length + length wear) is real G43/G44 compensation value



3.5 Tool Length measurement (G43/G44/G49)

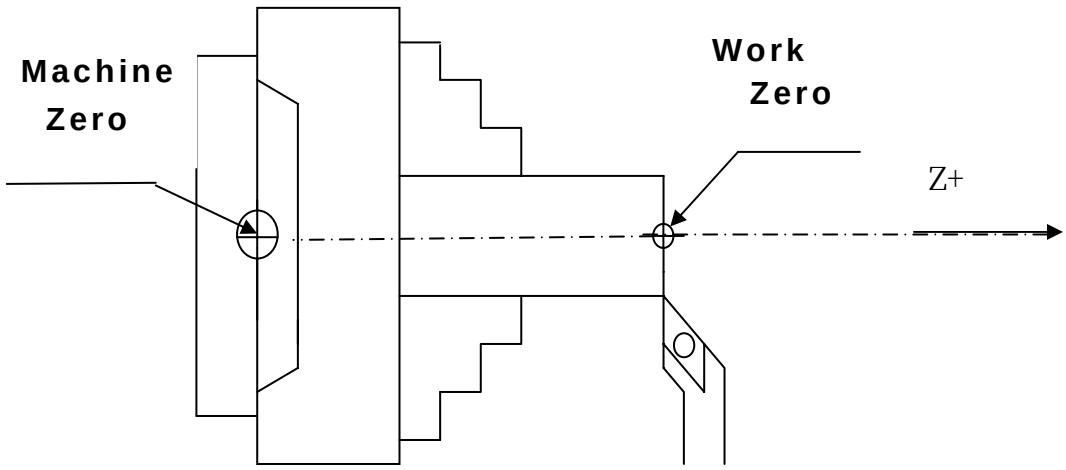
After NC code select T code ,then G43 is enable automatically ,so ,operator must set tool length data before cutting.

1. Press Group function key “Monitor”
2. Press F5 “Tool setting”to this screen
3. Press F2 “Tool len” to set tool length
4. Move the cursor to the compensation number for the target tool



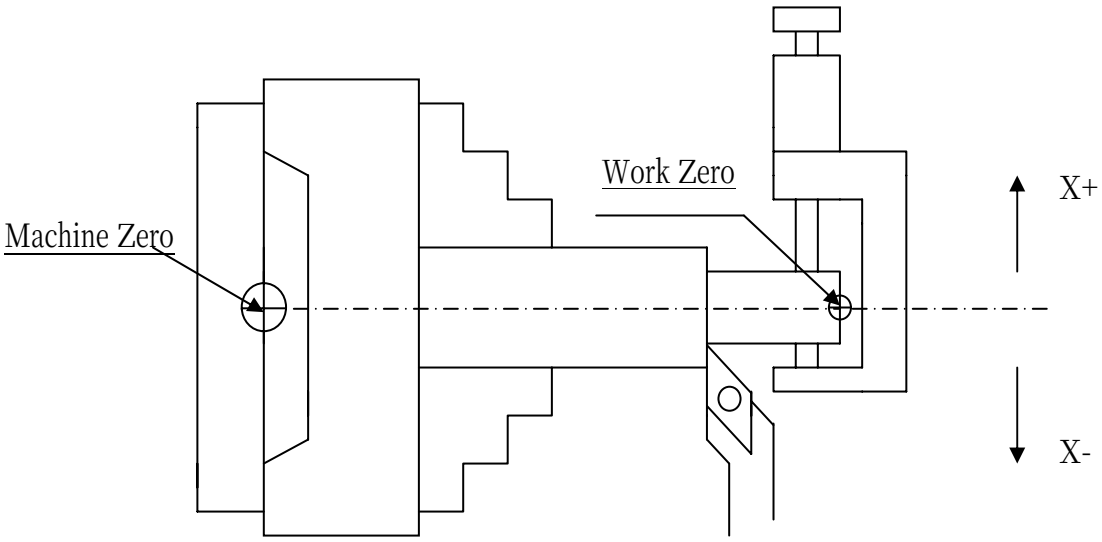
3.5.1 Z axis tool length measurement Procedure

1. Use manual operation to move the tool to be measured until it touches the z axis work zero point .
2. There are 3 methods to input z tool length
   (1) Absolute : enter work zero point machine coordinate ,by "A*** " ,tool length would be replaced by this value
   (2) Incremental : enter Incremental value ,by "I*** " , use this function to shift current tool length by this value
   (3) Teach : enter current position relative to work zero point by "Z**" ,for example "Z0" means current positon is work zero point ,and after this ,tool length would be the same with machine coordinate .



3.5.2 X axis tool length measurement Procedure

1. Use manual operation to move the tool to be measured until it touches the X axis work zero point .
2. There are 3 methods to input z tool length
   (1) Absolute : enter work zero point machine coordinate ,by "A*** " ,tool length would be replaced by this value
   (2) Incremental : enter Incremental value ,by "I*** " , use this function to shift current tool length by this value
   (3) Teach : enter current position relative to work zero point by "X**" ,for example "X0" means current positon is work zero point ,and after this ,tool length would be the same with machine coordinate .
3. Teach by cutting : procedure as below
   (1) Cut a distance along Z axis
   (2) Retract tool along Z axis , don't move X axis
   (3) Measure cutting diameter "D "
   (4) Input this value by teaching input "X+/-D"
   (5) The plus/minus sign is decided by tool tip position ,if tool tip is located at X+(tip is located above central line) , sign is "+" ,else if tool tip is located at X- sign is "-"



3.5.3 Tool wear setting

Use tool wear setting to adjust cutting dimension ,procadure as below

1. After tool length setting ,tool wear value is setted to zero automatically
2. measure cutting result ,if there is an error ,use tool wear setting to compensate tool length ,actually tool lemgth = tool length + tool wear
3. There are 2 methods to input tool wear
   (1) Absolute : enter tool wear absolute value ,by “A**” ,after this setting ,tool wear would be changed to this value
   (2) Incremental : enter tool wear incremental value ,by “I** “ after this .tool wear amount would be changed to pre_value plus this value
   (3) Modal setting : only enter “A” or “I” single character ,then “absolute “ or “ incremental” setting would be kept ,after this user can directly input value only .
4. plus/minus sign +/- :
   (1) Input value can be assigned sign “+/-”
   (2) The plus/minus sign is decided by tool tip would be adjusted direction
   (3) if tool tip is modified to move “Plus”direction ,then key in tool wear value by plus sign”+”
   (4) if tool tip is modified to move “minus”direction ,then key in tool wear value by minus sign”-”

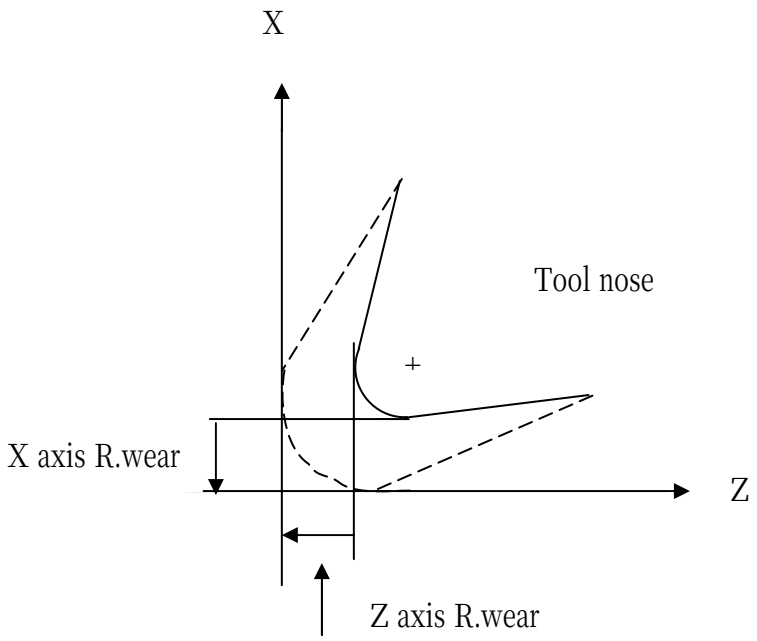
Example :

If cutting result is 10um too big in diameter ,and tool tip would be adjusted to X- ,then key in “ I–o.o1 “ to tool wear value ,then “-10um” would be added to current tool wear value ,next cutting ,tool tip cutting path would move toward “X- “direction 10um in diameter .

3.5.4 Tool nose setting :

Because tool nose is circular ,tool length measurement only measure tool length when precision cutting ,use tool nose setting to compensate tool nose dimensation error ,procedure as below

1. Press group function " monitor "
2. Press F5 " tool setting "
3. Press F3 " Tool Nose " to select tool nose compensate
4. There are 3 kinds tool nose data to input
   Radius : tool nose radius
   R.wear : tool nose redis wear ,actually the real tool nose is (Radius+R.wear)
   Nose : tool nose direction ,there are 8 tool nose directions to be selected ,it depends on tool nose shape ,please see SYNTEC programming manual more detail(G41/G42)
5. Use G41/G42 in NC files ,to enable tool nose compensation .
6. please see SYNTEC programming manual more detail(G41/G42)

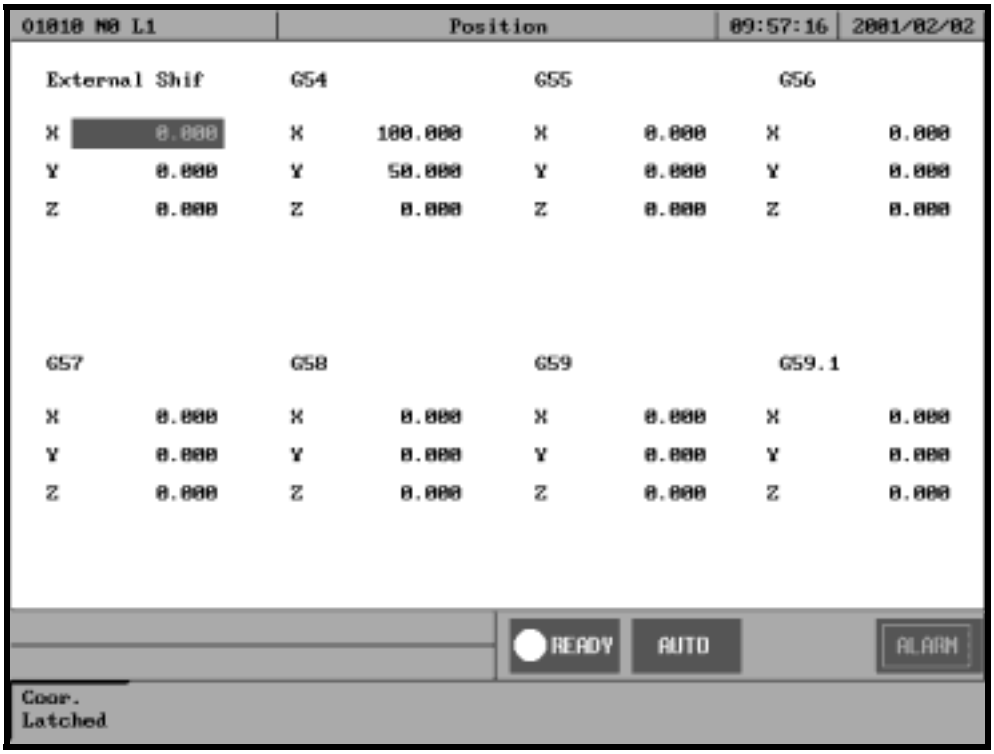


3.6 Setting the Workpiece origin offset value(G54..G59)

Procedure :

1. Press group function key “Position”
2. Press sub selection soft key “Workpiece coor.”

.



3. The screen for displaying the workpiece origin offset values consists two pages . Display a desired page by Press PageUp/PageDn key
4. Move the cursor to the workpiece origin offset to be changed
5. “Extenal shift” input the value ,which can shift the whole coordinate(G54..G59.8) simultaneous
6. F1”coor. Latched”: user can press this function key ,and CNC would latch current machine coordinate to the screen where sursor is located .

Note :

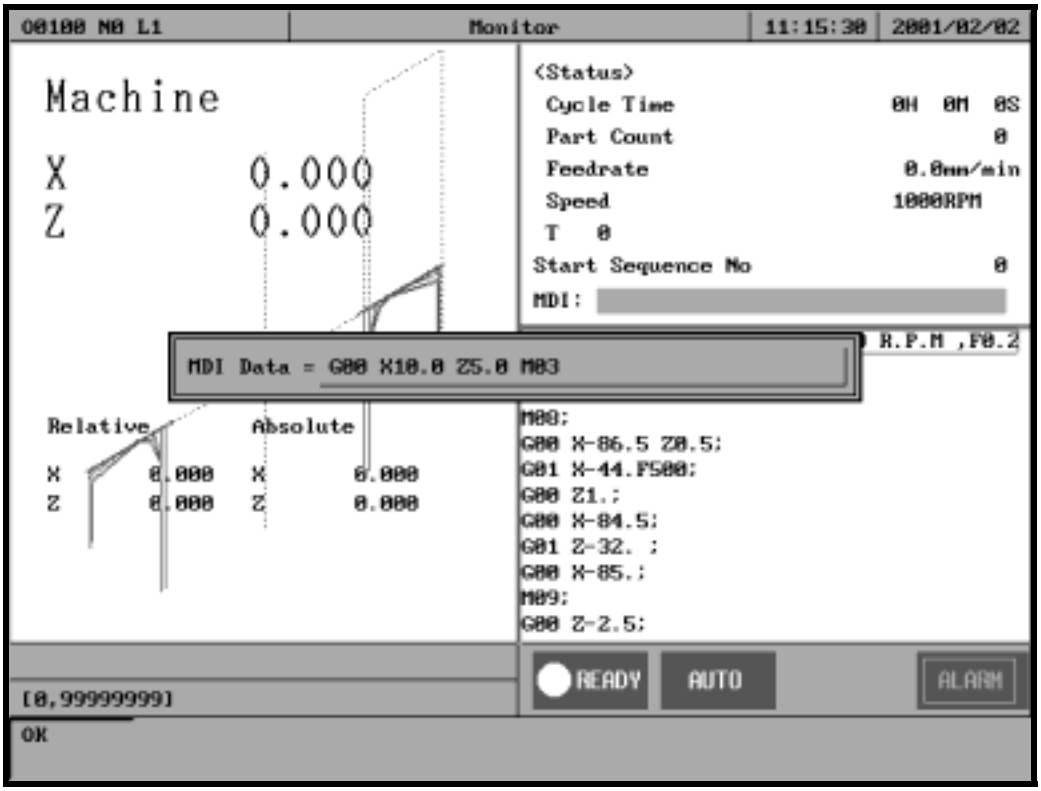
1. This function is enable by setting CNC parameter 3229 :
   disable workpiece coordinate by “ 1 “
2. System default is G54 ,and values are zero every axis
3. After change G54..G59 ,please re_set tool length again ,this is very important

3.7 Manual Data Input(MDI)

Procedure :

1. Mode select switch rotate to MDI mode
2. Press group function key “Monitor ”
3. Press sub selection soft key “MDI input”

.

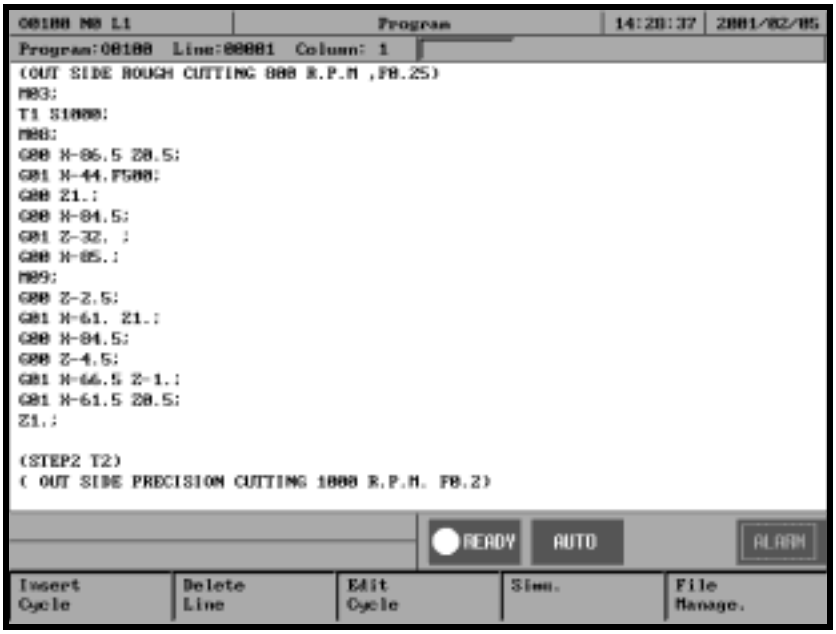


4. The screen display the MDI input window
5. Key in MDI data at input bar and PRESS “ENTER”
6. Press 2nd operation panel “START” to execute the current block
7. If current block SYNTAX is correct ,the data in the window would be disappear .

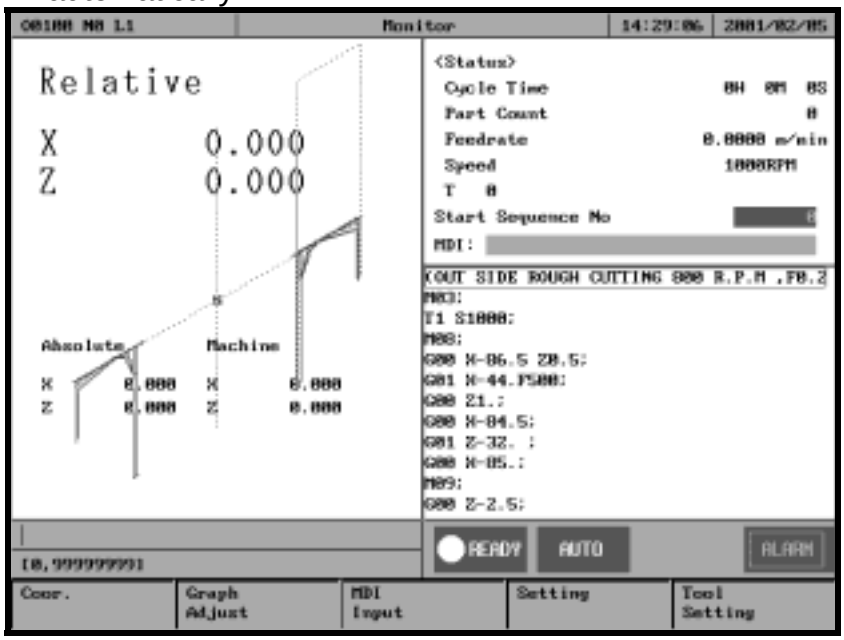
3.8 Assigned an executing NC file (AUTO)

Procedure :

1. Mode select switch rotate to AUTO mode
2. Make sure CNC status is “READY”
3. Press group function key “Program ”,select NC file what user want to execute



4. Press group function key “Monitor”,then executing file is assigned automatically .

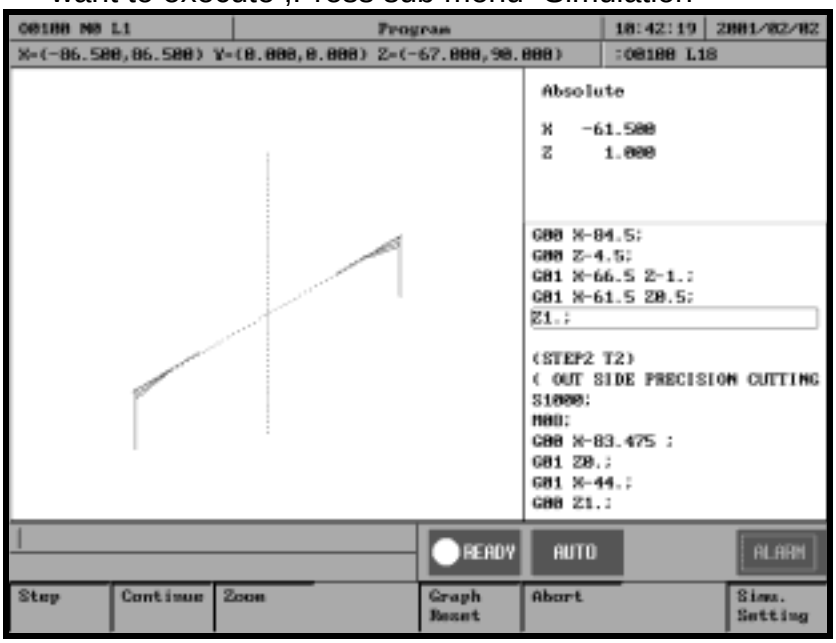


5. Please make sure CNC status is “READY”, that is the only available status to assigned executing NC file .

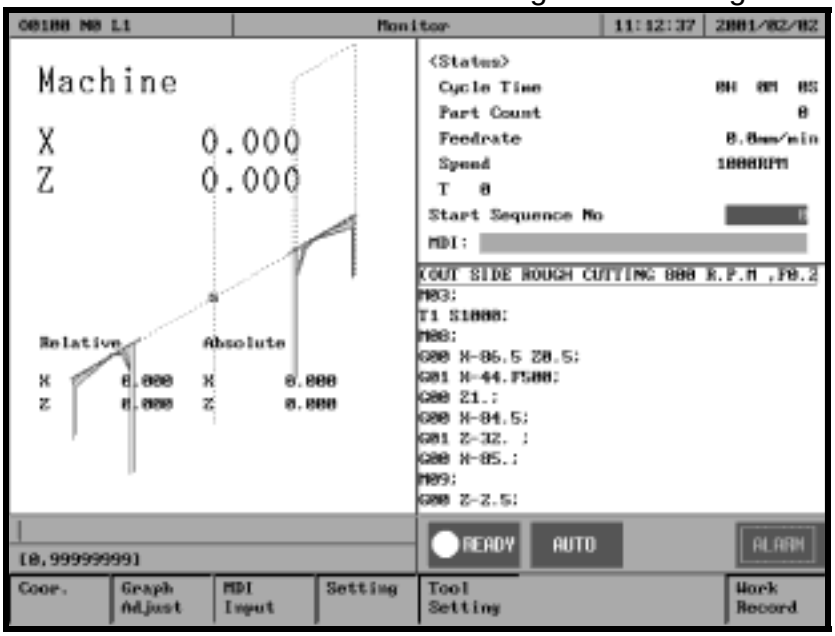
3.9 Graphic Simulation

Procedure :

1. Mode select switch rotate to AUTO mode
2. Press group function key “Program ”,select NC file what user want to execute ,Press sub menu “Simulation”



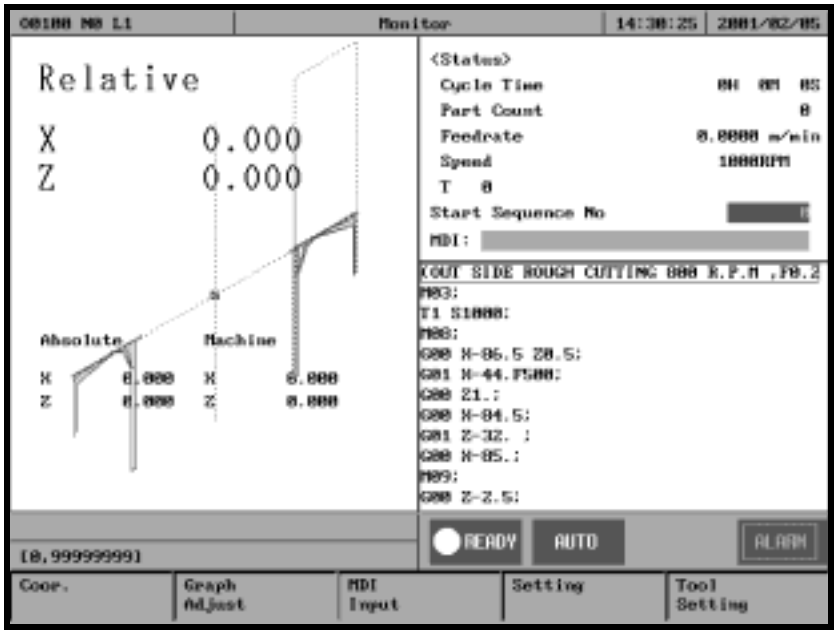
3. User can use “STEP” to check NC file STEP by STEP
4. use “continue” to simulation whole picture
5. Use “Zoom” check more detail
6. Use “simulation setting” set simulation parameter
7. CNC status is “READY”, Press group function key “Monitor” that is the available status to assigned executing NC file .



3.10 How to check NC file in SYNTEC controller

MPG simulation Procedure :

1. Mode select switch rotate to AUTO mode
2. Press group function key “Monitor”



3. Press “MPG simulation “key enable this function (led on)
4. Press “Start “ key ,CNC staus from “READY” to “BUSY”
5. Machien table current status is static
6. Operator rotate MPGs ,then table is start moving along cutting path
7. MPG rotates more fast ,table moves more fast ,MPG stop ,table is stop From monitor screen ,operator can see the cutting cursor move along simulation path
8. Operator also can Press “Single Block” key ,enable single block function when “MPG simulation “ function is ON ,then user can use test two function simutaneous ,check NC file STEP by STEP ,with” MPG simulation “ function .
9. “ MPG simulation “ can control table forward ,also can control backward too ,but NC file would stop at M,S,T code when backward .

3.11.1 RS232 FUNCTION

DNC SOFTWARE ⇔ CONTROLLER (SYNTEC NC CONTROLLER SOFTWARE)

DESCRIPTION：

Users can uses software of CAD/CAM transfer data to controller by RS232 cable. Let user can select be transferred NC file to working.

Operation：

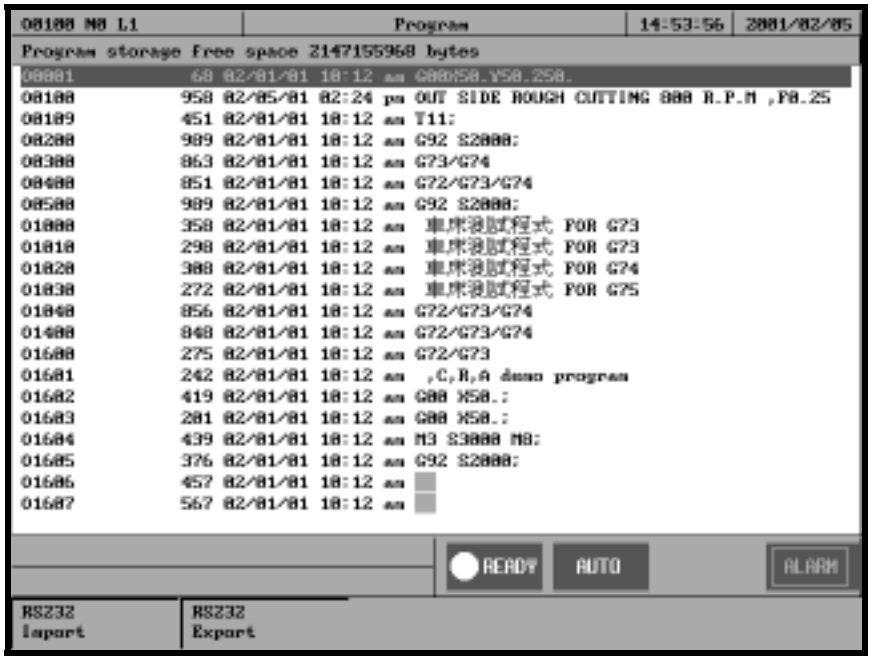
Step I：The hardware line link to COM1

Step II：Parameter of SYNTEC software system setting (Suggest value of default)

Param.	controller	meaning of parameter
3901	0	Dc protocol role, 0：CNC, 1：Device or PC
3903	1	File transfer port number(1:Com1, 2:Com2)
3921	2	Com1 baud(0:24;1:48;2:96;3:192;4:384..)
3922	8	Com1 data bit number
3923	0	Com1 exchange code type (0:ASCII;1:EIA;2:ISO)
3924	0	Com1 control code (0:No;1:DC2;2:DC4;3:DC2DC4)
3925	1	Com1 End-of-block output code(0:EOB;1:CR+EOB)
3926	0	Com1 DC3 control code parity (0:Off;1:On)
3927	2	Com1 flow control(0:No;1:CrsRts;2:XonXoff)
3928	1	Com1 parity check(0:No;1:Odd;2:Even)
3929	1	Com1 stop bit number(1:1 bit;2:2 bit)

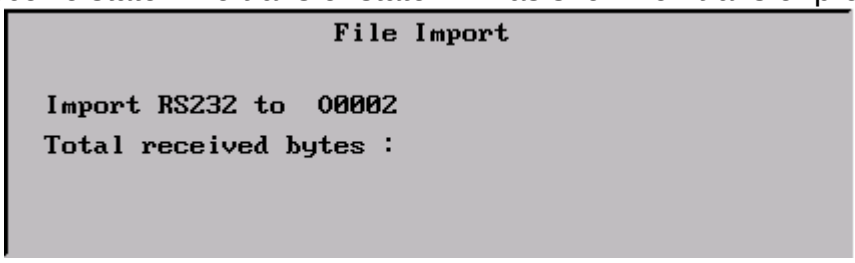
Step III：Turn on the controller power. The RS232 file input is ready. Operation has been stated as follows.

1.Press [program] [file manage] to file manage submenu. And than to press [=>] show up as follow:



2. press [RS232 Import].show up as follow：

3. After input the file name, Press [Ok] . Than the program will into waiting for receive state. The transfer state will has shown on transfer process.



Step IV：Executing DNC software. Does the RS232 transfer setting adjust is the same controller, or controller adjust is the same of DNC software. Executing the send function of DNC software. It can be transferred into which the file of output.

Attention：

1. When operating. The file input should be doing firstly. Let controller stay on the receive file state. And then to set up transfer file function of DNC software. It can be sure transfer correctly.

3.11.2 DNC Function

DNC SOFTWARE ⇔ CONTROLLER(SYNTEC NC CONTROLER SOFTWARE)

DESCRIPTION：

When user is use the CAD/CAM software. The controller doesn't capacity in save the too large size program in working. Can use this function to execute that aside transfer and aside working.

Operation：

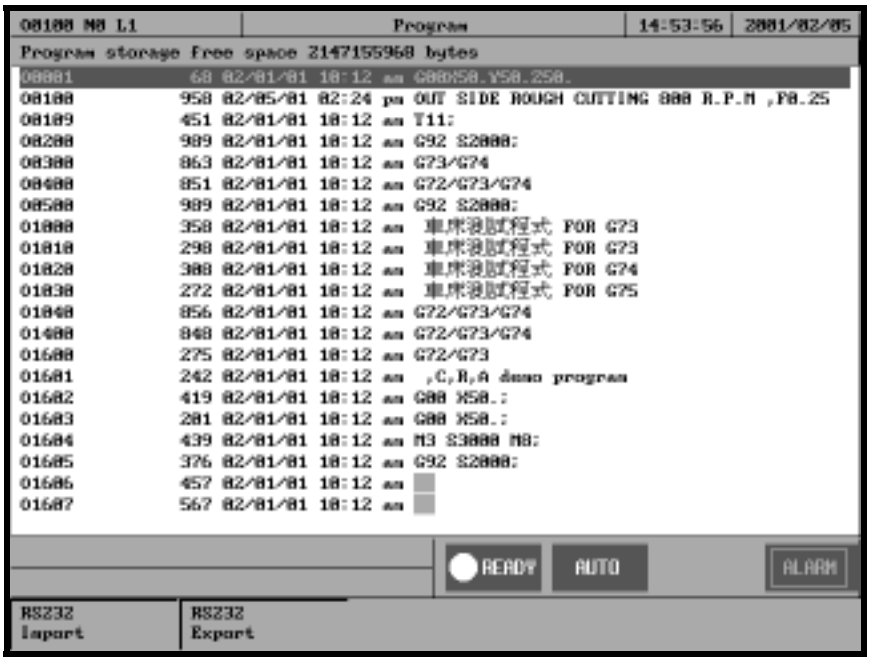
Step I：The hardware line link to COM1

Step II：Parameter of SYNTEC software system setting (Suggest value of default)

Param.	controller	meaning of parameter
3901	0	Dc protocol role, 0：CNC, 1：Device or PC
3903	1	File transfer port number(1:Com1, 2:Com2)
3921	2	Com1 baud(0:24;1:48;2:96;3:192;4:384..)
3922	8	Com1 data bit number
3923	0	Com1 exchange code type (0:ASCII;1:EIA;2:ISO)
3924	0	Com1 control code (0:No;1:DC2;2:DC4;3:DC2DC4)
3925	1	Com1 End-of-block output code(0:EOB;1:CR+EOB)
3926	0	Com1 DC3 control code parity (0:Off;1:On)
3927	2	Com1 flow control(0:No;1:CrsRts;2:XonXoff)
3928	1	Com1 parity check(0:No;1:Odd;2:Even)
3929	1	Com1 stop bit number(1:1 bit;2:2 bit)

Step III：Executing DNC software. Does the RS232 transfer setting adjust is the same of controller, or controller adjust is the same of the DNC software. Executing the send function of DNC software. It can be transferred into which the file of output.

Step IV：Turn on controller power. Into system by press [program] [file manage] to file manage submenu. Select the RS232 DNC Program is a working file. It is shown as follows.



Step Ⅴ：press “monitor”. Controller will read into file to working by RS232

Attention：

1. When operating. It should be doing that the working file from DNC software output wait controller to reading firstly. And select RS232 DNC Program is working file on controller. Finish, Setup working. It can be sure transfer correctly.

3.11.3 The software of SYNTEC controller replace the software of DNC to execute RS232 function

PC(SYNTEC software end of PC)⇔ controller

DESCRIPTION：

The SYNTEC controller software made user transfer into controller working by RS232 that CAD/CAM generate or writing file. But the SYNTEC controller software can't offer the DNC software that aside transfer and aside working function.

Operation：

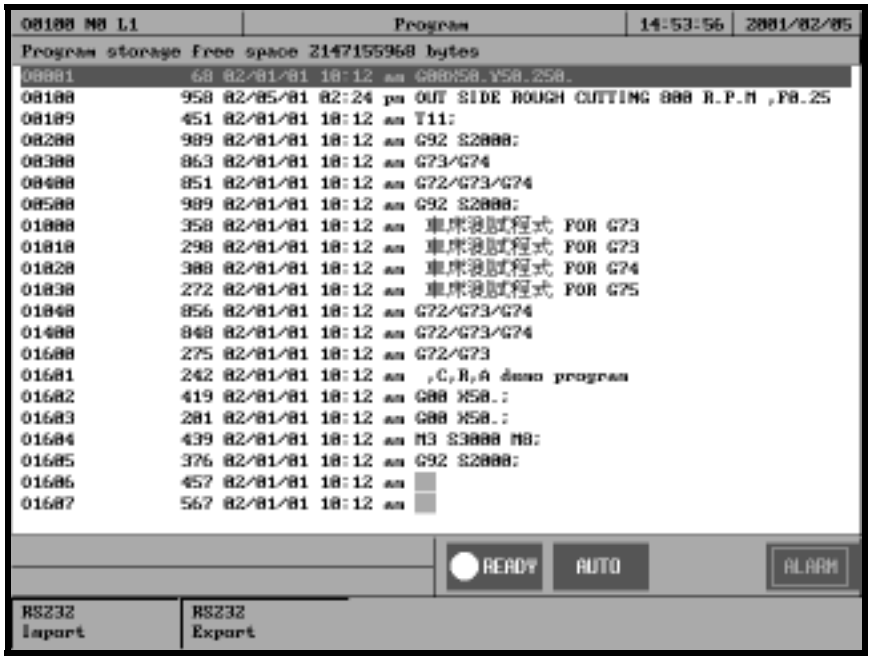
Step I：The hardware line link to COM1

Step II： Parameter of SYNTEC software system setting (suggest setting value. Exception of 3910 other all the same)

Param.	PC	controller	meaning of parameter
3901	1	0	Dc protocol role, 0：CNC, 1：Device or PC
3903	1	1	File transfer port number(1:Com1, 2:Com2)
3921	2	2	Com1 baud (0:24;1:48;2:96;3:192;4:384..)
3922	8	8	Com1 data bit number
3923	0	0	Com1 exchange code type (0:ASCII;1:EIA;2:ISO)
3924	0	0	Com1 control code (0:No;1:DC2;2:DC4;3:DC2DC4)
3925	1	1	Com1 End-of-block output code (0:EOB;1:CR+EOB)
3926	0	0	Com1 DC3 control code parity (0:Off;1:On)
3927	2	2	Com1 flow control (0:No;1:CtsRts;2:XonXoff)
3928	1	1	Com1 parity check (0:no;1:Odd;2:Even)
3929	1	1	Com1 stop bit number (1:1 bit;2:2 bit)

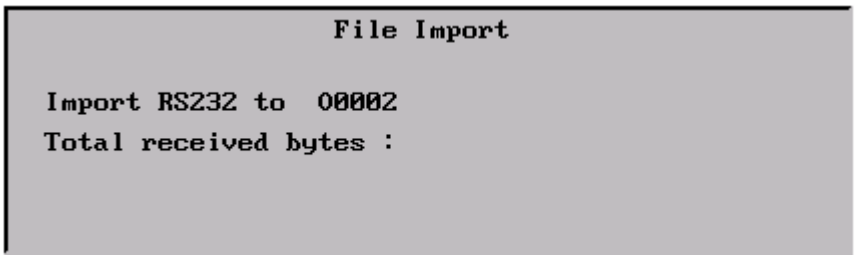
Step III：Turn on the controller power. The RS232 file input is ready.
Operation has been stated as follows.

1. Press [program] [file manage] to file manage submenu, And than to press [=>]show up as follow：



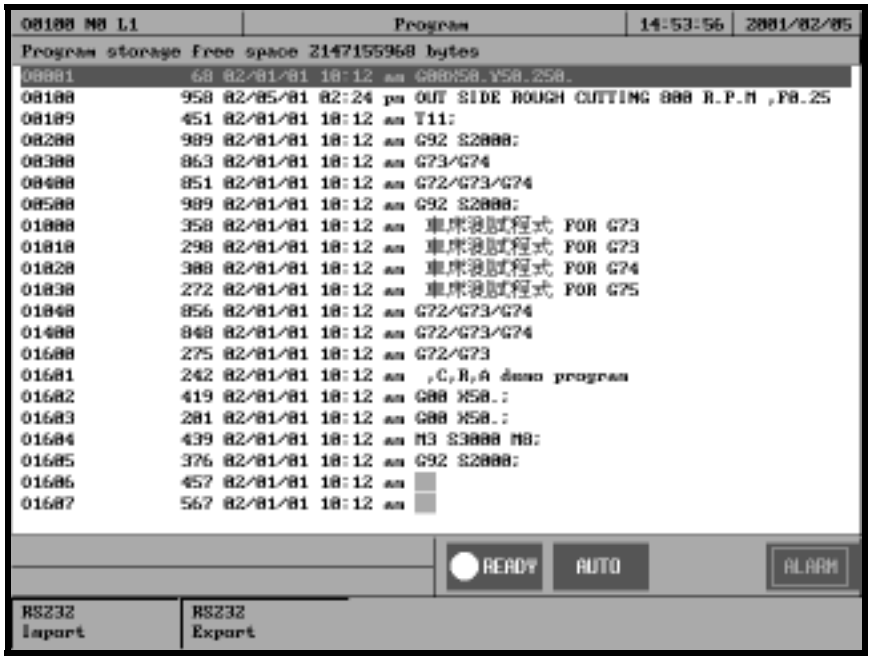
2. press [RS232 Import].Show up as follow：

3. After input the file name. Press [Ok].Than the program will enter into waiting for receive state. The transfer state will be shown on transfer process.

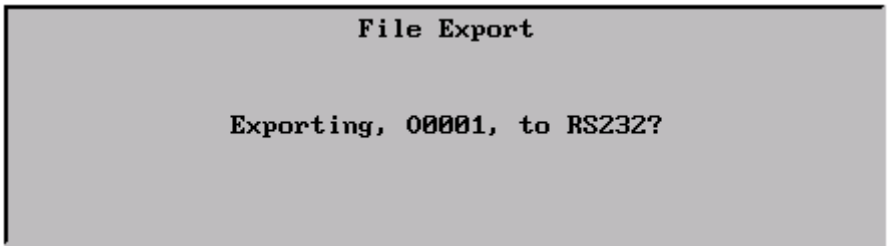


Step Iv：Upper End of PC. Executing c:\cnc\cnc into the controller simulation software. Does RS232 file output. Operation has been stated as follows.

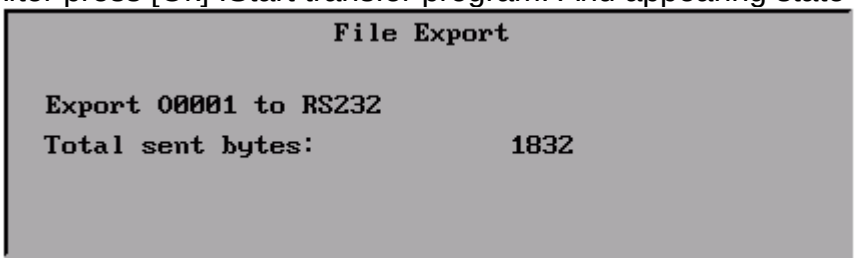
1. Press [program] [file manage] to file manage submenu. Than to press [=>] key to appearing. And press up/down key select that want output of file. Such as O1000.



2. press[RS232 Import].show up as follow：



3. After press [Ok] .Start transfer program. And appearing state of transfer.



Attention：

1. When operating. The file input should be doing firstly. Let controller stay on the receive file state. And to setup transfer file function of DNC software. It can be sure transfer correctly.

2. To be careful. SYNTEC controller software can offer RS232 file transfer function. But can't offer DNC software that aside transfer and aside

working function.

3. The most our must be careful. When PC executing. We suggest that add up /M /P parameter behind batch of cnc. Let controller software can execute simulation controller of function on PC. The /M is operate plate of simulation. Let user can setup working from keyboard. The /P will simulate ISR and cover off Interrupt. Convenient software can execute upper the Windows. **But will be invalid of transfer function if behind batch of cnc add up /P parameter.**