SwanSoft NANJING SWANSOFT

SWAN NC SIMULATION SOFTWARE

FANUC SYSTEM

INSTRACTION OF OPERATION AND PROGRAMMING

Nanjing Swan Software Technology Co.,Ltd.
Version 07/2006

PREFACE

Nanjing Swan Software Technology Company specialized in visualized software, mainly provides following services: CAD/CMD NC simulation， popularization and application of UG’s key technology.

Oriented to factory’s product research and innovation, our company supply customers with services which are highly in accordance to their individual demands i.e.overal design of product, technique consulation, quadratic research(second development service). We also develope CAD&CAM software , numerical-cotrolled system, and the technolgy of surface simulation. Besides, we provide UG-software-based quadratic research service, which can help companys establish their own strandard design prosedure so as to not only reduce new product’s researching period and designing cost but also improve the quality of product-design.

FANCUC, SINUERIK, MITSUBISHI,GSK,HNC,KND,DASEN,WA and processing simulation software ,developed by Nanjing Swan Software Technology Co.,Ltd are all based on both colleges’ teaching and machine factories’ manufacturing experience. By using this software, we can attain the aim of enabling students to have the experience of practical manipulation on a largely-reduced cost.

Nanjing Swan Software Technology Company

07/2007

CONTENTS

CHAPTER 1 SUMMARY OF SWAN NC SIMULATION SOFTWARE

1.1 BRIEF INTOUCTION OF THE SOFTWARE

Based on factories' manufacturing and colleges' teaching experience, Nanjing Swan Software Technology Co., Ltd developed the following software: FANUC, SIMUMERIK, MITSUBISHI, GSK, HNK, KND, DASEN, and simulation software. Through which, we can attain the aim of enabling students to have the experience of practical manipulation on a largely-reduced cost.

Swan series NC simulation software can be furthere devided in 8 major types, 28systems and 62 controlling surfaces. Equipped with FANUC, SIMUMERIK, MITSUBISHI, GSK, HNK, KND, DASEN software, swan NC simulation software can help students to learn operation of NC milling tool, lathe and machining center of each system. Meanwhlie CAM NC program can be programmed or read in by manual.By internet teaching,teachers can have the first-hand information of their students'current manipulating condition .

1.2 FUNCTION OF THE SOFTWARE

1.2.1 CONTROLER

1. The screen configrations can be realized and all the functions are the same with CNC machine used in the industrial system.
2. Interprets NC codes and edits cutting feed commands of machine real-timely.
3. Operation panels are similar with the real NC machine can be provided.
4. Single brick operation，automatic operation,editing pattern,dry running,and so on.
5. Rate of travel adjusting, change over switch of unit millimeter pulse.

Fig.1.2-1 FANUC 0-MD(milling machine)

（1）Choose the blank function key at the left tool frame

（2）Choose reference mandril.

（3）Choose ordinance of reference mandril and thickness of spacer gauge.

（4）Preset workpiecedirectly and confirm that according to special hint on the bottom-left of window.

（5）Coordinate Z workpiece nullpoint = current coordinate Z – length of reference mandril – thickness of spacer gauge.

（6）Put the output:Z、Y、X axes workpiece nullpoint into G54～G59.

Fig.1.2-2 FANUC 0-TD(lathe)

Fig.1.2-3 FANUC 0i(milling machine)

1.2.2 FUNCTON INTRODUCTION

★ The first domestic NC simulation software which can be downloaded and updated automatically for free.

★ Vivid 3DM NC machine and operation panels.

★ Support ISO-1056 preparatory function code（G code）、secondary function code（M code）and other operation codes.

★ Support system self-defining code and canned cycle.

★ Callin CAD/CAM postposition tailor file such as UG、PRO-E、Mastercam directly for simulation to processing.

★ Windows macro record and playback.

★ AVI files record and playback.

★ Placement and mounting of workpiece.

★ toochange mechanical hand、square-tool rest、all direction- tool rest.

★ rectifying tool by benchmark、rectifying tool by test cutting .

★ Components cutting， with processing coolant、processing sound、scrap iron and so on.

★ Tools such as edge detector、spacer gauge、micrometer、caliber rule.

★ Adopt data base management tools and performance parameter library.

★ There are many kinds of tools.

★ Support custom-defined tool function.

★ 3DM measurement function of processed model.

★ Measurement of components roughness based on cutting parameter of tools.

CHAPTER 2 OPERATIONS OF SWANSC NC SIMULATION SOFTWARE

2.1 STARTUP INTERFACE OF THE SOFTWARE

2.1.1 STARTUP INTERFACE OF PROBATIONAL VERSION

Fig. 2.1-1

（1）Choose PROBATIONAL VERSION in the left document frame.

（2）Click the left window to choose NC system needed.

（3）You can also select Super Demo if needed.

（4）Click Try It to login system interface after choose one system.

2.1.2 STARTUP INTERFACE OF NETWORK VERSION

Fig. 2.1-2

（1）Choose NETWORK in the left document frame.

（2）Choose the name of system needed in the top bar-frame at right.

（3）Choose your custom name and input password in the below tow frames.

（4）Choose between Remember Me and Remember My Password.

（5）Input the IP address of server.

（6）Click Sign in to login system interface.

（7）Startup SSCNCSRV.exe to login the main interface of SERVER,as the following Fig. show:

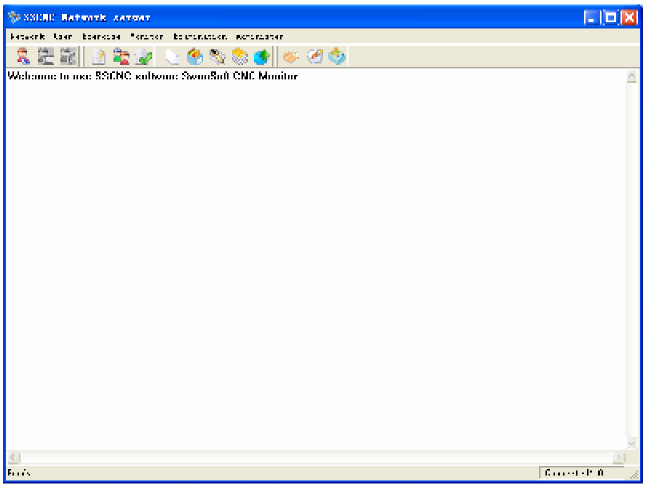


Fig. 2.1-3

（8）**After** click the icon“CUSTOM STATUS” in toolbar，it will show all the custom status,as the following graph show:

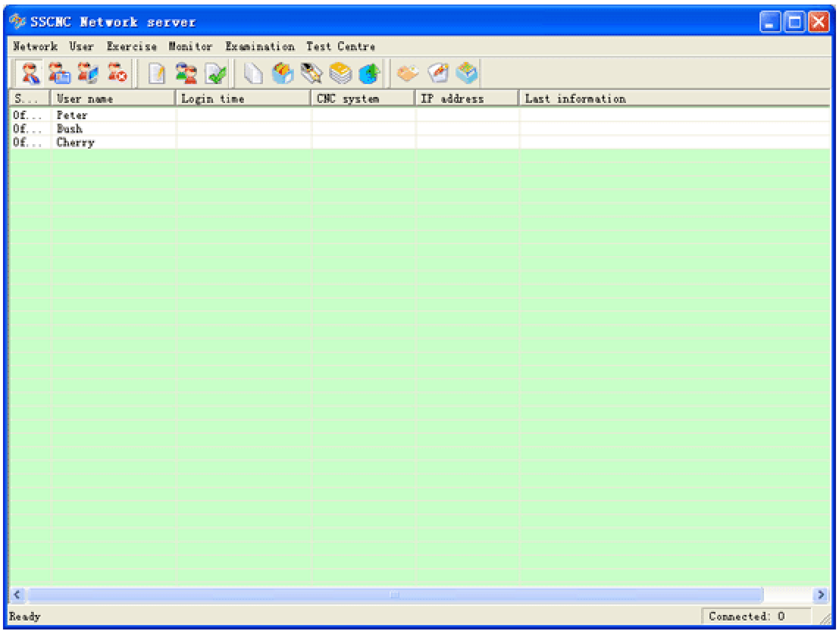


Fig. 2.1-4

（9）Choose a custom in Custom Statue List,and then click the icon "SET TEACHER’S COMPUTER" to set it Teacher’s Computer.

（10）After click the icon "CUSTOM MANAGEMENT", a dialog box " CUSTOM MANAGEMENT " will pop-up,as the following graph show:

Add custom name and its authority in the dialog box one by one or by batch.

a. In one by one pattern, input custom name ,name, secret code and code confirmation,and also you can set necessary authority then clik SAVE.

b. In batch pattern, input start numbering and number of customs, and also you can set necessary authority then clik SAVE.

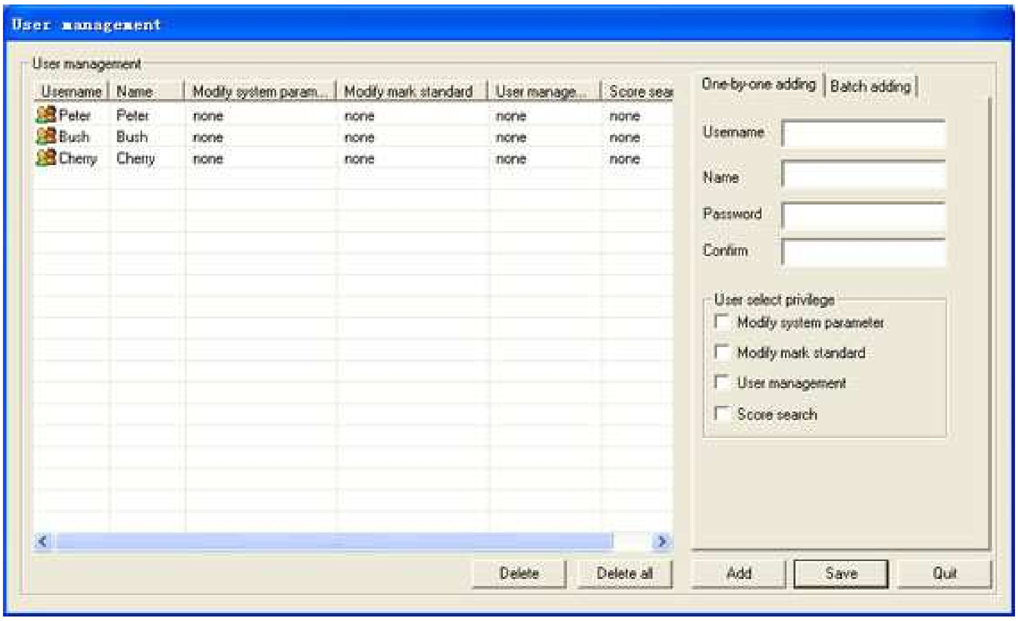


Fig. 2.1-5

2.1.3 SINGLE MACHINE VERSION STARTUP INTERFACE

Fig. 2.1-6

（1）Choose SINGLE MACHINE VERSION in the left document frame.

（2）Choose the name of system needed in the right bar-frame.

（3）Select one option between PC Encryption and Softdog Encryption.

（4）Click Run to login system interface.

2.2 SETUP OF TOOLBAR AND MENU

All the commands can be executed from the left toolbar in the window.System will show the name of its function when cursor points each button,and meanwhile the tip help of the function will be showed in the bottom statue bar.

Brief introduction of toolbar：

Setup new NC file

Open saved file(such NC file)

Save file(such as NC file)

Save as

Machine parametar

Cutter library management

Pattern of workpiece display

Choose size of workblank and coordinate of workpiece

Open/close machine door

Scrap iron display

Screen arrange： change screen arrange function by fixed sequence

Whole screen zoom up

Whole screen zoom down

Screen zoom up,zoom down

Screen translation

Screen revolve

X-Z plane selection

Y-Z plane selection

Y-X Plane selection

Machine encloser swich

Workpiece measurement

voice controler

Coordinate display

Jacket water display

Workblank display

Component display

Clarity display

ACT display

Display tools spacing number

Cutter display

Cutter path

Online help

REC parameter setup

REC start

REC stop

teaching start/stop

2.3 FILE MANAGEMENT MENU

Program file（*.NC）、tool file（*.ct）and workblank file（*.wp）callin and save and relevant function,such as the function used to open or save data file where NC code editing process is put.

Open

Open respective dialog box to choose the code file needed to disply the NC code in window. Process step into auto way automatically after whole code is loaded; Schedule of code is showed on the bottom of screen.

New

Delete NC code being edited and loaded.If code is alternated system will register that whether to save the code.

Save

Save the code edited on the screen.If execute this command to new loaded existing file nothing will be changed and system will ask for a new file name in despite of whether the file is loaded just now.

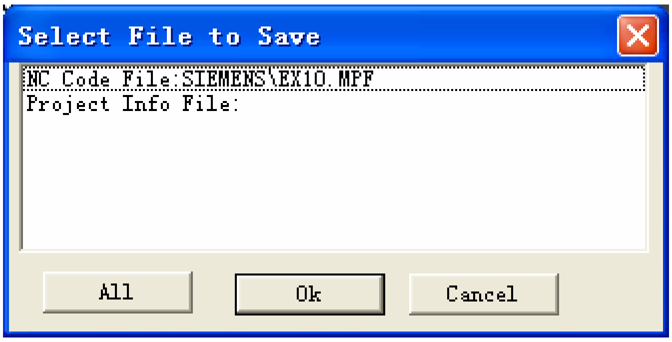


Fig.2.3-1

Save as

Save a file with a new file name known to the existing name.

Load project file

Save all the relevant data files(wp; nc; ct) into a engineering file (extension name: *.pj), called project file. This function is used to load saved file in new condition..

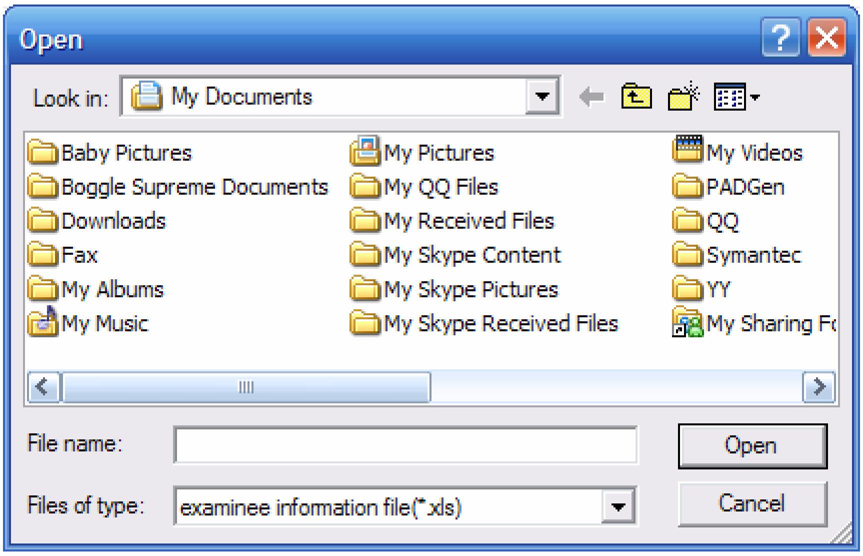


Fig.2.3-2

Project file save

This function save all the handled data into file.The blamx block on screen can be modified.

2.3.1 MACHINE PARAMETER

a. Machine parameter setup:

Drag dieblock of diago box"Parameter Setup"to choose appropriate toochange rate.

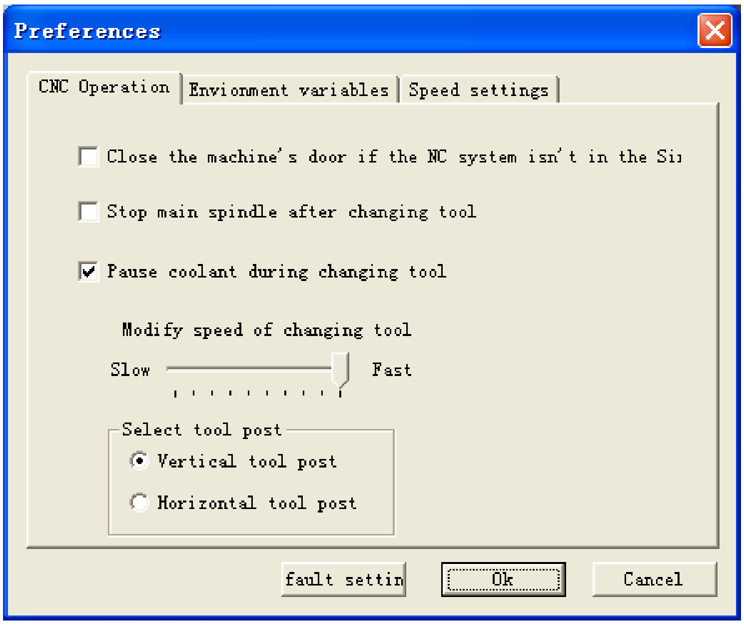


Fig.2.3-3

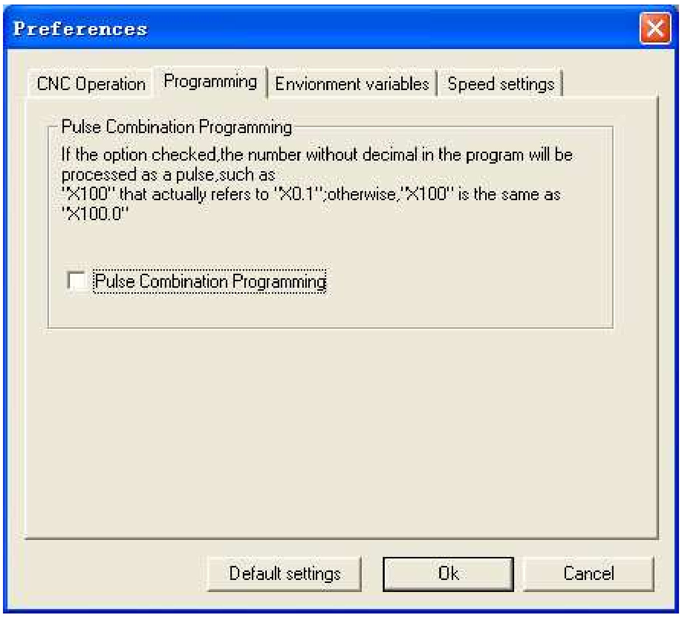


Fig.2.3-4

Click“Color Choose”to change background color of machine.

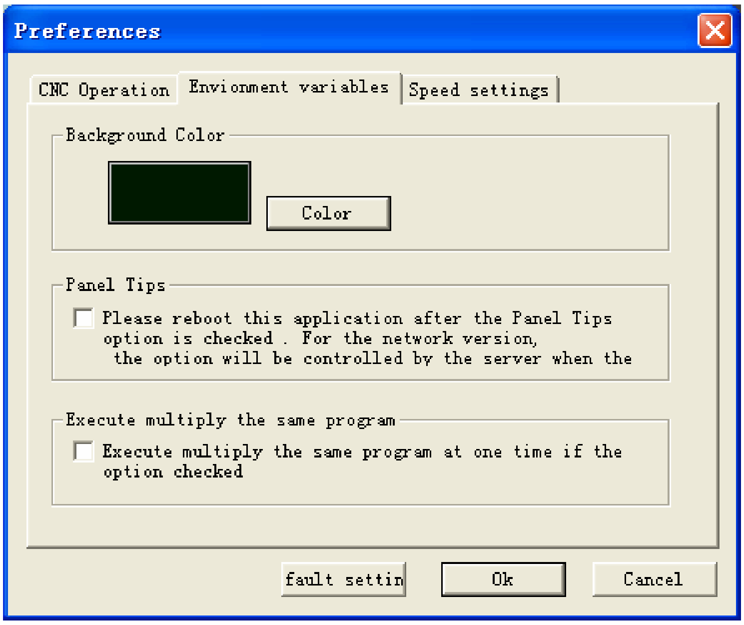


Fig.2.3-5

Adjust“Processing Drawing Display Acceleration”and“Display Precision”to gain appropriate speed of service of simulation software.

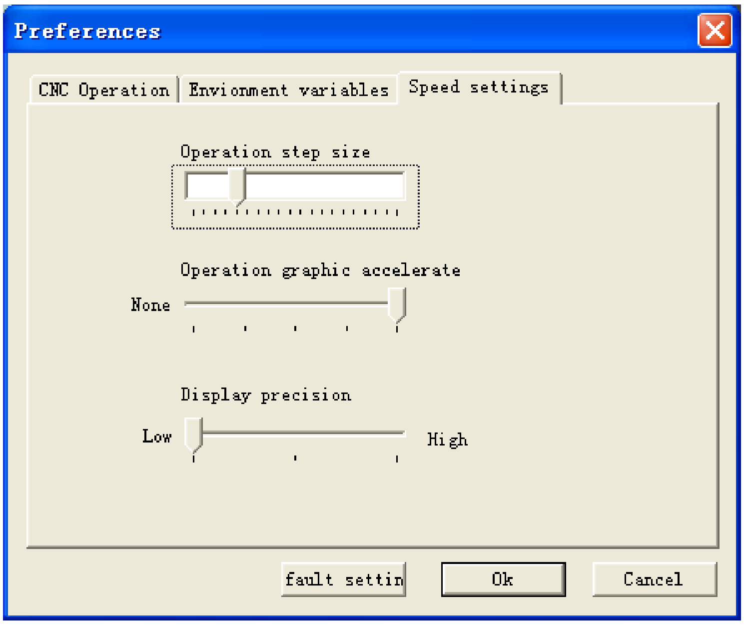


Fig.2.3-6

b.Display color：

Click “Confirm” after choose feeding route and color of machineing.

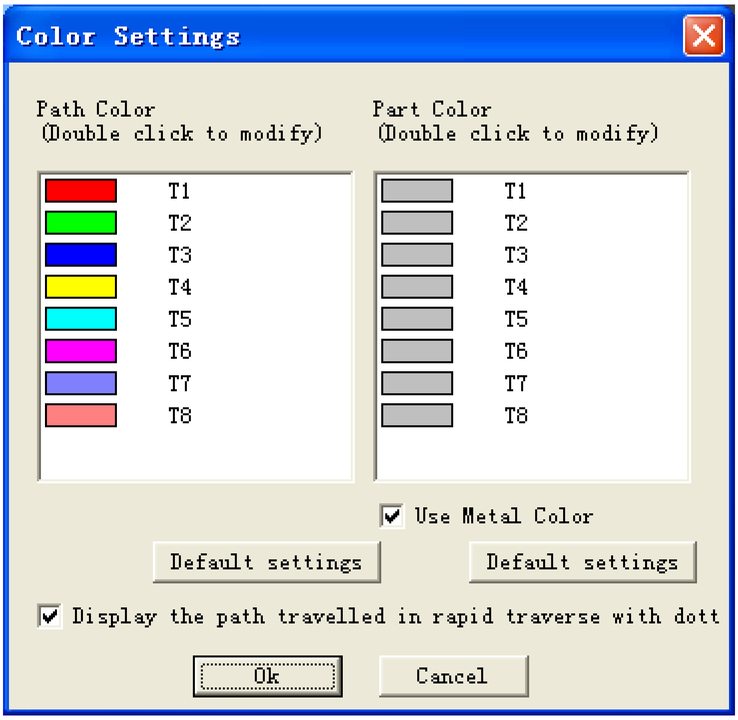


Fig.2.3-7

2.3.2 CUTTER MANAGEMENT

a. Milling machine

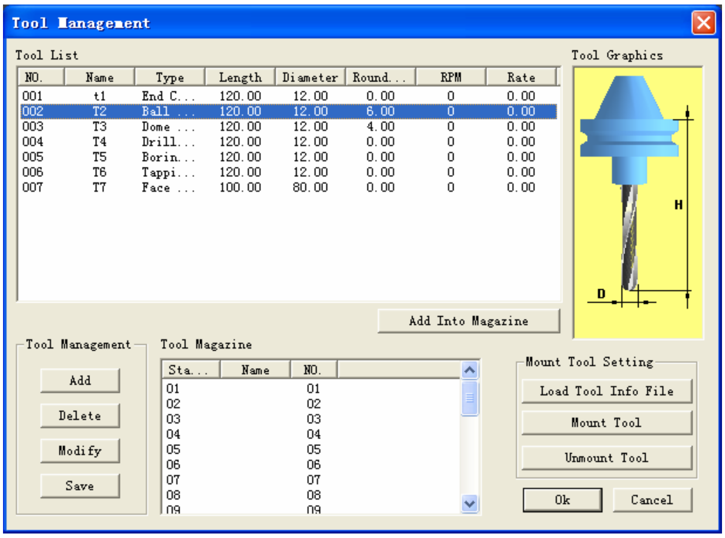


Fig.2.3-8

Add

(1).Input the number of tool

(2).Input the name of tool

(3). End-milling tools、buttonhead tools、dome-end tools、aiguilles、boring tools can be choosed.

(4). Diameter、length of tool hoder、rotation rate、cutting feeding rate can be defined.

(5).Click“Confirm”to add them to tool management library.

Add tool to chief axes

(1).Choose the tool needed in the tool data-base, such as tool “01”.

(2).Press mouse left key and hode it, then pull it to machine library.

(3).Add to top rest, then click “confirm”.

b.lathe

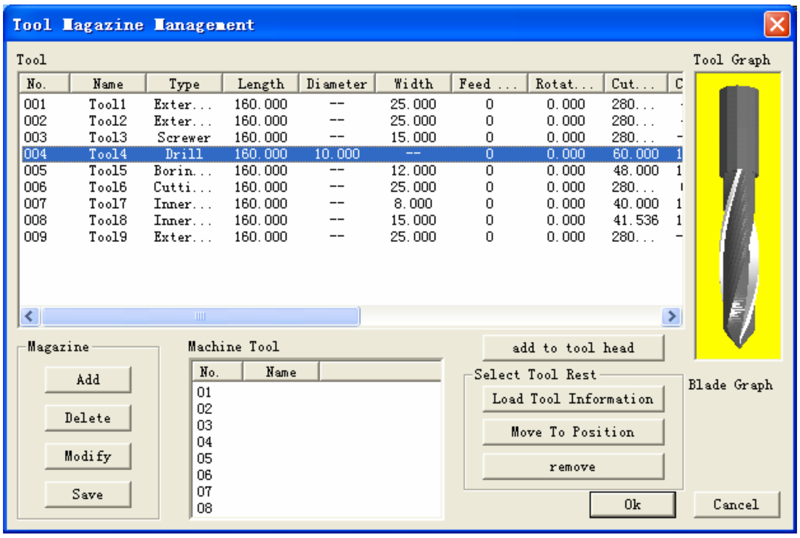


Fig. 2. 3-9

add

(1). Input the number of tool.

(2). Input the name of tool.

(3). billmpse tool、cutting off tool、internal tool、aiguille、boring tool、screw tap、screwthread tool、internal screwthread tool、internal circle tool can be choosed.

(4).Many kinds of cutting blade、side length of cutting blade、thicknesscan be defined.

(5). Click“Confirm”to add them to tool management library.

Internal circle tool adding：

(1)Click“add”，popup diago box“add tool”，as the fowing graph show：

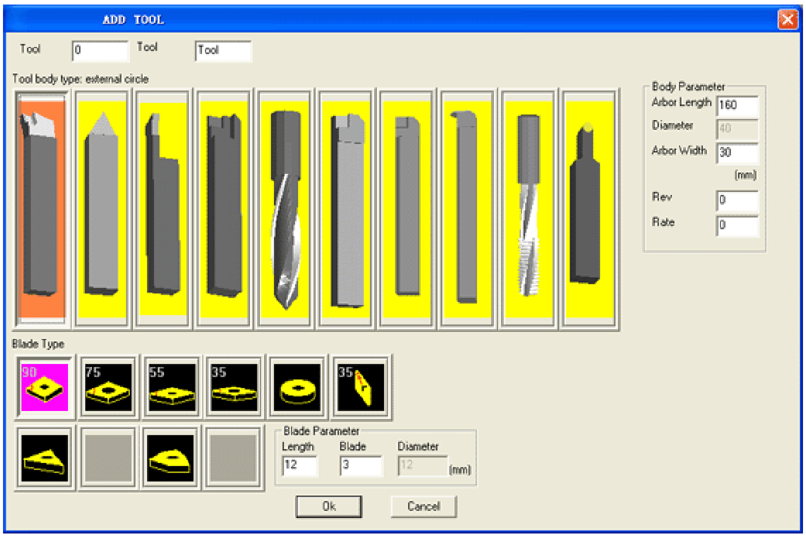


Fig. 2.3-10

(2)Choose bull-nose tool in diago box“add tool”，then popup “tool”， as the fowing graph show：

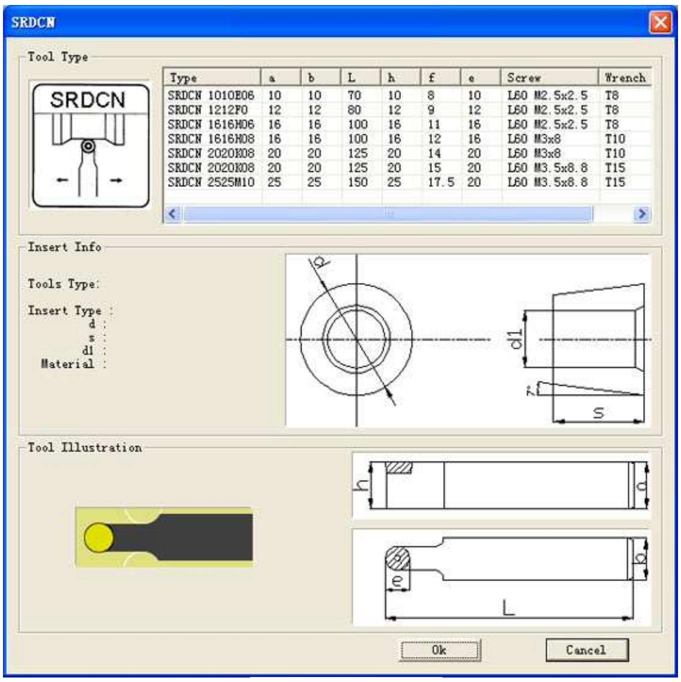


Fig. 2.3-11

(3)Choose the tool needed in diago “tool” and click “confiem”, then reverse back to “add tool”to input the number of tool and the name of tool.

Add tool to chief axes

(1) .Choose the tool needed in the tool data-base, such as tool “01”.

(2). Press mouse left key and hode it, then pull it to machine library.

(3). Add to top rest, then click “confirm”.

2.3.3 WORKPIECE PARAMETER AND ACCESSORY

a. milling machine

Size of workblank、coordinate of workpiece

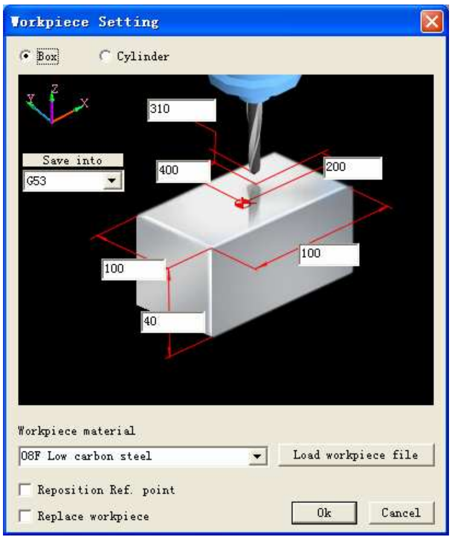


Fig. 2.3-12

(1)Define the length ,width and highness of workblank and its material.

(2)Define orgin of workpiece X、Y、Z.

(3)select changing machining orgin、changing workpiece.

b.Lathe

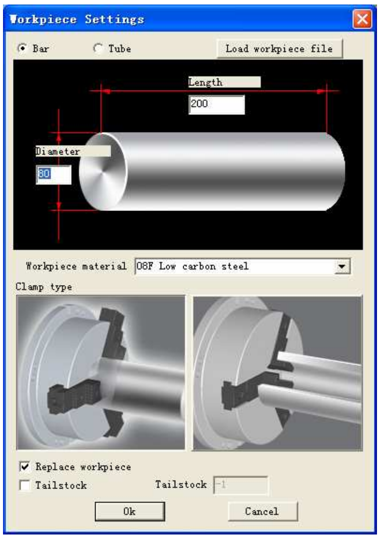


Fig. 2.3-13

(1)Define workblank type, length, diameter and its material.

(2)Define fixture.

(3)Choose tailstock.

Choose workholding fixture

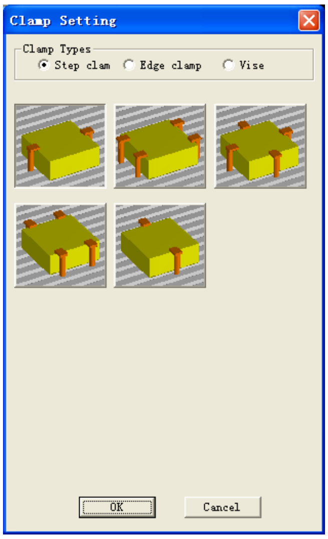


Fig. 2.3-14

Workpiece placement

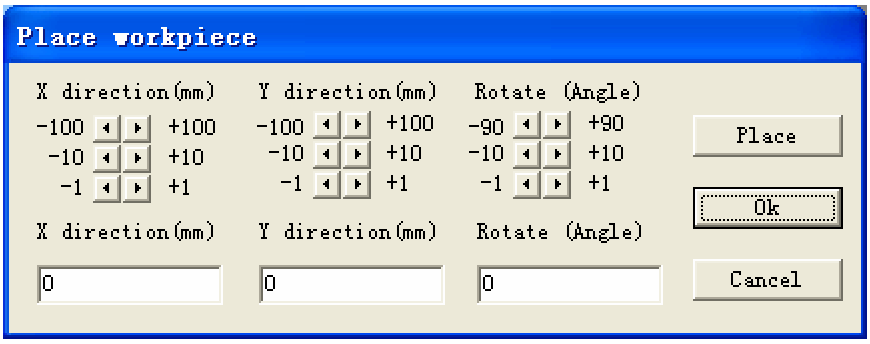


Fig. 2.3-15

(1)Choose the placement of direction X.

(2)Choose the placement of direction Y.

(3)Choose the placement of angle.

(4)Press“Place”and“Confirm”.

Edge detector measures null point of workpiece, so choose the edge detector needed in model list.

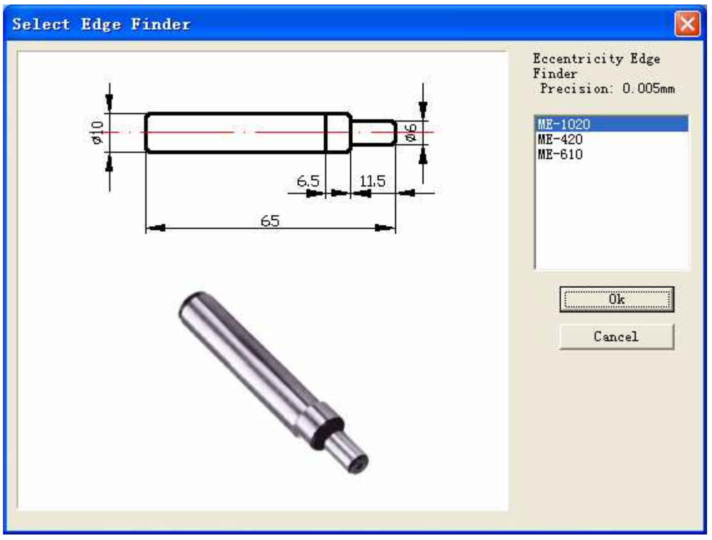


Fig. 2.3-16

Coolant pipe adjusting

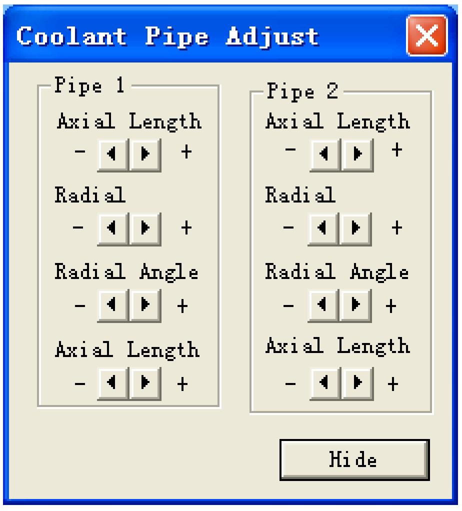


Fig. 2.3-17

2.3.4 RAPID SIMULATIVE MACHINING

(1)Programme by EDIT.

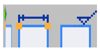
(2)Choose tool.。

(3)Choose workblank and workpiece null point.

(4)Placement mode AUTO.

(5)Press the key to rapid simulative machining without machining.

2.3.5 WORKPIECE MEASUREMENT



Three modes of measurement

(1)Feature point.

(2)Feature line.

(3)Distribution of roughness.

You can use Up, Down, Left and Right on keyboard to measure size, also you can input value into diago box..

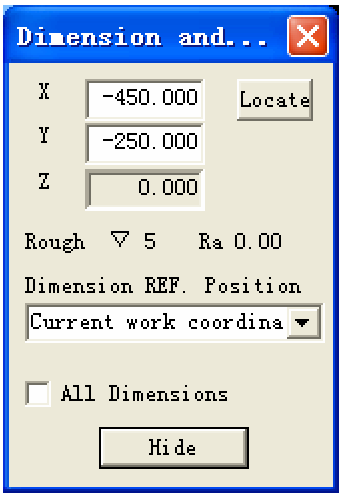


Fig. 2.3-18

2.3.6 REC PARAMETER SETUP

Three modes of REC area selection,setup as

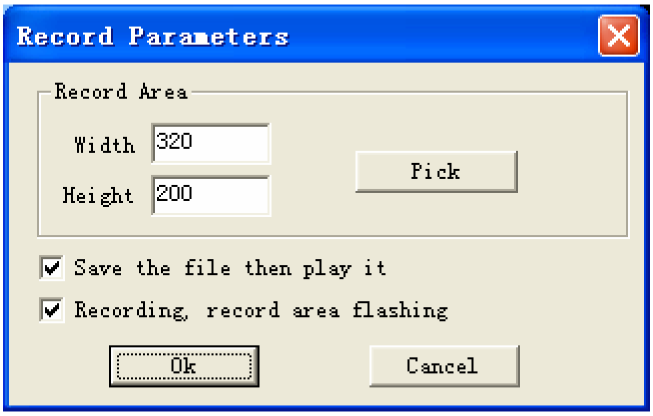


Fig. 2.3-19

2.3.7 WARING MESSAGE

Output current message files	Output all message files
Last day message	Next day message
Delete current message files	Parameter setup

When click "Parameter setup" , window "Info window parameter"will be appearance.

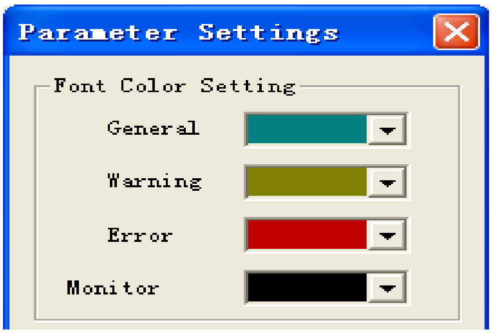


Fig. 2.3-20 Font color setup

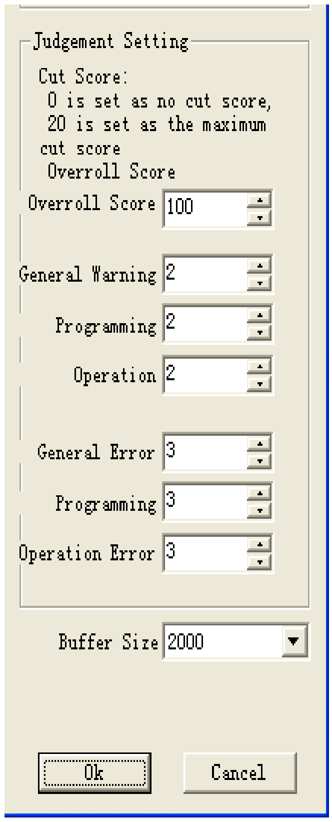


Fig. 2.3-21 Gradeing standard

1. VULGAR WARINGS

Return to reference point!

Backoff measuring piercing point bar of spindle(for milling machine only)!

Program protection is locked out, and it's unable to edit!

Program protection is locked out, and it's unable to delete program!

Modality is not booked！ Please book first!

Input format: X*** or Y*** or Z*** (FANUC measurement)!

Cutter parameter is incorrect!

There is a tool hasing this tool number, please input new tool number!

No tool hasing this tool number in top rest!

Please backoff measuring piercing point bar before auto-toochange!

Please choose the mode Auto、Edit or DNC before open file!

The file is over the Max size,so it is unable to place workpiece!

2. PROGRAMMING WARING

Search program， no O****!

Program protection is locked out, and it's unable to edit new program number!

3. MACHINE PPERATION WARING

Electric source is not opened or intense electricity is unavailable!

Spindle startup should be in JOG、HND、INC or WHEEL mode!

Please close machine door!

Startup NCSTART， then switch to AUTO、MDI、TEACHING or DNC mode!

4. VULGAR ERRORS

Please backoff spindle measurement piercing point bar before startup NCSTART

X direction overshoot

Y direction overshoot

Z direction overshoot

5. PROGRAMMING ERRORS

General G code and cyclic program are something the matter!

No O*** in program direction!

Cutter number is on-unit!

Radius compensation register number D is on-unit!

Length compensation register number H is on-unit!

Modality O*** is not booked! It can’t be deleted!

Vice program number is inexistence in subprogram call!

Vice program number is error in subprogram call!

It is lack of value F in G code!

There is no straightaway leadingin in tool compensation!

There is no straightaway eduction in tool compensation!

6. MACHINE OPERATION ERRORS

Cutter comes up against workbench!

Measuring piercing point bar comes up against workbench!

End face comes up against workpiece!

Cutter comes up against holding fixture!

Spindle is not stared,tool collision!

Measuring piercing point bar comes up against tool!

Cutter collision! Please replace small type measuring piercing point bar or raise spindle!

Teacher sends examination questions to student, and he or she can grade it which student finish and send to teacher by Swan simulation network server. Also teacher can control the machine operation panel of student and tips of error message.

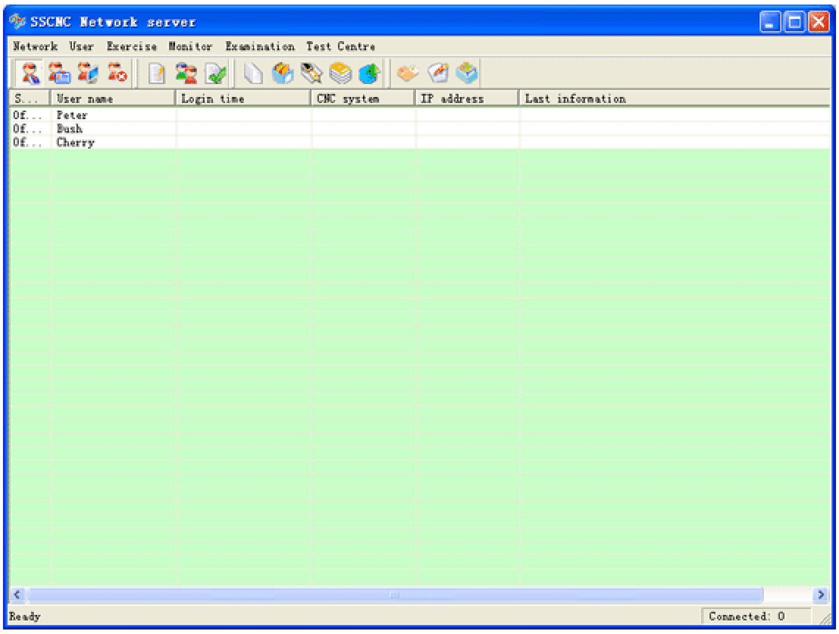


Fig. 2.3-22 Network management

CHAPTER 3 FANUC 0D OPERATION

3.1 FANUC 0D MACHINE PANEL OPERATION

Machine operation panel is on the bottom-right of window, as the followinggraph show. The panel composed with Choosing botton, Program Running Control Switch and so on is used to control the running status of machine.

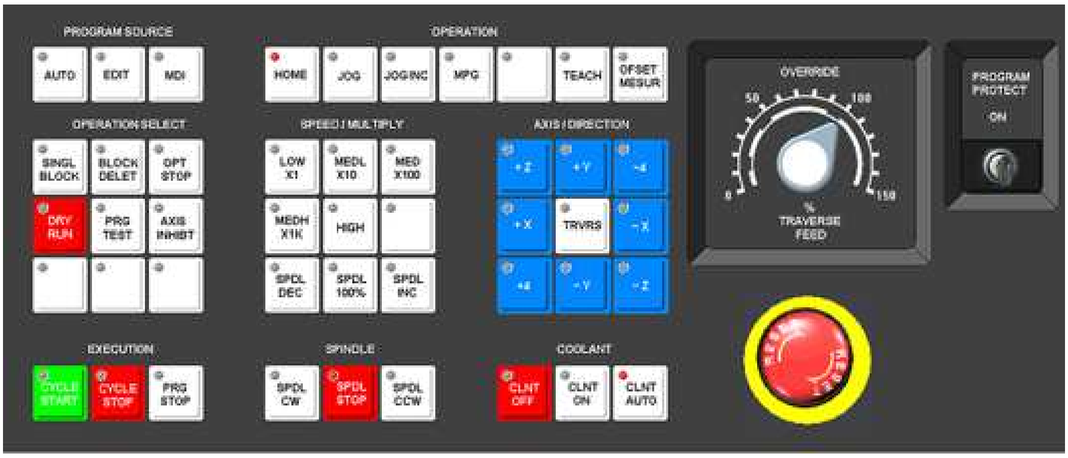


Fig. 3.1－1 FANUC 0-MD(milling machine)panel

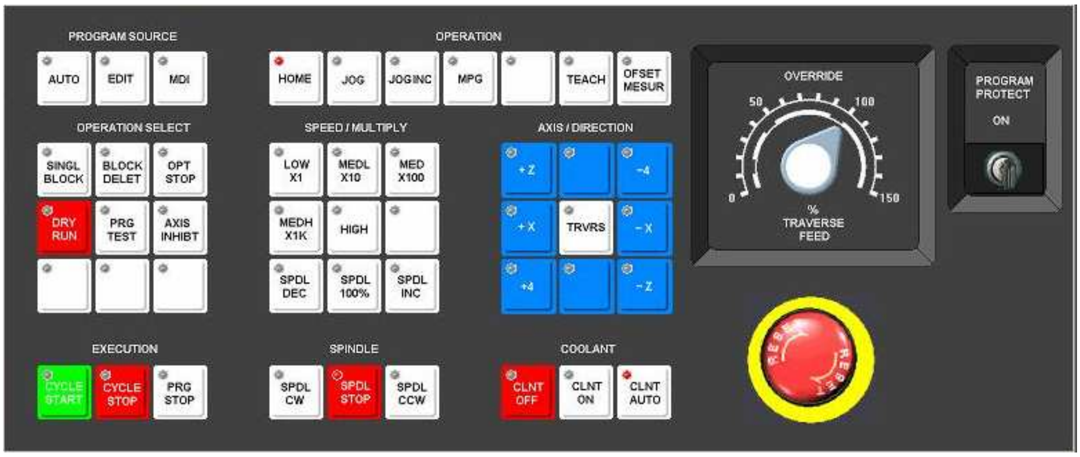


Fig. 3.1－2 FANUC 0-TD(lathe)panel

Put cursor on haft,and click mouse left key to choose mode

AUTO：Auto-machining mode.

EDIT: Input and edit NC code by operation panel directly.

MDI：Manual data input..

MPG: Move mesa or tool in hand wheel mode.

HOME: Return to reference point.

JOG：Manual mode, Move mesa or tool manually and continuously.

JOG INC：Manual pulse mode.

MPG：Rapid hand wheel mode.

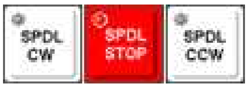
NC PROGRAM RUNNING CONTROL SWITCH

Program running startup. when pattern selection knob point to “AUTO” and “MDI” pressing is effective,otherwise ineffective.

Program running stop.Press it to stop program running when program is running.

Program running M00 stop.

MANUAL CONTROL SWITCH OF SPINGDLE

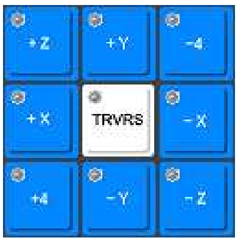


Manual starting corotation of spindle.

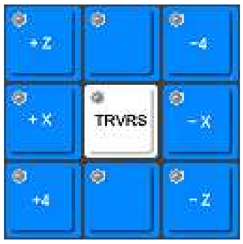
Manual starting reversion of spindle.

Manual stalling of spindle.

MANUAL MOVING MACHINE PANEL BUTTON

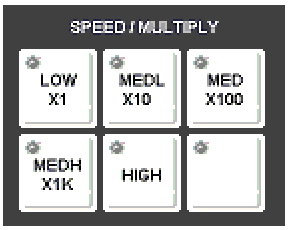


Milling machine moving button



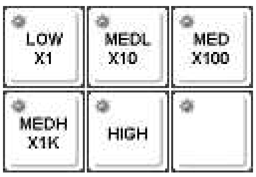
Milling machine moving button

SINGLE STEP AMOUNT OF FEED CONTROL KNOB



When you choose manual panel ,distance of every step:X1-0.001mm， X10-0.01mm，X100-0.1mm， X1kK-1mm. Put cursor on the knob and click mouse left key to choose.

INCREMENT FEEDING MAGNIFICATION CHOOSING BUTTON



When you choose moving axes of machine,the distance of every step：×1-0.001mm，×10-0.01mm，×100-0.1mm， ×1K-1mm. Put cursor on the knob and click mouse left key to choose.

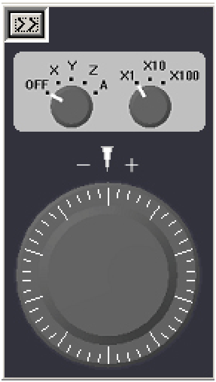
FEED-RATE(F) ADJUSTING KNOB

Adjust feed-rate in program rinning. Adjustment range: 0～150% . Put cursor on the knob and click mouse left key to choose.

SPINGDLE SPEED ADJUSTMENT KNOB

Adjust speed of spindle. Speed adjustment range:0～120%.

MANUAL PULSE



Put cursor on the knob, click mouse left key， and move your mouse cursor. When the hand wheel rotates clockwise, the machine moves along positive direction. Otherwise, on the contrary.

MACHINE LOCKING KEY

Put it at“ON”.program run， but each axes dosn’t rotate.

MACHINE BLANK RUNNING

Put it at“ON”. each axes rotates at a fixed rate.

3.2 FANUC 0D NC SYSTEM OPERATION

NC system operation keyboard is at the top right corner of window,and its program display screen is at the left. As the following graph shows:

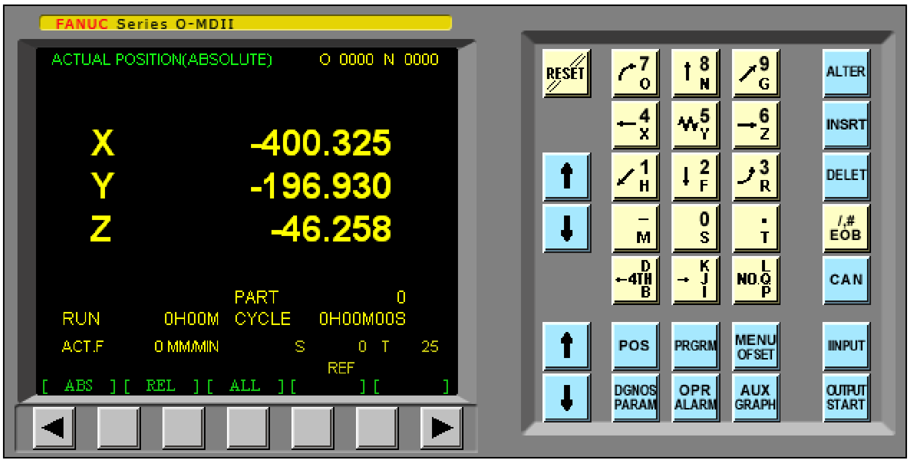


Fig. 3.2－1 FANUC 0-MD(milling machine)

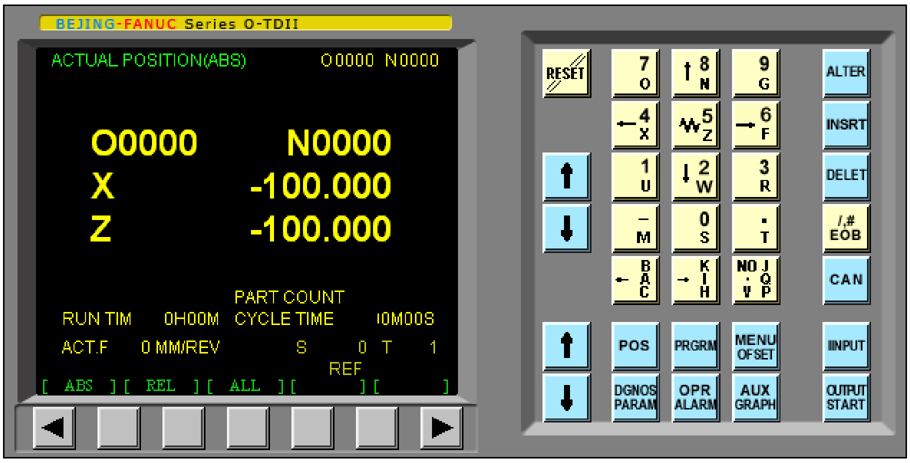
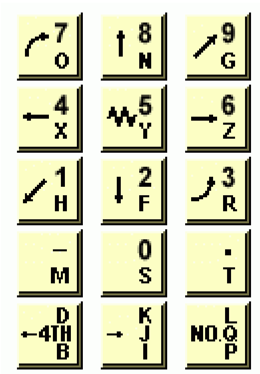


Fig. 3.2－2 FANUC 0-TD(lathe)

3.2.1 KEYSTOKE INTRODUCTION

Number/Letter key



Number/letter key is used to input data into input ragion(as the following graph shows)， and system will distinguish which to adopt, letter or number by itself.

Fig. 3.2—3

K J I Input sequence of key：K→J→I→K•••forloop.

EDIT KEY

ALTER Replace key. The data inputed replace the data curor pointing.

DELET Delete key. Delete the data curor pointing；Or delete a NC program or all the programs.

INSRT Insert key. Insert the area behind curor with data which is in the input region.

CAN Modifier. Erase data which is in input region.

PAGE SWITCH KEY

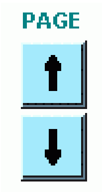
PRGRM NC program display and editing page.

POS Position display page. There are three display mode, and press button PAGE to choose.

MENU OFSET Parameter input page. First press to login coordinate setup page. Second press to login tool compensation setup page.Press button PAGE to switch different page.

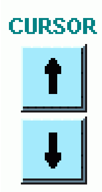
/,# EOB Withdraw and linefeed key. End input of a row of program and then feed line.

PAGE TURNING KEY（PAGE）



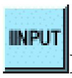
Down or up page turning.

CURSOR MOVING（CURSOR）



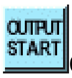
Down or up cursor moving.

INPUT KEY



Input key. Input data which is in input region in parameter page or Input a external NC program.

OUTPUT KEY



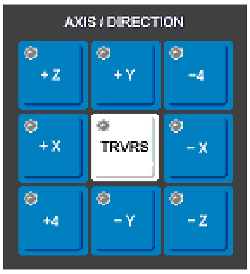
Output key. Output current NC program into computer.

Manual operation of virtual NC milling machine.

RETURN TO REFERENCE POINT

Put mode knob at“HOME”.

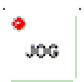
Choose each axes,and press button to reference point at once.



MOVE

There are three methods for manual moving of machine：

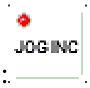
Method 1: continuously move.It is used for long-distance mesa moving.

(1) Put mode knob at “JOG”： 

(2) Choose each axes,then press direction button.Hold pressing to make the mesa move, otherwise stop.

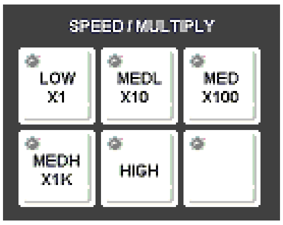
(3) Aujust travelling speed.

Method 2: Pointing control（JOG）. It is used for microadjustment,such as preseting reference operation.

(1) Put mode knob at “JOG INC”: 

(2) Choose each axes,then press the button. Mesa move one step every pressing.

(3) Adjust distance of one step by using single step amount of feed control knob.



Method 3: Use“Manual Pulse”（MPG）. It is for microadjustment. Operator can adjust working position easy by using “Manual Pulse”in practical production.

(1) Put mode knob at “MPG”：

(2) Put cursor on “Hand Wheel” 

,then hold pressing mouse for rotating. Loosen for stoping moving of machine.

(3) Adjust distance of every lattice hand wheel rotating across by using single step amount of feed control knob.

(4) Choose the axes to be moved.

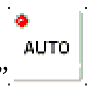
START、STOP SPINDLE

Put mode knob at “JOG”, "JOG INC"or“MPG”.

Press 

to start、stop spindle.

START PROGRAM TO MACHINEING COMPONENT

Put mode knob at “AUTO”. 

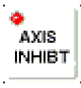
Select a NC program

Press 

in NC program running control switch.

TEST RUN

Just run program, no cutting.

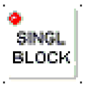
Set lock of machine at “ON” 

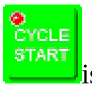
Select a NC program

Press 

in NC program running control switch.

SINGLE STEP RUN

Put single step switch at “ON” 

Every order is executed every time 

is pressed when NC program is running.

CHOOSE A NC PROGRAM

There are two methods to choose

Method 1: serch according to numbering

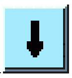
Put choosing mode at EDIT

Press 

to key in letter “O”

Press 

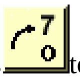
to key in number“7”.The serching number keyed in is：“O7”

Press CURSOR  to start serch； After found, “O7”displays in the place of program numbering at the corner of top right of screen, and NC program displays on screen.

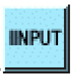
Method 2: Put choosing mode at AUTO

press 

to key in letter“O”

press 

to key in number“7”. The serching number keyed in is：“O7”

press 

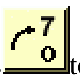
to start serch； “O7”displays at the corner of top right of screen, and NC program displays on screen.

DELETE A NC PROGRAM

Set choosing mode at EDIT

press 

to key in letter“O”

Press 

to key in number“7”

Key in the numbering of program to be deleted：“O7”

Press DELET, “O7”NC program is deleted.

DELETE ALL NC PROGRAMS

Put choosing mode at EDIT

Press PRGRM

press O 7 to key in letter“O”

Key in“9999”

Press DELET to delete all NC programs

SEARCH A SPECIFYED CODE

A specifyed code can be ： a letter or a complete code.Such as：“N0010”，“M”，“F”，“G03”and so on. Searching is processed in current NC program. The operation as the following words describe：

在 AUTO 或 EDIT

Press PRGRM

To choose a NC program

Input the letter or code to be searched

Press CURSOR ↓ to search in current NC program.

EDIT NC PROGRAM（DELETE、INSERT、REPLACE）

Set mode at EDIT

Select PRGRM

Input NC program name edited,such as“07”， and press INSRT to edit.

MOVE CURSOR

Method 1：press PAGE ↓ or ↑ to turn page.Press CURSOR ↓ or ↑ to move cursor.

Method 2:Move cursor by using the method which is used to search a specifyed code.

Input data： Click number/letter key using cursor,then the data will be inputed in input region. The key CAN is used to delete data in input region.

DELETE、INSERT、REPLACE

Press DELET to delete the data cursor specifyed.

Press INSRT to insert the area behind the code specifyed by cursor with the data in input region.

Press ALTER to replace the code specifyed by cursor with the data in input region.

INPUT NC CODE BY HAND WITH CONTROL BOX OPERATION PANEL

Put mode switch at EDIT

Press PRGRM to login program page.

Press 7 O to input“O7”-program number

Iput program name which can’t be the same with the one existent.

Press INSRT to start inputing.

Just one code can be inputed every time;The operation is similar with the operation of deleting ,inserting, replacing and so on in NC code editing.

Input sequentially after finish inputing of one line and get a new line with CRLF key /,# EOB.

INPUT A NC PROGRAM FROM COMPUTER

Set mode at DNC

Link PC and NC machine with 232 cable conductor to choose NC file for transmission.

Press PRGRM to switch to PROGRAM page.

Input program numbering“Oxxxx”

Press INPUT to reed in NC code.

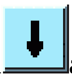
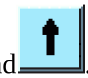
INPUT ORGIN PARAMETER OF COMPONENT

Put switch at EDIT or AUTO

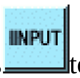
Press MENU OFSET to login parameter setup page, and then press “Workpiece”

Switch between No1～No3 and No4～No6 coordinate system page with PAGE ↓ and ↑ ,and No1~No6 and G54~G59 are one to one correspondence.

Fig. 3.2—4

Choose coordinate system with CURSORand.

Input address word（X/Y/Z）and numerical value into input region.Please consult “Input Data”operation.

Pressto input the data in input region into the specifyed place.

INPUT CUTTER COMPENSATION PARAMETER

Put mode switch at EDIT or AUTO

Pressto login parameter setup page,and then press“Redress”

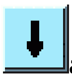
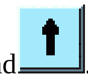
Select length compensation and radius compensation with PAGEand.

Fig. 3.2—5

Choose compensating parameter numbering with CURSOR 和 .

Input compensation value to length compensation H or radius compensation D.

POSITION DISPLAY

Press POS to switch position display page.

There are three mode for position display, and switch them by PAGE and .

Workpiece coordinate system（absolute coordinate system）position：Display tool contact points in current workpiece coordinate system.

Relative coordinate system：Display relative position preseted by operator.

Synthetic display：Display tool contact points position in following coordinate system at the same time.

Fig. 3.2－6

Position in workpiece coordinate system（ABSOLUTE）

Position in relative coordinate system（RELATIVE）

Position in machine coordinate system（MACHINE）

Residual distance in current moving order（DISTANCE TO GO）

3.2.2 MANUAL OPERATION OF VIRTUAL NC MACHINE

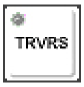
1.RETURN TO REFERENCE POINT

(1) Put mode knob at "REF.R".

(2) Press +Z +X +Y to return to reference point.

2.MOVE

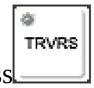
THERE ARE FOUR METHODS FOR MANUAL MOVING:

Method 1：Continuously moving（

）. It is for long distance moving.

(1) Set mode at“JOG”：

(2) Choose each axes’s direction key +X +Y +Z or–X –Y –Z，and click each key to make machine move,release for stop.

(3) Press

to get rapid moving.

Method 2：Jog（

）.It is for microadjustment,such as preseting reference point.

Set mode at“JOG”.

Method 3：Increment feeding (

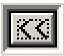
)

(1) Set mode at “JOGINC”.

(2) Choose multiplying power：×1-0.001mm，×10 -0.01mm，×100-0.1mm，×1K-1mm.

(3) Choose axes.One step per pressing.

Method 4: “Hand Pulse”using（

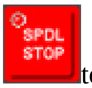
）.It is for microadjustment.It’s easy foroperator to control and observe the movement of machine. “Hand Pulse”is at the top-right corner of software interface ，and it is emergent after click.

3.START、STOP SPINDLE

(1) Put mode knob at“JOG” 

.

(2) Press 

to start the spindle, while press 

to stop the spindle.

4.START PROGRAM AND MACHINE COMPONENT

(1) Choose a program under “EDIT”mode or “AUTO”mode.(please consult following process)

(2) Put mode knob at“AUTO” 

(3) Press 

5.PROGRAM TEST RUN

Just run program, no cutting.

(1) Choose a program under “EDIT”mode or “AUTO”mode.(please consult following process)

(2) Put mode knob at“AUTO” 

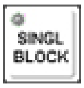
(3) Press 

（4）Press 

6.SINGLE STEP RUN

(1) Choose a program under “EDIT”mode or “AUTO”mode.(please consult following process)

(2) Put mode knob at“AUTO” 

（3）Put single step switch 

at“ON”.

（4）Just one code block is executed every time press 

when program is running.

7.CHOOSE A PROGRAM

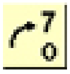
There are two methods to choose：

SEARCH ACCORDING TO PROGRAM NUMBERING

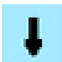
(1) Choose“EDIT”mode

(2) Press 

to input letter“O”

(3) Press 

to input nimber“7”. Search program nembered“O7”.

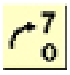
(4) Press cursor  to search； After found，“O7”is showed at the top right corner of screen，and“O7”NC program is on the screen.

CHOOSE AUTO 

MODE

(1) Press 

to input letter “O”

(2) Press 

to input number“7”. Input the numbering of program “O7”.

(3) Press 

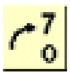
to start searching .“O7” is showed at the top ight corner of screen.

8.DELETE A PROGRAM

(1) Set mode at “EDIT”

(2) Press 

to input letter “O”

(3) Press 

to input number“7”. Key in the numbering of program to be deleted “O7”.

(4) Press DELET to delete “O7”NC program.

9.DELETE ALL PROGRAMS

(1) Set mode at “EDIT”

(2) Press PRGRM to input letter “O”

(3) Input“-9999”

(4) Press DELET to delete all programs

10.SEARCH A SPECIFYED CODE

A specifyed code can be: a letteror a complete code. Such as：“N0010”，“M”，“F”，“G03”and so on. Searching is processed in current program. The operation step ：

（1）Set mode at“AUTO”or“EDIT”

（2）Press PRGRM

（3）Choose a NC program

（4）Input the needed letter or code, such as “M”，“F”，“G03”

（5）Press CURSOR：↓ to search in current program.

11. EDIT NC PROGRAM（DELETE、INSERT、REPLACE）

（1）Set mode at“EDIT”

（2）Select PRGRM

（3）Input edited NC program name, such as“07”. Press INSRT to edit.

（4）Move cursor：

Method 1：Press PAGE ↑ or ↓ to turn page，and press CURSOR ↑ or ↓ to move cursor.

Method 2：Use the method serching a specifyed code to move cursor.

（5）Input data：Click number/letter key by mouse. CAN is used to delete data in input region.

Delete、insert、replace：

Press DELET to delete code pointted by cursor

Press INSRT to insert the place behind code specifyed by cursor with data in input region.

Press ALTER to replace code specifyed by cursor with data in input region.

12.MANUAL INPUT OF NC PROGRAM WITH OPERATION PANEL

（1）Put mode switch at“EDIT”.

（2）Press PRGRM ， and then press LIB to login program page.

（3）Press 7 O ， and input“O7”program numbering (the numbering keyed in can’t be the same with existing numbering).

（4）Press /,# EOB → INSRT to get a newline， and then start to input program.

（5）Just one section of code can be inputted in input region when input.

（6）Press /,# EOB to finish the input of current line and get a newline ,then input sequentially.

13.INPUT A PROGRAM FROM COMPUTER

You can build a text to write NC code in computer by keyboard. But the suffix name of text file(*.txt) must be changed to *.nc or *.cnc.

（1）Choose EDIT mode， and press PRGRM to shift to program page.

（2）New a program name, and then press INSRT to login programming page.

（3）Press to open NC file under the list of computer,and the program displays on current screen.

14.INPUT COMPONENT ORGIN PARAMETER

（1）Put switch at the mode of“MDI”or“JOG”.

Press MENU OFSET to login parameter setting page， and then press “Workpiece”.

Switch between No1～No3 and No4～No6 coordinate system page by PAGE and ,and No1~No6 and G54~G59 are one to one correspondence.

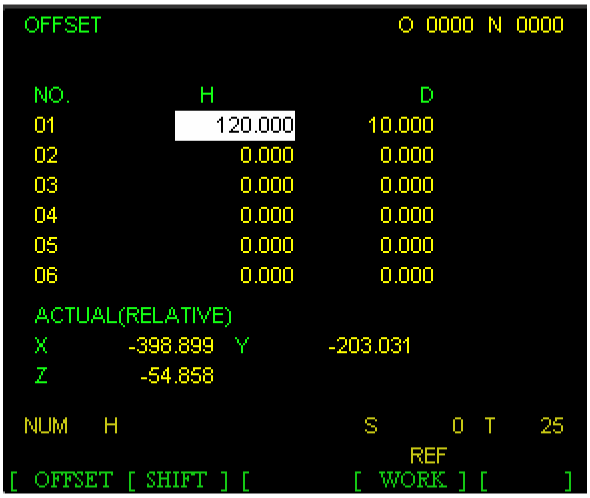


Fig. 3.2—7 workpiece coordinate systempage

（2）Select coordinate by CURSOR：  and  .

Input address letter（X/Y/Z）and numberical value to input region.

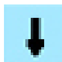
（3）Press  to input the data in input region to specifyed place.

15.INPUT CUTTER COMPENSATION PARAMETER

INPUT RADIUS COMPENSATION PARAMETER：

（1）Put mode switch at“JOG”

（2）Press  to login parameter setup page,and press “Redress”.

（3）Select length compensation , radius compensation by PAGE：  and  .

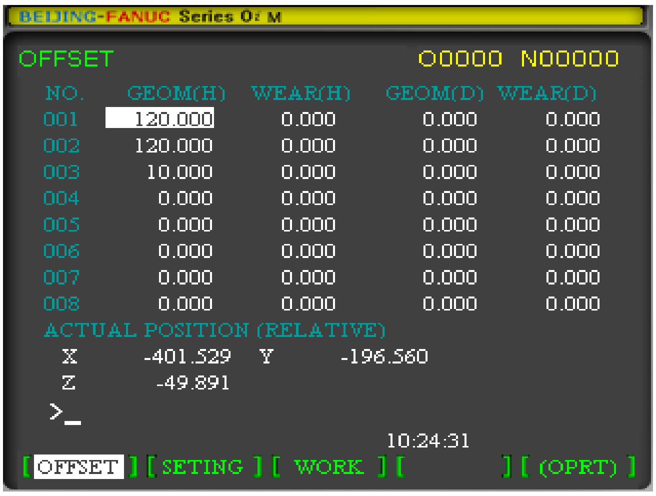
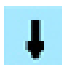
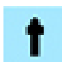


Fig. 3.2—8tool redress page

（4）Select compensation parameter numbering by CURSOR：  and  .

（5）Input compensation value to length compensation H or radius compensation D.

（6）Press 

 to input the inputted compensation value to specifyed place.

16.COORDINATE DISPLAY

Press 

 to shift to coordinate display page.There are three methods for coordinate display：

Absolute coordinate system：Display the position of machine in current coordinate system.

Relative coordinate system：Display the coordinate of machine with respect of the last position.

Synthetic display：Display positions of machine in following coordinate system at the same time.

Fig. 3.2—9

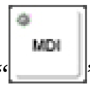
Position in workpiece coordinate system（ABSOLUTE）

Position in relative coordinate system（RELATIVE）

Position in machine coordinate system（MACHINE）

Residual distance in current moving order（DISTANCE TO GO）

17.MDI(MANUAL DATA INPUT)

(1) Set mode at“

”

(2) Press 

， and press 

 .Iput program, then press 

.

(3) Press 

 or 

 to run program.

CHAPTER 4　FANUC 0i OPERATION

4.1　FANUC 0i PANEL OPERATION

OPERATION PANEL

The operation panel of machine is on the bottom-right of window,as the following graph shows.The panel composed of pattern selection button, operation control switch and so on is used to control running status of machine mainly，as the following instruction shows：

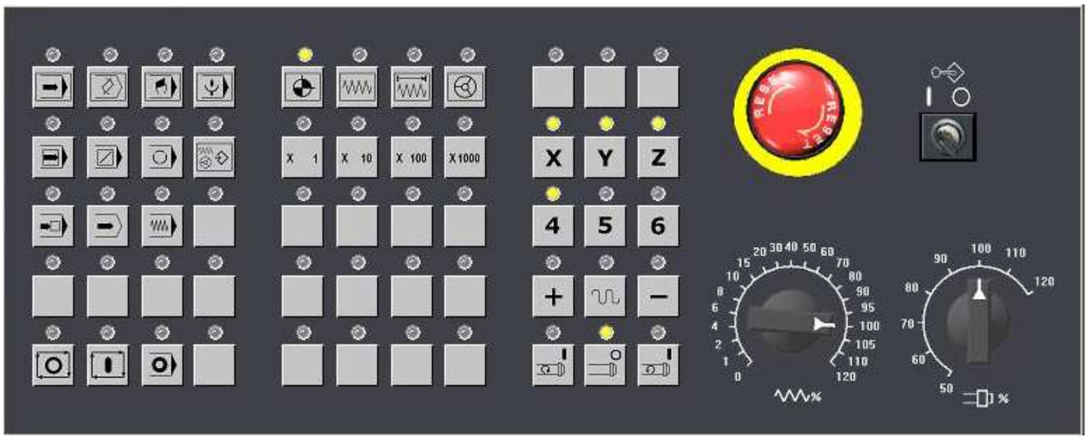


Fig. 4.1－1 FANUC 0i(milling machine)panel

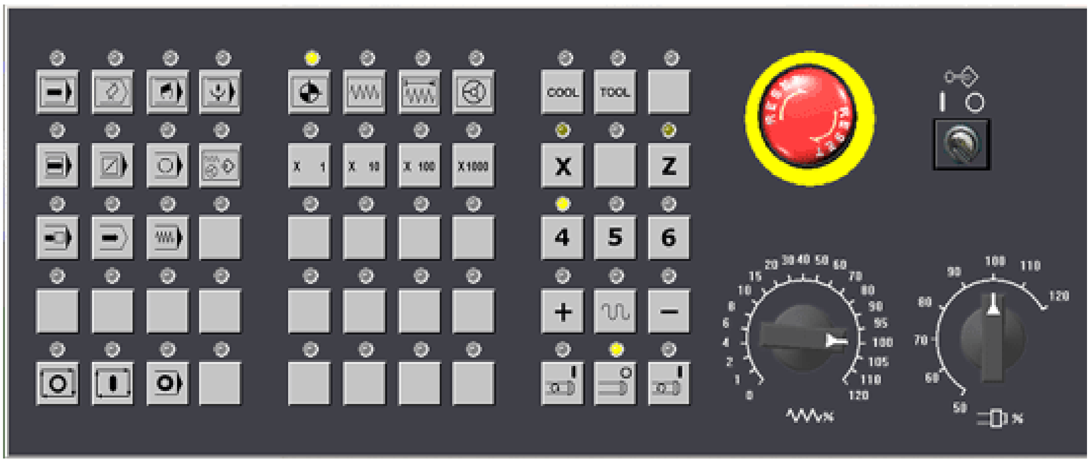
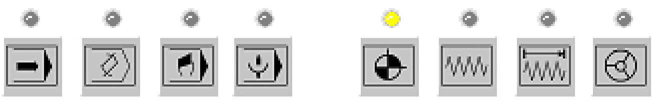
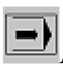
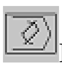


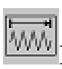
Fig. 4.1－2 FANUC 0i(lathe)panel

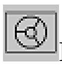


AUTO：Auto-processing mode.

EDIT：It is used to input NC program and edit code through operation panel directly 用.

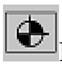
MDI：Manual Data Input.

INC：Increment feed.

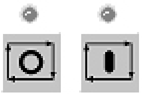
HND：Move mesa or tool in hand wheel mode.

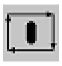
JOG：Manual mode. Move mesa or tool continuously by hand.

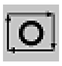
DNC：Link PC and NC machine with 232 cable conductor to select program for transmission and processing.

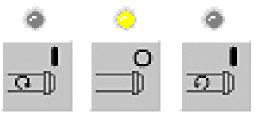
REF：Return to reference point.

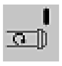
NC program running control switch

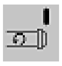


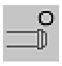
 Program run start；When put moode choosing knob at “AUTO”and“MDI”, pressing is effective, otherwise ineffective.

 Program run stop；Press it to stop running when program is running.机床主轴 Manual control switch

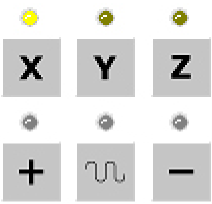


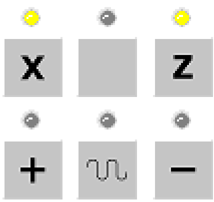
 Manual start of spingdle for corotation.

 Manual start of spingdle for reversal.

 Manual stop of spingdle

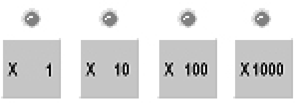
Manual moving of machine mesa





Milling machine button Lathe button

Button for singlestep feeding magnification chooseing

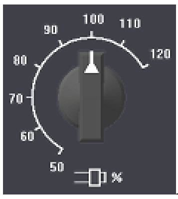


When choose mobile axes of machine,the distance of one step is：×1-0.001mm，×10-0.01mm，×100-0.1mm，×1000-1mm. Put cursor on button, and then click mouse left key to choose.

Feed rate(F) adjusting knob

Adjust feed rate in program running, range of adjusting: 0～120% . Put cursor onknob, click mouse left key for rotation.

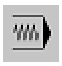
Spindle speed adjusting knob



Adjust spindle speed, range of adjusting:0～120%.

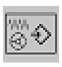
Put cursor on hand wheel to choose axial direction. Press mouse left key and move the mouse. Clockwise rotation of hand wheel for positive direction moving of the corresponding axes;anticlockwise rotation of hand wheel for negative direction moving of the corresponding axes.

Dry running of machine



Press the button, and then each axes rotate at a fixed rate.

Manual teaching

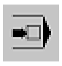
 Choose tool in tool library

Press it to choose tool.

Locking key of program editing

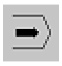
Put knob at“”to edit and modify program.

Restart program



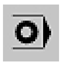
Program can be started from specifyed block after tool breakdown.

Locking key of machine



Each axes is locked and only program can be runned after press this key .

M00 Program stop



M00 stop when program is running.

Emergency stop knob

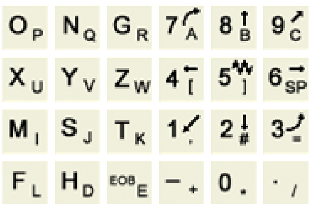
4.2 FANUC 0i NC SYSTEM OPERATION

Fig. 4.2－1　FANUC 0i (milling machine)panel

Fig. 4.2—2 FANUC 0i (lathe)panel

4.2.1 BUTTON INTRODUCTION

Number/letter key



Number/letter key is used to input data to input region（as the follwing graph shows）.System will distinguish which to adopt, number or letter.

Press  to shift input mode，for example：O—P，7—A.

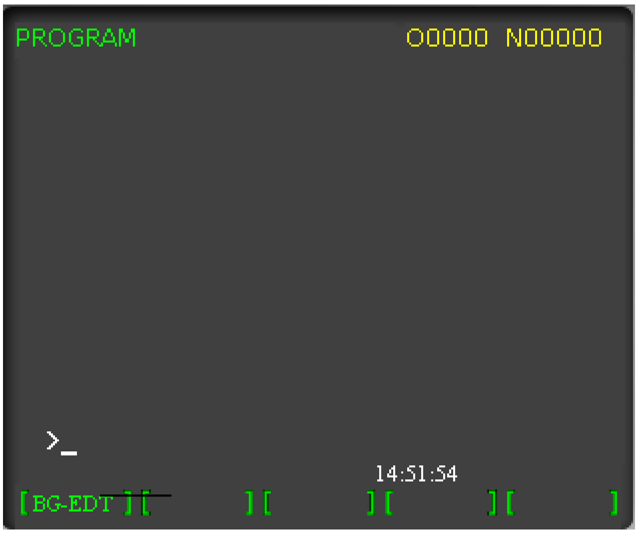


Fig. 4.2—3 FANUC 0i-M(milling machine)input of number and symble

Fig. 4.2－4 FANUC 0i-T(lathe) input of number and symble

Edit key

ALTER Replace key. The data inputed replace the data curor pointing.

DELTE Delete key. Delete the data curor pointing; Or delete a NC program or all the programs.

INSERT Insert key. Insert the area behind curor with data which is in the input region.

CAN Cancel key. Remove the data in input region.

EOB E Carriage return & line feed key.

SHIFT Upper case key.

Page shift key

PROG Program display and editing page.

POS Position display page. There are three mode for position display, and press PAGE to choose one mode.

OFSET SET Parameter input page. First press to login coordinate setup page;second press to login tool compensation page. Press PAGE to shift diferrent page.

SYSTM System parameter page **MESGE** Info page.Such“Alarm”.

CUSTM GRAPH Fig. parameter setup page.

System help page. Reset key.

Page turning button（PAGE）

Turn up Turn down

Cursor moving（CURSOR）

Move up Move left

Move down Move right

Input key Input key. Input data in input region into input parameter page.

4.2.2 MANUAL OPERATION OF MACHINE

RETURN TO REFERENCE POINT

(1) Put mode knob at .

(2) Choose axeses . Press the button to return to reference point.

Move

THERE ARE THREE METHODES FOR MANUAL MOVING OF AXES：

Method 1：Rapid moving . It is for long distance of work bench moving.

(1) Set mode at“JOG”

(2) Choose axeses. Press direction key and hold it to move machine,and release for stop.

(3) Press to make axeses move rapidly.

Method 2：Increment moving . It is for microadjustment,such as preseting reference.

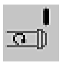
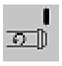
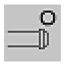
(1) Set mode at ： Choose for stepping amount.

(2) Choose axeses. Each axes move one step every time press it.

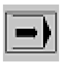
Method 3：“Hand Pulse”using . It is for microadjustment. It’s easy foroperator to control and observe the movement of machine. “Hand Pulse”is at the top-right corner of software interface ， and it is emergent after click.

START、STOP SPINDLE

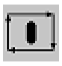
(1) Put mode knob at“JOG”.

(2) Press   to make spingdle get positive and negative rotation. Press  to stop spingdle.

START PROGRAM TO MACHINE COMPONENT

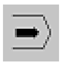
(1) Put mode knob at “AUTO” 

(2) Choose a program (please consult following process)

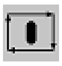
(3) Press program starting button 

PROGRAM TEST RUNNING

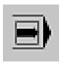
Just run program, no cutting in test running.

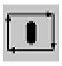
（1）Set mode at  .

（2）Press to call out program after choose a program such as O0001.

（3）Press program starting button  .

SINGLE STEP RUN

（1）Put single step switch  at “ON”.

（2）Only one command is executed, every time you press the button  when program is running.

CHOOSE A PROGRAM

There are two methods for choosing：

serch according to program numbering

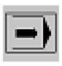
(1) Choose“EDIT”mode

(2) Press

to input letter“O”

(3) Press to input number“7”. Input the searching number：“O7”

(4) Press CURSOR： to start searching；After found, “O7”is showed at the top right corner of screen， and“O7”NC program is on the screen.

SET MODE AT AUTO 

(1)Press PROG to input letter"O"

(2)Press 7 A to key in number"7".The searching number keyed in is: "07"

(3)Press [(OPRT)] →

[O.SRH] ,and"O7"displays on the screen .

(4)You can input block number"N30", and then press [N.SRH]to search block .

DELETE A PROGRAM

(1) Set mode at "EDIT"

(2) Press PROG to input letter "O"

(3) Press 7 A to input number"7". Key in the numbering of program to be deleted "O7".

(4) Press DELTE to delete "O7"NC program.

DELETE ALL PROGRAMS

(1) Set mode at "EDIT"

(2) Press PROG to input letter "O"

(3) Input"-9999"

(4) Press DELTE to delete all programs

SEARCH A SPECIFYED CODE

A specifyed code can be: a letteror a complete code. Such as: "N0010", "M", "F", "G03"and so on. Searching is processed in current program. The operation step :

(1) Choose"AUTO" or"EDIT" mode

(2) Press PROG

(3) Choose a NC program

(4) Input the needed letter or code, such as "M", "F", "G03"

(5) Press[SRH↓]in[BG-EDT] [O.SRH] [SRH↓] [SRH↑] [REWIND]to start searching from current programs.

EDIT NC PROGRAM (DELETE、INSERT、REPLACE)

(1) Choose"EDIT" mode

（2）Choose PROG

（3）Input edited NC program name, such as“07”. Press INSERT to edit.

（4）Move cursor：

Method 1：Press PAGE: ↑PAGE **or** PAGE↓ to turn page，and press CURSOR: ↓ **or** ↑ to move cursor**.**

Method 2：Use the method serching a specifyed code to move cursor.

（5）Input data：Click number/letter key by mouse. CAN is used to delete data in input region.

(6) Input number of automatically generating block：Press OFSET SET → [SETING] as graph 4.2-5 shows. Input “1”in sequence number of parameter page ，and the edited program will generate block number automatically. (such as：N10…N20…)

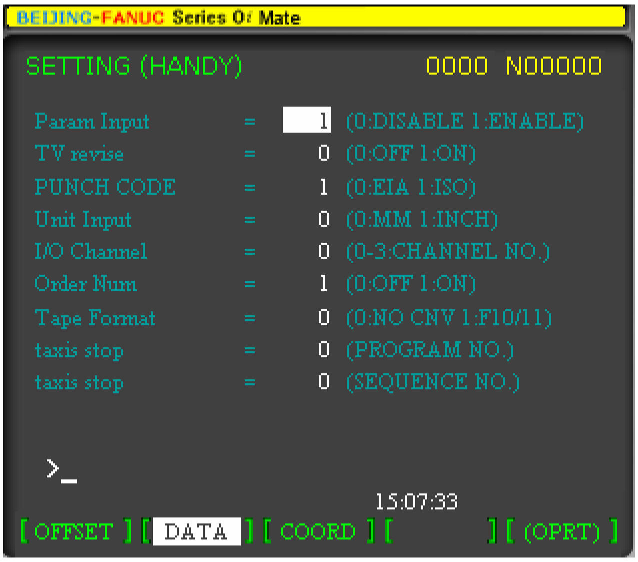


Fig.4.2－5

Delete、insert、replace：

Press DELTE to delete code pointted by cursor

Press INSERT to insert the place behind code specifyed by cursor with data in input region.

Press ALTER to replace code specifyed by cursor with data in input region.

MANUAL INPUT OF NC PROGRAM WITH OPERATION PANEL

（1）Put mode switch at“EDIT”.

（2）Press PROG， and then press DIR to login program page.

（3）Press 7 A ，and input“O7”program numbering (the numbering keyed in can’t be the same with existing numbering).

（4）Press EOB E→ INSERT to start input.

（5）Press EOB E→ INSERT to get a newline， and then start to input program sequentially.

INPUT A PROGRAM FROM COMPUTER

You can build a text to write NC code in computer by keyboard. But the suffix name of text file(*.txt) must be changed to *.nc or *.cnc.

（1）Choose EDIT mode， and then press PROG to shift to program page.

（2）New a program name, and then press INSERT to login programming page.

（3）Press to open NC file under the list of computer,and the program displays on current screen.

INPUT COMPONENT ORGIN PARAMETER

（1） Press OFSET SET to login parameter setting page， and then press“Coordinate System”。

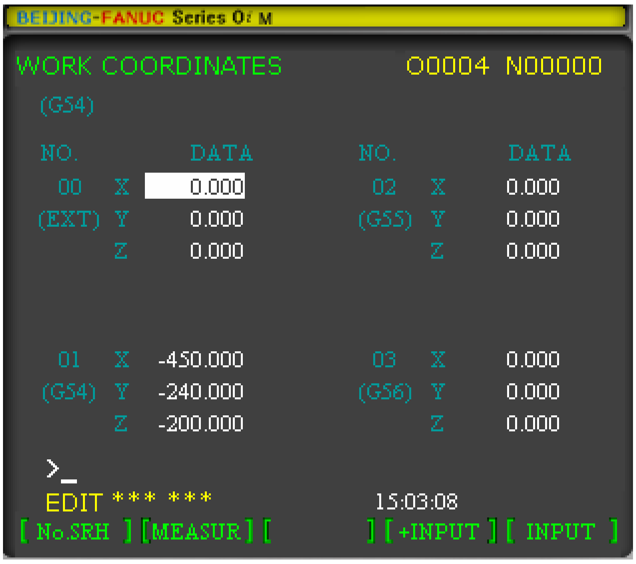


Fig. 4.2－6 FANUC 0i-M(milling machine)

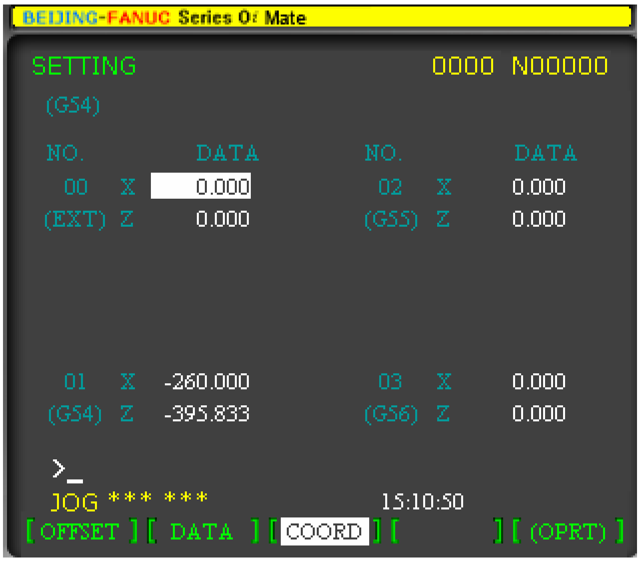


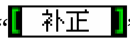
Fig. 4.2—7 FANUC 0i-T(lathe)

（2）Select coordinate by PAGE:   CURSOR: .

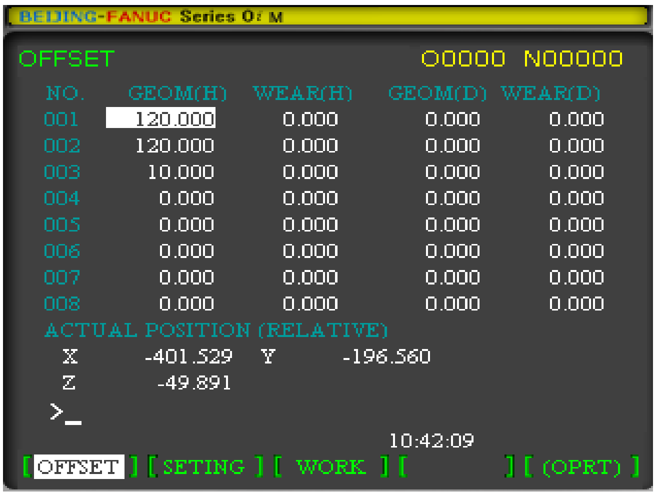
Input address letter（X/Y/Z）and numberical value to input region. Please consult“Input Data”operation.

（3）Press  to input the data in input region to specifyed place.

INPUT CUTTER COMPENSATION PARAMETER

（1）Press  to login parameter setup page,and press “Redress”“”.

（2）Select length compensation , radius compensation by PAGE:  and  .



Grapg 4.2—8 FANUC 0i-M(milling machine) tool redress page

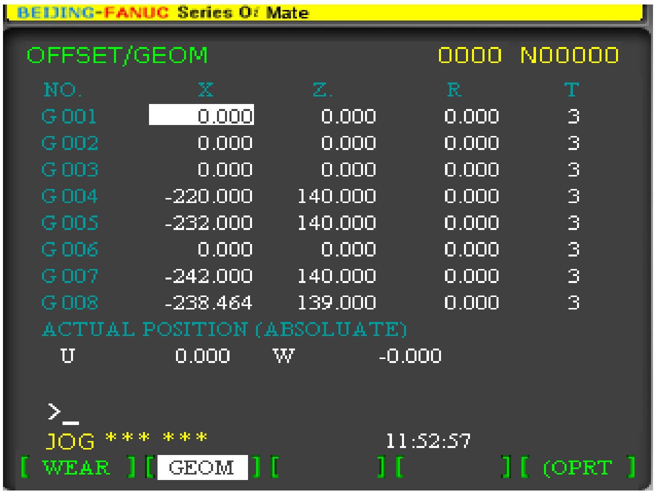


Fig. 4.2－9 FANUC 0i-T(lathe) tool redress page

（3）Select compensation parameter numbering by CURSOR： ↓ and ↑.

（4）Input compensation value to length compensation H or radius compensation D.

（5）Press INPUT to input the inputted compensation value to specifyed place.

POSITION DISPLAY

Press POS to shift to position display page.Shift by PAGE: PAGE↓ and ↑PAGE or by soft key.

MDI(MANUAL DATA INPUT)

(1) Press to shift to “MDI”mode

(2) Press PROG ，and then press MDI → EOB E to Input block number “N10”，such the input program：G0X50.

(3) Press INSERT,and the program “N10G0X50” is inputted.

(4) Press to start program.

MIRRORIMAGE FUNCTION

Press OFSET SET →[SETING]→ PAGE↓ ,as the graph 4.2-10 shows.

MIRROR IMAGE X、MIRROR IMAGE Y、MIRROR IMAGE Z and mirrorimage functions of Xaxes、Yaxes and Zaxes are one to one correspondence in parameter page.

For example: If you input“1” mirrorimage starts.

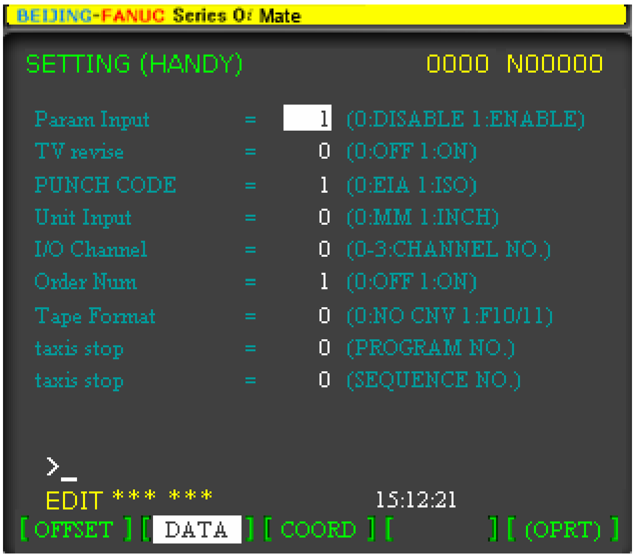


Fig. 4.2—10

POSITION OF WORKPIECE COORDINATE SYSTEM（ABSOLUTE COORDINATE SYSTEM）

Absolute coordinate system： Display the position of machine in current coordinate system.

Relative coordinate system： Display the coordinate of machine with respect of the last position.

Synthetic display： Display positions of machine in following coordinate system at the same time.

Fig. 4.2—11 FANUC 0i-M(milling machine) Fig. 4.2—12 FANUC 0i-T(lathe)

Position in workpiece coordinate system（ABSOLUTE）

Position in relative coordinate system（RELATIVE）

Position in machine coordinate system（MACHINE）

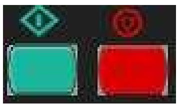
Residual distance in current moving order（DISTANCE TO GO）

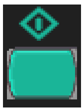
CHAPTER 5 FANUC 18i OPERATION

5.1 FANUC 18i PANEL OPERATION

Machine operation panel is on the bottom-right of window, as the followinggraph show. The panel composed with Choosing botton, Program Running Control Switch and so on is used to control the running status of machine.Detail instruction of every part is as the following words describe:

Mode choosing： EDIT /MDI （manual data input） /JOG （auto） /INC increment feeding /AUTO （automatic cycle） /REF （return to reference point）

 **PROGRAM RUN CONTROL SWITCH**

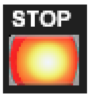
 Program run start；when pattern selection knob point to “AUTO” and “MDI” pressing is

effective,otherwise ineffective.


Program run stop； Press it to stop program running when program is running.

MANUAL CONTROL SWITCH OF SPINGDLE

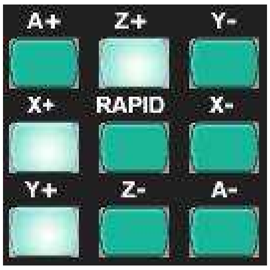
Manual starting corotation of spindle.



Manual starting reversion of spindle.

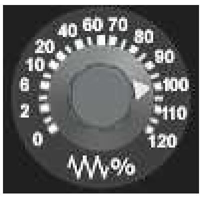
Manual stalling of spindle.

MANUAL MOVING MACHINE PANEL BUTTONS



INCRESMENT FEEDING MAGNIFICATION CHOOSING BUTTON

When you choose moving axes of machine,the distance of every step： 1 - 0.001 mm， 10 - 0.01 mm， 100 - 0.1 mm， 1000 - 1 mm， 10000 - 10 mm. Put cursor on the knob and click mouse left key to choose.



FEED-RATE(F) ADJUSTING KNOB

Adjust feed-rate in program rinning. Adjustment range: 0～120% . Put cursor on the knob and click mouse left key to choose.

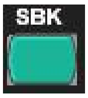
SPINGDLE SPEED MAGNIFICATION ADJUSTING KNOB

Adjust speed of spindle. Speed adjustment range:0～120%.

MANUAL PULSE

Make button 

be at effective statue. Put cursor on the knob, click mouse left key， and move your mouse cursor. When the hand wheel rotate clockwise, the machine move along positive direction.Otherwise on the contrary.



SINGLE STEP SWITCH

One command is executed every time you press it.

BLOCK SKIP

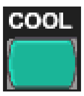
Press the button in Auto mode ,and all the program which has “ / ” in front of itself will be skiped over.

PROGRAM SELECTION STOP

Stop selecting when meet M01 program in Auto mode.

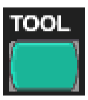
MACHINE TOO DRY RUNNING

Press it and each axes rotates at a fixed rate.



COOLANT SWITCH

Press the button to open the coolant;Press again to close.



CHOOSING CUTTER IN CUTTER LIBRARY

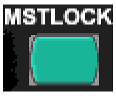
Press it to choose tool.

LOCKING KEY OF PROGRAM EDITING

Put it at“  ”to edit or modify program.

机 LOCKING KEY OF MACHINE

Each axes is locked and only program can be runned after press this key .



M ST LOCK

M 、S 、T code in program will not be xecuted when the button is in effective statue in program running.

EMERGENCY STOP KNOB

5.2 FANUC 18i NC SYSTEM OPERATION

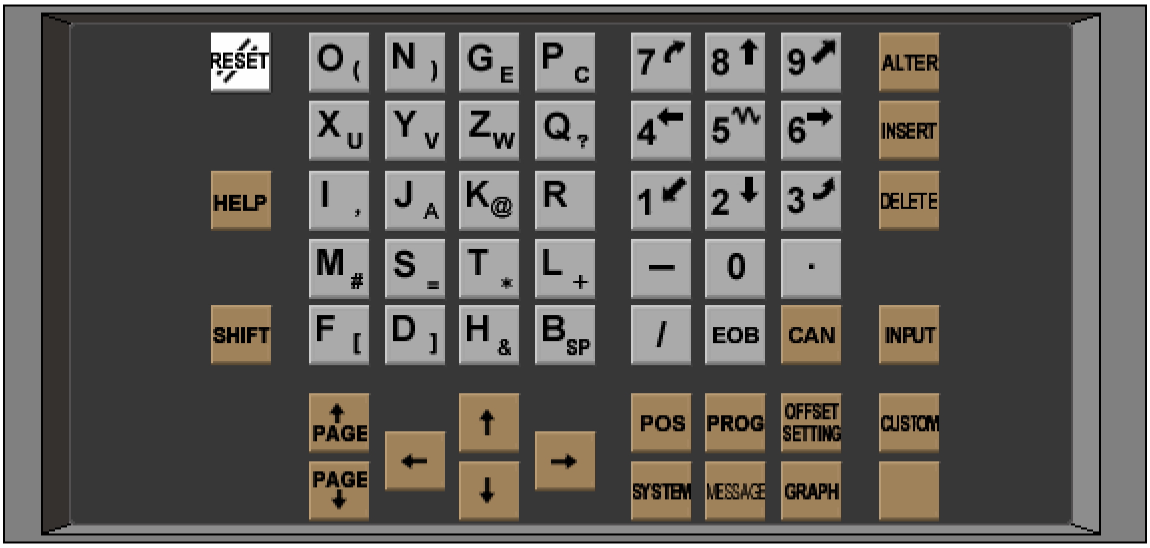
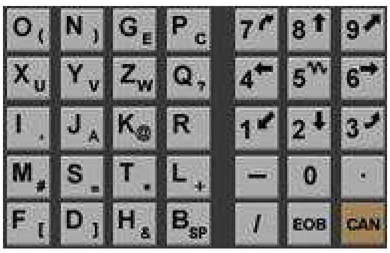


Fig. 5.2－1 FANUC 18i (milling machine) panel

5.2.1 BUTTON INTRODUCTION

Number/letter key



Number/letter key is used to input data to input region（as the follwing graph shows）.

Shift lowercase and capital in combination key by 

,such as： X — u ， Y — v .

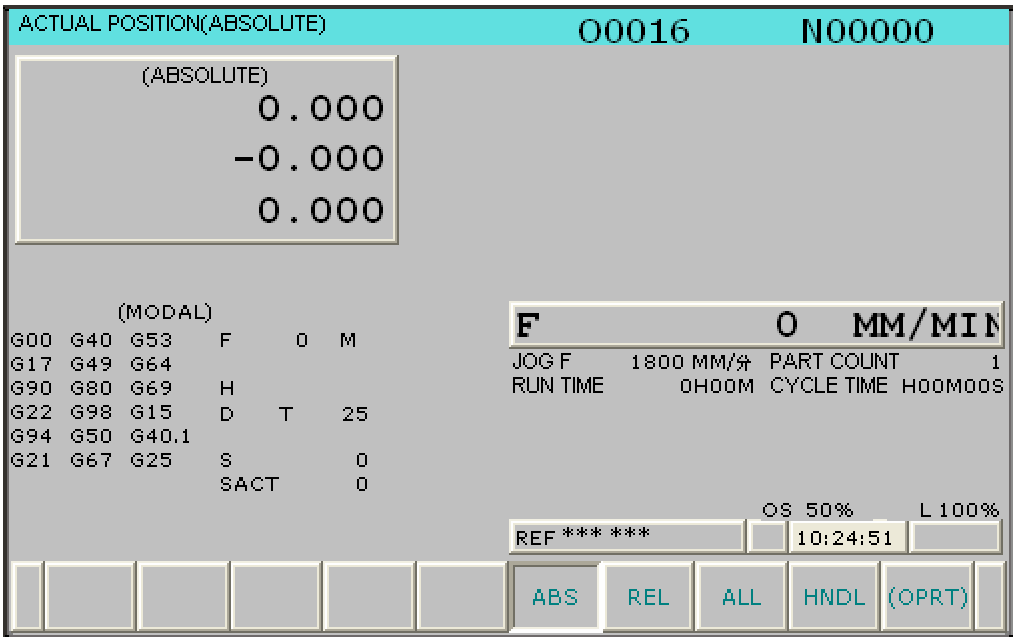


Fig. 5.2－2 FANUC 18i-M(milling machine) input of number and symble

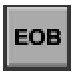
Edit key

Replace key. The data inputed replace the data curor pointing.

Delete key. Delete the data curor pointing； Or delete a NC program or all the programs.

Insert key. Insert the area behind curor with data which is in the input region.

Cancel key. Remove the data in input region.



Carriage return & line feed key.

Upper case key.

Program display and editing page.

Position display page. There are three mode for position display（absolute / relative / integration）.

Parameter input page.

System parameter page

Info page.Such“Alarm”.

Fig. parameter setup page.

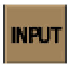
System help page.

Reset key.

Page turning button （ PAGE ）

Turn up 

Turn down

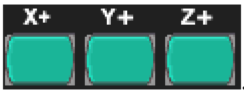


Input key. Input data in input region into input parameter page.

Return to reference point

(1) Put mode knob at 

.

(2) Choose axeses 

. Press the button to return to reference point.

5.2.2 MANUAL OPERATION OF MACHINE

THERE ARE THREE METHODES FOR MANUAL MOVING OF AXES：

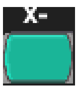
Method 1：Rapid moving 

. It is for long distance of work bench moving.

(1) Set mode at“JOG” 

.

(2) Click positive and negative direction button of axes and hold pressing to make axes move,release for stop. For example：After click 

， machine move to positive direction of X axes；After click 

， machine move to negative direction of X axes.

(3) Press 

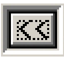
to make axeses move rapidly.

Method 2：Increment moving ， It is for microadjustment,such as preseting reference.

(1) Set mode at : Choose 1 、 10 、 100 、 1000 、 10000 for stepping amount.

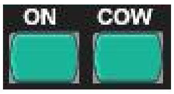
(2) Choose axeses. Each axes move one step every time press it.

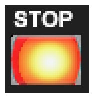
Method 3：“Hand Pulse”using 

, It is for microadjustment. It’s easy foroperator to control and observe the movement of machine. “Hand Pulse”is at the top-right corner of software interface  ， and it is emergent after click.

START、STOP SPINDLE

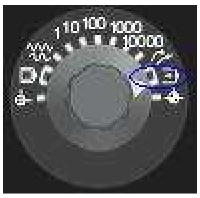
(1) Put mode knob at“JOG” .

(2) Press 

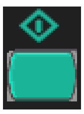
to make spingdle get positive and negative rotation.Press 

to stop spingdle.

START PROGRAM TO MACHINE COMPONENT

(1) Put mode knob at “AUTO” 

(2) Choose a program (please consult following process)

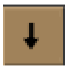
(3) Press program starting button 

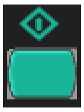
PROGRAM TEST RUNNING

Just run program, no cutting in test running.

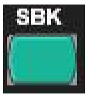
(1) Set mode at 

.

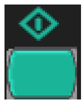
(2) Press  to call out program after choose a program such as O0001.

(3) Press program starting button 

SINGLE STEP RUN

(1) Put single step switch 

at “ON”.

(2) Only one command is executed, every time you press the button  when program is running.

CHOOSE A PROGRAM

There are two methods for choosing：

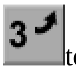
i.serch according to program numbering

(1) Choose“EDIT”mode

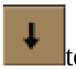
(2) Press 

,and then press 

to input letter“ O ”.

(3) Press 

to input number “ 3 ”. The searching number keyed in is：“ 03 ”

(4) Press CURSOR：  to start serching; After found, “ O3 ” is showed at the top right corner of screen， and“O3”NC program is on the screen.

ii. SET MODE AT AUTO .

(1) Press 

to input letter“O”

(2) Press 

to key in number“3”.The searching number keyed in is：“03”

(3) Press 

→ 

→

,and “ O3 ” displays on the screen .

(4) You can input block number“N30”， and then press 

or 

to search program block .

DELETE A PROGRAM

Set mode at “EDIT”

Press 

to input letter “O”

Press 

to input number“3”. Key in the numbering of program to be deleted “O3”.

Press 

to delete “O3”NC program.

DELETE ALL PROGRAMS

Set mode at “EDIT”

Press 

to input letter “O”

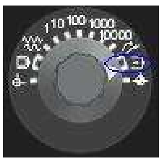
Input“-9999”

Press 

to delete all programs

SEARCH A SPECIFYED CODE

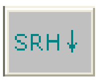
A specifyed code can be: a letteror a complete code. Such as：“N0010”，“M”，“F”，“G03”and so on. Searching is processed in current program. The operation step ：

Choose“AUTO” 
or“EDIT” 
mode

Press 

Choose a NC program

Input the needed letter or code, such as “M”，“F”，“G03”

Press 

in 

to start searching from current programs.

EDIT NC PROGRAM（DELETE、INSERT、REPLACE）

Choose“EDIT” mode

Choose PROG Input edited NC program name, such as“02”. Press INSERT to edit.

Move cursor：

Method 1：Press PAGE: PAGE ↑ or PAGE ↓ to turn page， and press CURSOR: ↓ or ↑ to move cursor.

Method 2：Use the method serching a specifyed code to move cursor.

Input data：Click number/letter key by mouse. CAN is used to delete data in input region.

Input number of automatically generating program block：Press OFFSET SETTING → SETING. Input “1”in sequence number of parameter page ， and the edited program will generate program block number automatically (such as：N10…N20…) .

DELETE、INSERT、REPLACE：

Press DELETE to delete code pointted by cursor

Press INSERT to insert the place behind code specifyed by cursor with data in input region.

Press ALTER to replace code specifyed by cursor with data in input region.

MANUAL INPUT OF NC PROGRAM WITH OPERATION PANEL

Put mode switch at“EDIT” 。

Press PROG， and then press DIR to login program page.

Press 3， and input“O3”program numbering (the numbering keyed in can’t be the same with existing numbering)

Press EOB → INSERT to start input.

Press EOB → INSERT to get a newline， and then start to input program sequentially.

INPUT A PROGRAM FROM COMPUTER

You can build a text to write NC code in computer by keyboard. But the suffix name of text file(*.txt) must be changed to *.nc or *.cnc.

Choose EDIT mode， and then press PROG to shift to program page.

New a program name “ Oxxxx ”, and then press INSERT to login programming page.

open NC file under the list of computer,and the program displays on current screen.

INPUT COMPONENT ORGIN PARAMETER

(1)Press OFFSET SETTING to login parameter setting page， and then press“Coordinate System”。

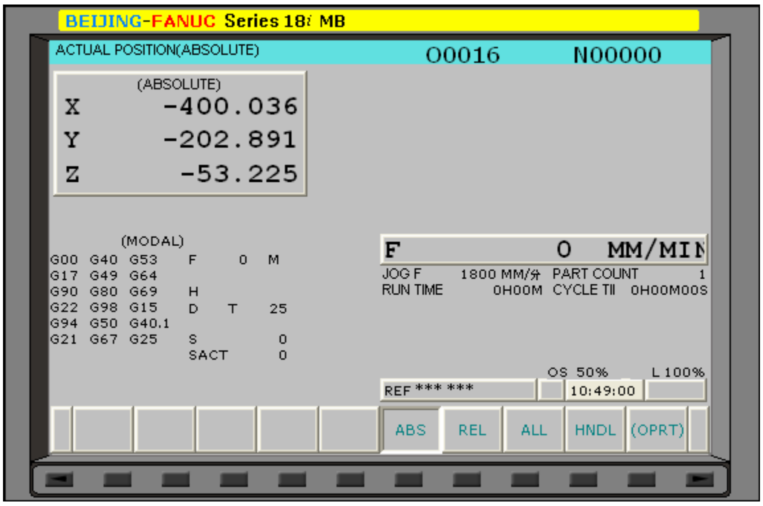


Fig. 5.2-3 FANUC 18 i-M(milling machine)

(2) Select coordinate by PAGE： PAGE↓ ↑PAGE or CURSOR: ↓ ↑.

Input address letter（X/Y/Z）and numberical value to input region. Please consult“Input Data”operation.

(3)Press INPUT to input the data in input region to specifyed place.

INPUT CUTTER COMPENSATION PARAMETER

(1)Press OFFSET SETTING to login parameter setup page,and press “Redress”“ 补正 ”.

(2)Select length compensation , radius compensation by PAGE： PAGE↓ and ↑PAGE.

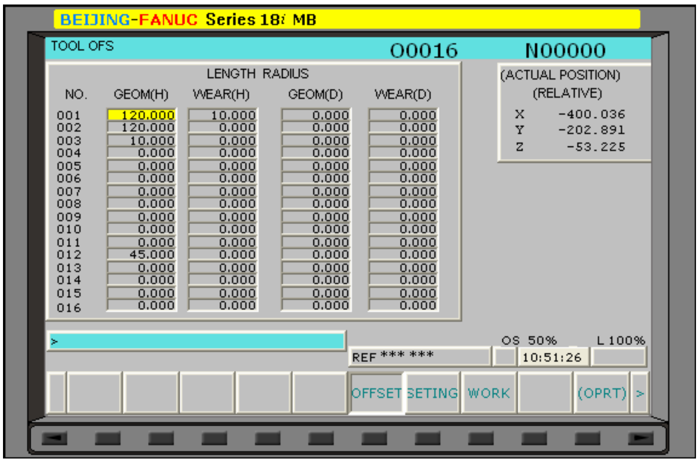


Fig. 5.2-4 FANUC 18 i-M(milling machine)

(3) Select compensation parameter numbering by CURSOR： ↓ and ↑.

(4) Input compensation value to length compensation H or radius compensation D.

(5) Press INPUT to input the inputted compensation value to specifyed place.

POSITION DISPLAY

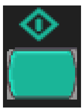
Press POS to shift to position display page.Shift by PAGE: PAGE ↓ and ↑ PAGE or by soft key.

MDI(MANUAL DATA INPUT)

(1)Press  to shift to “MDI”mode

(2)Press PROG ， and then press MDI → EOB to Input block number “N10”， such the input program： G0X50.

(3)Press INSERT ,and the program “N10G0X50” is inputted.

(4)Press  to start program.

MIRRORIMAGE FUNCTION

Press OFFSET SETTING → SETING → PAGE ↓ .MIRROR IMAGE X、MIRROR IMAGE Y、MIRROR IMAGE Z and mirrorimage functions of Xaxes、 Yaxes and Zaxes are one to one correspondence in parameter page.

For example: If you input“1” mirrorimage starts.

POSITION OF WORKPIECE COORDINATE SYSTEM（ABSOLUTE COORDINATE SYSTEM）

Absolute coordinate system：Display the position of machine in current coordinate system.

Relative coordinate system：Display the coordinate of machine with respect of the last position.

Synthetic display：Display positions of machine in following coordinate system at the same time.

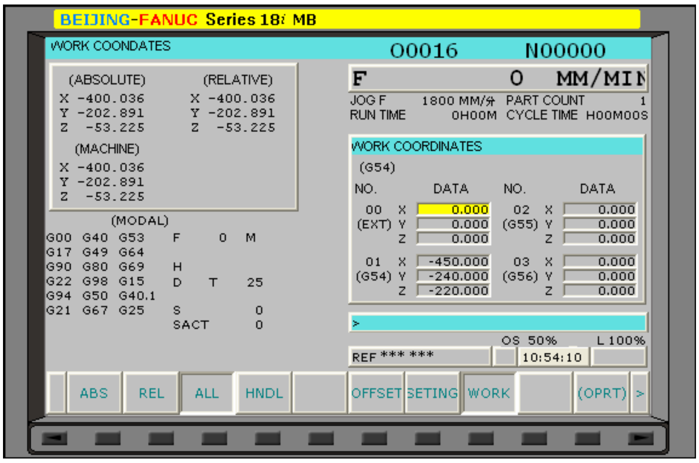


Fig. 5.2-5 FANUC 18 i-M(milling machine)

Position in workpiece coordinate system（ABSOLUTE）

Position in relative coordinate system（RELATIVE）

Position in machine coordinate system（MACHINE）

Residual distance in current moving order（DISTANCE TO GO）

5.3 AUXILIARY FUNCTION（M FUNCTION）

Auxiliary function includes many kinds of function used to sustain machine operation,such as start and stop of spingdle、program stop、open and close of coolant and so on.

code	instruction
M00	Program stop
M01	Choosing stop
M02	End of program (Reset)
M03	Spingdle corotation (CW)
M04	Spingdle reversal (CCW)
M05	Spingdle stop
M06	Too change
M08	Open coolant
M09	Close coolant
M19	Spingdle directive stop
M28	Return to orgin
M30	End of program (Reset) and recur

M48	Spingdle over loading cancel ineffective
M49	Spingdle over loading cancel effective
M60	APC loop start
M80	Rotary table corotation (CW)
M81	Rotary table reversal (CCW)
M94	Cancel mirrorimage
M95	Coordinate X mirrorimage
M96	Coordinate Y mirrorimage
M98	Subprogram call
M99	End of subprogram

Table 5.3－1 auxiliary function (M function) list

5.4 EXAMPLES

Example 1:

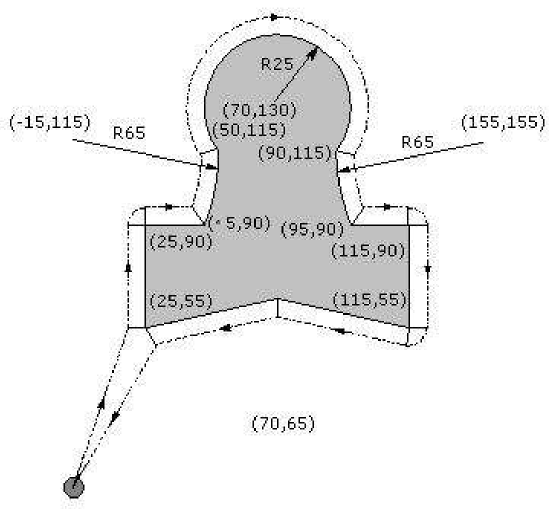


Fig. 5.4-1

T1 buttonhead milling tool Ø12。

N10 G40 G49 G80 G17 M06 T1 ； exchange Ø20 aiguille，cancel fixed cycle

N20 G54 G90 G00 X0 Y0 ； choose G54 as workpiece coordinate

N30 G43 H1 Z50 ； call length compensation

N40 Z2 M03 S800

N50 G1 Z-10 F200

N60 G41 X25.0 Y55.0 D1 ； left out tool compensation of tool radius

N70 Y90.0

N80 X45.0

N90 G03 X50.0 Y115.0 R65.0

N100 G02 X90.0 R-25.0

N110 G03 X95.0 Y90.0 R65.0

N120 G01 X115.0

N130 Y55.0

N140 X70.0 Y65.0

N150 X25.0 Y55.0

N160 G00 G40 X0 Y0 ; cancel left out tool compensation of tool radius

N170 Z100

N180 M5

N190 M30

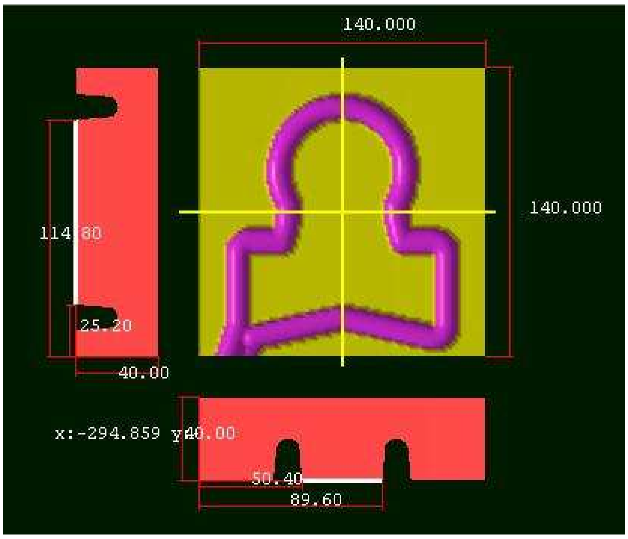


Fig. 5.4-2

Example 2:

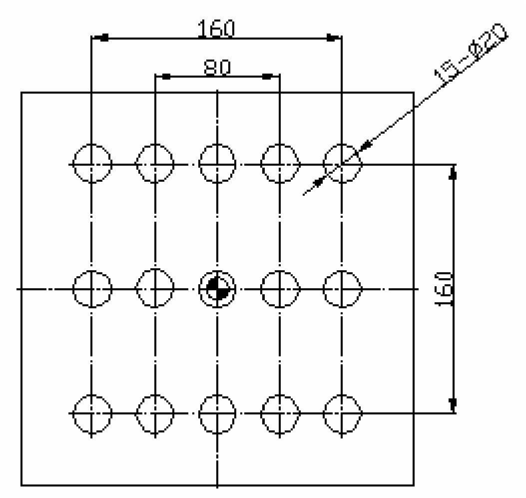


Fig. 5.5-3

N10 G40 G49 G80 G17 M06 T1 ; exchange Ø20 aiguille，cancel fixed cycle

N20 G54 G90 G0 X-80 Y-80 ; call G54 as workpiece coordinate and move to hole site

N30 G43 H1 Z50

N40 M3 S800

N50 M8

N60 G99 G83 Z-30 R1 Q2 F200 ; run drilling circle

N70 G91 X40 K4 ; repeat drilling

N80 Y80

N90 G91 X-40 K4

N100 Y80

N110 X40 K4

N120 G80 G90 G0 Z50 ; cancel fixed cycle

N130 M5 M9

N140 G91 G28 Z0 Y0

N150 M30

Example 3:

N010 G94 G54 G90 G0 X0 Y0

N020 G43 Z50 H1

N030 M3 S1000

N040 M8

/N050 M95 ; selection of coordinate X mirrorimage

/N060 M96 ; selection of coordinate Y mirrorimage

N070 G0 X-100 Y-100

N080 G81 Z-30 R1 F200

N090 G80 G0 Z50

N100 M94 ; cancel mirrorimage

N110 M5 M9

N120 G91 G28 Z0 Y0

N130 M30

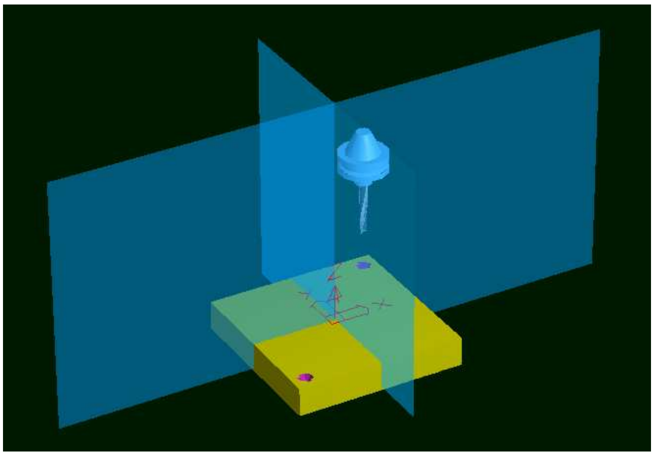


Fig. 5.5-4

CHAPTER 6 FANUC MILLING MACHINE PROGRAMMING

6.1 COORDINATE SYSTEM

The right-angle of descartes' set of coordinates is used in programming coordinate.

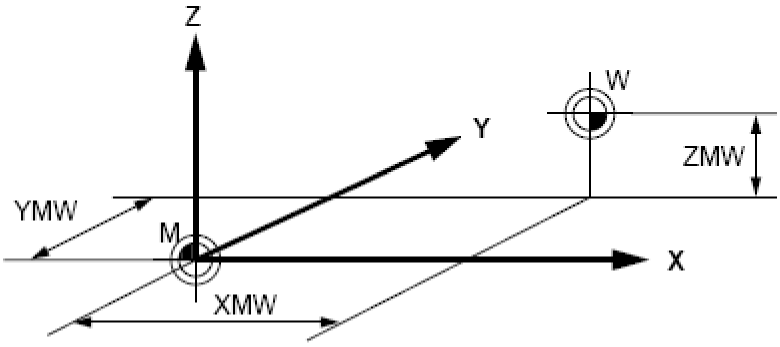


Fig.6.1-1

Remaining distance of moving

This function is not attached to the setting of coordinates , only the distance between the position of the aim and the machine tool currently when the order of movement has been sent out can be displayed in this function. Only when all of the axis' remaining distance is zero can this function be completed.

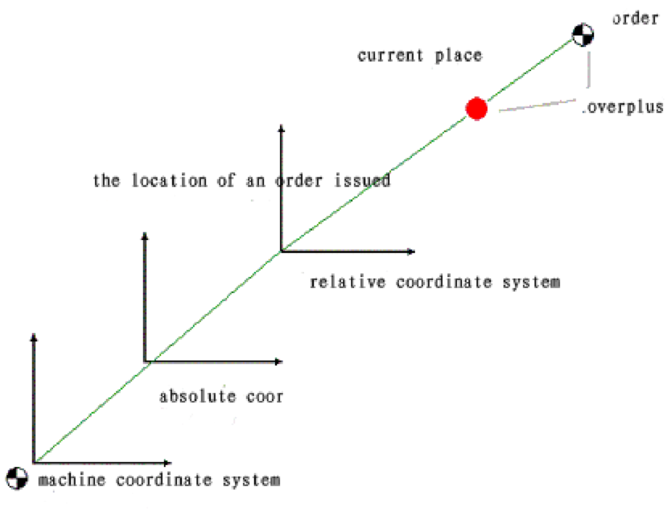


Fig.6.1-2

6.2 POLAR COORDINATE

Commands of polar coordinate

（G15,G16）

Coordinate values can be inputted in with polar coordinates (radius and angle).

The angle is positive when the first axis of the selected plane is anticlockwise, and negative when it is clockwise.

Radius and angle can use the command of absolute value or increment value(G90,G91).

Format of command

G □□ G〇 〇　 G16；Start the command of polar coordinates (polar coordinates mode)

G00　IP_；　Order of polar coordinate.

G15 ；Cancel the order of polar coordinate.

G16 ；Order of polar coordinate.

G□□　　Selection of the plane of the order of the polar coordinates (G17, G18 or G19).

G〇 〇　　G90 Specify the zero of workpiece coordinate as the origin point of the polar coordinates and measure the radius from this point.

G91 Specify the current position as the origin point of the polar coordinates, measure the radius from this point.

IP_　Specify the address and value of the selected axis of the polar coordinates.

The first axis: radius of the polar coordinates

The second axis: polar angle

Specify the zero point of workpiece coordinate as the origin point of the polar coordinate

Use the programming order of absolute value to specify the radius (distance between the zero and the point of programming). Specify the zero of the work's coordinates as the origin point of the polar coordinates. When use the part coordinates (G52), the origin point of the part coordinates become the center of the polar coordinates.

Specify the current position as the origin point of the polar coordinates

Specify the radius with the command of increment value programming (the distance between the current position and the point of programming). The current position is appointed as the origin point of the polar coordinates.

Specify radius and angle with command of absolute value

N1　G17　G90　G16 ;

Specify the order of the polar coordinates and select the plane of X-Y, specify the zero of the work's coordinates as the origin point of the polar coordinates.

N2 G81 X100.0　Y30.0　Z-20.0　R-5.0　F200.0

Specify the distance of 100 mm and the angle of 30 degrees.

N3　Y150.0;

Specify the distance of 100 mm and the angle of 150 degrees.

N4 Y270.0;

Specify the distance of 100 mm and the angle of 270 degrees.

N5 G15 G80;

Cancel the order of polar coordinate

Specify the angle with increment value

Specify the polar radius with absolute value

N1 G17 G90 G16 ;

Specify the order of polar coordinates and select the plane of X-Y, specify the zero of the work's coordinates as the origin point of the polar coordinates.

N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0

Specify the distance of 100mm and the angle of 30 degrees

N3 Y150.0;

Specify the distance of 100mm and the angle of 150 degrees

N4 Y270.0;

Specify the distance of 100mm and the angle of 270 degrees

N5 G15 G80;

Cancel the order of polar coordinate

Specify the radius in the mode of polar coordinate

In the mode of polar coordinates, do interpolation to the circle or cutting with the spiral thread (G20,G03) specify the radius with "R".

In the mode of polar coordinates, the axis is not looked as the part of the order of the polar coordinates.

The axis which are specified by the following order are not looked as the part of the order of the polar coordinates.

－Pause (G04)

-- Inputs of the programmable data (G10)

-- Set the local coordinate (G52)

-- The conversion of the work's coordinates (G92)

-- Select the coordinate of the machine tool (G53)

-- Store checking of journey (G22)

-- Circumrotation of the coordinates (G68)

-- Zooming of the proportion (G51)

Beveling and corner circle transition in any angle

In the mode of polar coordinates, beveling and corner circle transition in any angle cannot be realized.

6.2 COMMANDS OF G CODE

6.2.1 G code set and its meaning

The function of "mode code" can still exist after it has been used, but "current code" reacts only when it has received the order

Codes which define the movement always are "mode code", like line、circle and cycle codes, otherwise, like the code of return of the origin point is "current code"

Each code is belonged to the group of the similar codes. In the "mode code", current code would be replaced by the codes of the same group.

G code	**Function**
G00	Positioning(rapid moveing)
G01	Linear interpolation
G02	Circular interpolation/Helical interpolation CW
G03	Circular interpolation/Helical interpolation CCW
G04	Dwell, Exact stop
G15 G16	Polar coordinates command
G17	XpYp plane selection
G18	ZpXp plane selection
G19	YpZp plane selection
G28	Return to reference position
G30	2nd, 3rd and 4th reference position return
*G40	Cutter compensation cancel/Three dimensional compensation cancel
G41	Cutter compensation left/Three dimensional compensation
G42	Cutter compensation right
*G43	Tool length compensation + direction
*G44	Tool length compensation – direction
G49	Tool length compensation cancel
G53	Machine coordinate system selection
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G73	Peck drilling cycle
G74	Left-spiral cutting circle
G76	Fine boring cycle
*G80	Canned cycle cancel/external operation function cancel

G81	Drilling cycle, spot boring cycle or external operation function
G82	Drilling cycle or counter boring cycle
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
*G90	Absolute command
G91	Increment command
G92	Setting for work coordinate system or clamp at maximum spindle speed
*G98	Return to initial point in canned cycle
*G99	Return to R point in canned cycle
G50 G51	Scaling command
G68 G69	Coordinate rotation/Three dimensional coordinate conversion
	Support the programming of macroprogram

Table. 6.2-1 G code set and its meaning

(Codes with the sign of "*"means they can be initialized when boot-strap)

6.2.2 Explanation of G code

G00

➢ Rapid positioning(G00)

1. Format

G00 X_ Y_ Z_

This order take the cutter move from the current place to the appointed place(in the condition of absolute coordinates), or to the position where have been given the distance(in the condition of increment coordinates)

2. The positioning of the mode of cutting along un-straight line

Our definition is: we use the unattached speediness movement ratio to determine the position of each axis. The path of the cutter is not a line, according to the sequence of arriving, in following, the axis of the machine would stop at the place where the order has appointed.

3. positioning with straight line

The path of the cutter is like line cutting, and can orient to the aimed position in the shortest time (not over each of the axis's speediness movement ratio)

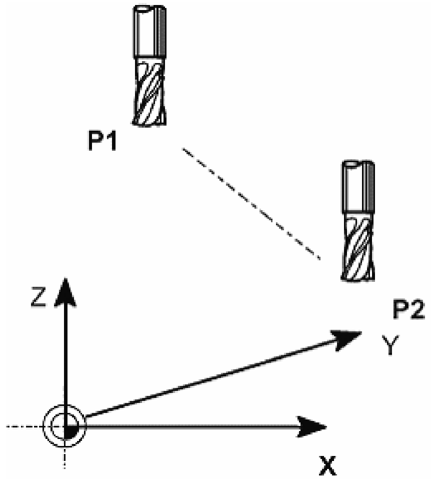


Fig.6.2-1

G01

➢ Feeding of linear cut(G01)

1. Format

G01 X_ Y_ Z_F_

This order will make the cutter move in line, the rate as the code F gives, from the current to the aimed position. The rate of the code F is the recombination rate of the appointed axis in the program.

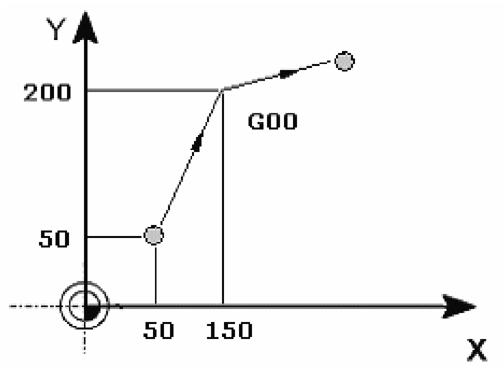


Fig.6.2-2

➢ Arc cutting(G02/G03)

Format

When the arc is on the plane of XY
G17 G02 (G03) G90 (G91) X_ Y_ F_; or G17 G02 (G03) G90 (G91) I_ J_ F_;
Or G17 G02 (G03) G90 (G91) R_ F_;
When the arc is on the plane of XZ
G18 G02 (G03) G90 (G91) X_ Z_ F_; or G18 G02 (G03) G90 (G91) I_ K_ F_;
Or G18 G02 (G03) G90 (G91) R_ F_;
When the arc is on the plane of YZ
G19 G02 (G03) G90 (G91) Y_ Z_ F_; or G19 G02 (G03) G90 (G91) J_ K_ F_;
Or G19 G02 (G03) G90 (G91) R_ F_;

The plane of the arc is specified with the code of G17, G18, G19.But, if we have defined these orders in the preparatory program, we can also need not these orders. The turning direction of the arc like what shows in the following chart, specified with the order of G02 and G03.After the turning direction of the arc is specified, we specify the coordinates of the end point of the cutting. G90 is appointed in the condition of the absolute value while G91 is in the condition of the increment value. And if G90/G91 has been given in the preparatory program, it can be overpass. The end point of the arc is appointed by two axis' coordinates value which are contained in the plane of the order infliction.(For example , in the plane of XY, G17 use the value of X and Y) . The end point's value can be setup like G00 and G01. The position of the center of the arc or its radius should be set only after the end point of the arc has been set. The center of the arc is set in the form of the distance of the relative distance to the start point of the arc, and denote I 、J and K in corresponding to the axis of X、Y and Z. The result of the center value of the arc subtract the value of the start point of the arc in corresponding to the axis of I、J and K.

2. Example

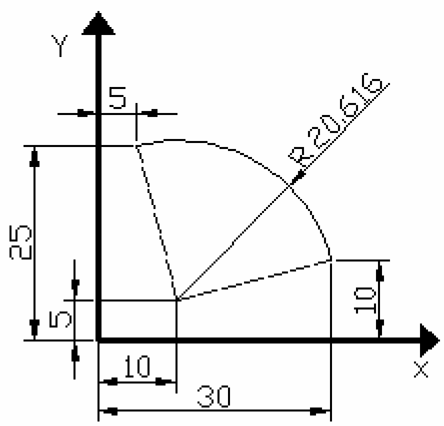


Fig.6.2-3

Jumping-off point of X value -----------30

The arc's center point X value-----------10

So, "I" is -20

Jumping-off point of Y value -----------10

The arc's center point Y value-----------5

So, "J" is -5

This arc program is like the followings

G17 G03 G90 X5 Y25 I-20 J-5.   or

G17 G03 G91 X-25 Y15 I-20 J-5

or the radius of the arc program :

G17 G03 G90 X5 Y25 R20.616   or

G17 G03 G91 X-25 Y15 R20.616

Attention:1) when set the center of the arc like "I" "J" "K", we should set it as the increment value from the jumping-off point to the center of the arc.

Attention:2) We can bypass the "I0" "J0" "K0" in the program.

G15/G16 Commands of polar coordinate

The value can be inputted in the form of the polar coordinates (radius and angle)

The angle is position when the first axis of the selected plane is anticlockwise, and negative when it is clockwise.

Radius and angle can use the command of absolute value or increment value (G90,G91)

Use the command of absolute value to specify the angle and radius

N1 G17 G90 G16

Appoint the polar coordinates command and choose the plane of XY, set the zero of the work coordinates as the origin point of the polar coordinates.

N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0

Specify the distance of 100mm and the angle of 30 degrees

N3 Y150.0

Specify the distance of 100mm and the angle of 150 degrees

N4 Y270.0

Specify the distance of 100mm and the angle of 270 degrees

N5 G15 G80

Cancel the order of the polar coordinates

Specify the angle with increment value and specify the radius with absolute value

N1 G17 G90 G16

Appoint the polar coordinates command and choose the plane of XY, set the zero of the work coordinates as the origin point of the polar coordinates.

N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0

Specify the distance of 100mm and the angle of 30 degrees

N3 G91 Y120.0

Specify the distance of 100mm and 120 degrees of the angle increment .

N4 Y120.0

Specify the distance of 100mm and 120 degrees of the angle increment .

Cancel the order of the polar coordinates

G17/G18/G19 Selection of planes

Use the parameters to appoint the circumrotate axis is X Y or Z ,or the axis that parallel to these axis , specify the G code to select the plane , to this plane , the circumrotate axis is the appointed line axis. For example, when the circumrotate axis is the axis that parallel to the X axis ,G17 should specify the plane of X and –Y, and only circumrotate axis one can be set.

When we omit the address of the axis of X Y and Z , we consider the third axis's address is omitted

In the program, which are not appointed by the order of G17 G18 G19,the plane stay the same.

G28/G30

➢Return to orgin automatically(G28/G30)

1. Foemat

The first origin point return：

G28 G90 (G91) X_Y_Z_；

The second, third and fourth origin point return：

G30 G90 (G91) P2 (P3, P4) X_Y_Z_；

#P2, P3, P4: select the second, third and the fourth origin point return(if omitted , system will return in the form of the second origin point return)

The place set by X Y and Z is entitled as the middle point. The machine tool move to this point first, then return to the origin point , and we omit the middle axis with no movement , only in the program we appoint the axis of the middle point can it execute the order of the origin point return.

In the order of the origin point return , each axis is executed unaided, like the order of G00, always the path of the cutter is not a line , so we should set middle point to each axis , in order to avoid the collision between the machine tool and the work during the origin point return.

2. Example

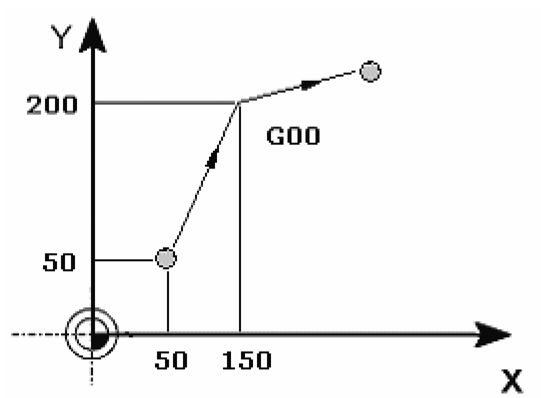


Fig.6.2-4

G28 (G30) G90 X150. Y200.； or

G28 (G30) G91 X100. Y150.；

Attention: In the example, move to the middle point is just like the fastness movement order of the following.

G00 G90 X150. Y200.； or

G00 G91 X100. Y150.；

If the middle point is the same with the current cutter (For example, the order is –G28 G91 X0Y0 Z0;) the machine tool would return to the origin point from the current position . If the program is executed in the single block, the machine tool would stop at the middle point(the current position)

G40/G41/G42

➢Offset function of cutter's diameter(G40/G41/G42)

Format

G41 X_ Y_D_；
G42 X_ Y_D_；

When we process the workpiece ("A"), like showed in the following chart, the path of the cutter("B") is the basic path, the distance to the work should not be less than half of the cutter's diameter. Here, the path "B" is called as the work's compensatory path through R. so , the offset of the cutter's diameter can give the path "A" give by the program automatic and the cutter's offset value appointed separately, can work out the path "B" which has been compensated . In other words, the customers can make the program by the figure of the work, and we can not consider the cutter's diameter. So, when do cutting, we set the diameter of the cutter as the offset value of the cutter, they can get the accurate result, all because the system itself calculate the exact compensated path.

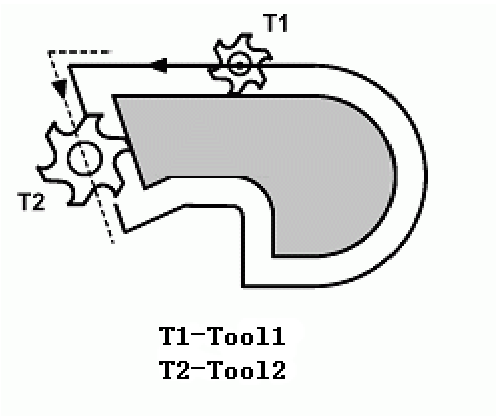


Fig.6.2-5

In programming, the customer can only insert the direction of the offset vector and the address of the offset. So the customer insert the number of the offset, and the machine will calculate the diameter of the cutter , then get the radius.

2. Offset function

Codes	Function
G40	Cancel the offset of
G41	Offset at the left of moving direction of the cutter
G42	Offset at the right of moving direction of the cutter

Table.6.2-1

G43/G44/G49

➢ Length offset of cutter (G43/G44/G49)

1. Format

```
G43 Z_ H_;
G44 Z_ H_;
G49 Z_;
```

2. Offset function

First, we set a milling cutter as the basic cutter, and use the Z axis of the work's coordinates, and oriented it on the surface of the workpiece, and its position is z0. We should remember that if the cutter's radius is not long enough, then in the process the cutter may not attach the work, though the machine tool can move to z0. But, if the cutter is longer than the basic cutter, the cutter would hit the work and do harm to the machine tool. In order to not cause this happen, we can input the increment value of each cutter and the basic cutter to the machine, and let the machine carry out the function of the cutter's offset in the program.

Codes	Function
G43	Make the value of the cutter's offset add to the value of Z coordinates of the program
G44	Make the value of the cutter's offset subtract the value of Z coordinates of the program
G49	Cancel the offset of the length of the cutter

Table.6.2-2

In the process of setting the length of the offset, we use the sign of "+" "-", if we change the sign, the order of G43 and G44 will do in the opposite direction. So, this order has many different kinds of expressions.

For example:

First, follow these steps to measure the length of the cutter

1 Put the workpiece on the table

2 Change the cutter to be measured

3 Adjust the axis line of the cutter, make it close to the work, and make the front of the cutter to

the surface of the work

4 Here, the value of the Z axis in the comparative coordinates is inputted to the list of the offset as the offset value of the cutter

If we do these above, if the cutter is shorter than the basic cutter, the offset is negative, and longer than the basic cutter ,it is positive.

So, in the program only the order of G43 can allow us to make the offset of the cutter.

3. Example

G00 ZO；

G00 G43 Z0 H01；

G00 G43 Z0 H03； or

G00 G44 Z0 H02； or

G00 G44 Z0 H02；

Once the command G43, G44 or G49 is sent, their function will keep in the program , for they are belong to the “mode code ”. so , if the order of G43 or G44 in the program we use after the change of the cutter , then G49 would be in use after this work , and in use of changing the cutter.

Attention 1) when we use G43(G44) H or G49 to omit the movement of the Z axis , the act of the cutter’s offset would be like the order of G00 G90 Z0. In other words, the customer should be care about this all the time.

Attention 2) Customer can use G49 to cancel the offset of the length of the cutter, we can also use the offset number H0 (G43/G44 H0) to get the same result.

Attention 3) If we amend the number of the offset in the period of the compensation of the cutter, the former value would be replaced by the new value.

G53

➢ Select the coordinate of machine (G53)

1. Format

(G90) G53 X_ Y_ Z_；

2. Function

The order would move the cutter to the position of the coordinates of the machine tool, as G53 is the “common” code, and it would function only in the program where has G53. And it can be used in the order of G90, but not in G91. In order to move the cutter to the fixed position of the machine, like the position change the cutter, the program would be use in the coordinates has G53

Attention (1) Offset of the cutter’s diameter length and position should be cancelled in the order of it use the order of G53.

Attention (2) When we use the order of G53, we should make the machine to the origin point with G28. As a result of the coordinates of the machine should be setup before the order of G53.

G54～G59

➢Selection of workpiece coordinate(G54～G59)

1. Format

G54 X_ Y_ Z_；

2. Function

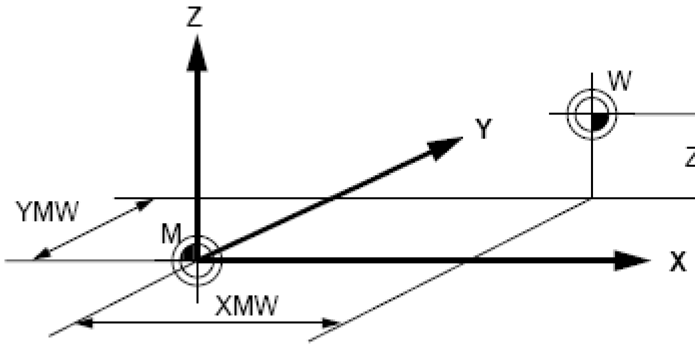


Fig.6.2-6

We can use G54-G59 to set at most six workpiece coordinates (one to six).

After we switch on the power and finish the return to the origin point , the system would select the work's coordinates1(G54), they are both mode order , we can keep its function after we execute the order of one coordinate till the order of other coordinates are emitted.

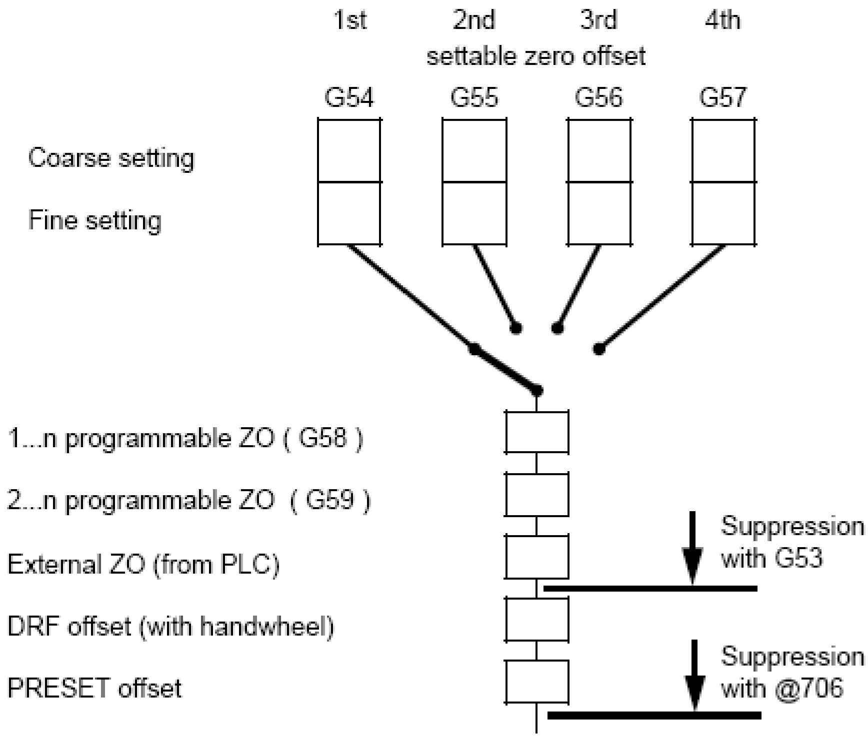


Fig.6.2-7

G73

➢Rapid depth drill circle(G73)

1. Format

G73 X__Y__Z__R__Q__ F__K__

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

Q_: Depth of cutting per time（no symbol, increment）

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

Feed to the bottom of the hole and withdraw cutter quickly

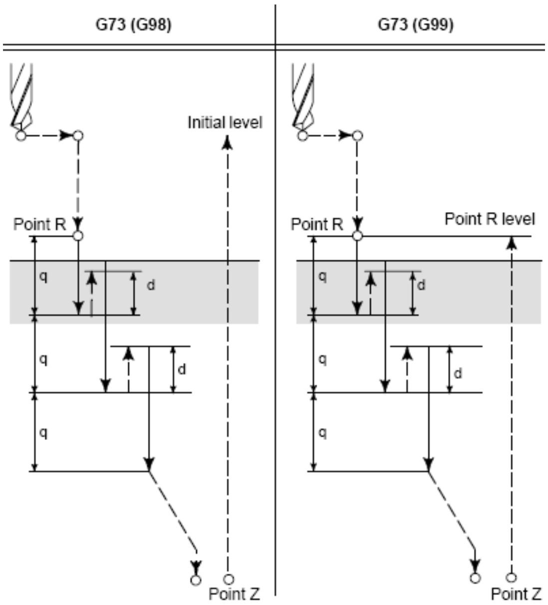


Fig.6.2-8

3. Example

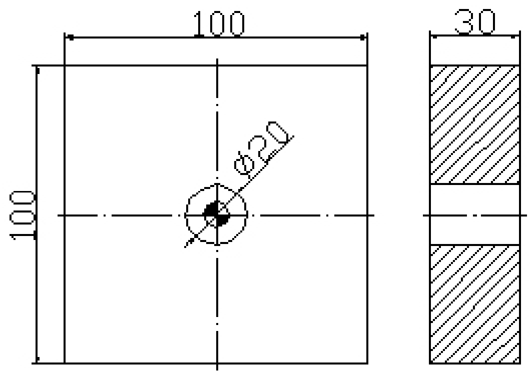


Fig.6.2-9

N005 G80 G90 G0 X0 Y0 M06 T1 ； change aiguille of Ø20，

N010 G55 ； call workpiece coordinate of G55

N020 M03 S1000

N030 G43 H1 Z50

N040 G98 G73 Z-30 R1 Q2 F200 ； gun drilling，start feeding 1MM beyond surfaceofworkpiece

Cut 2MM every time

N050 G80 G0 Z50 ; cancle canned cycle

N060 M05

N070 M30

G74

➢ Left spiral tap cycle (G74)

This cycle carry out the action of left circle tap, when the cutter comes to the bottom of the hole, the axis spire clockwise.

1. Format

G74 X__Y__Z__R__ P__F__K__

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

P_: Depth of cutting per time（no symbol, increment）

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

The spindle do tapping anticlockwise, when get to the bottom of the hole , in order to return, the spindle rotates clockwise, and this action has produced a reversely whorl.

During this period, ratio of feeding is omitted. Whwn the supply is paused, the machine will not stop till the return action is finished. Before we use G74, we can use the assistant to make the principal axis spire anticlockwise.

When we have appointed the repeated times (K), this function would only play in the first hole, and the others would be not.

If we specify the offset of the cutter’s length in the fixed cycle(G43,G44 or G49), the offset would be added when oriented to the R point.

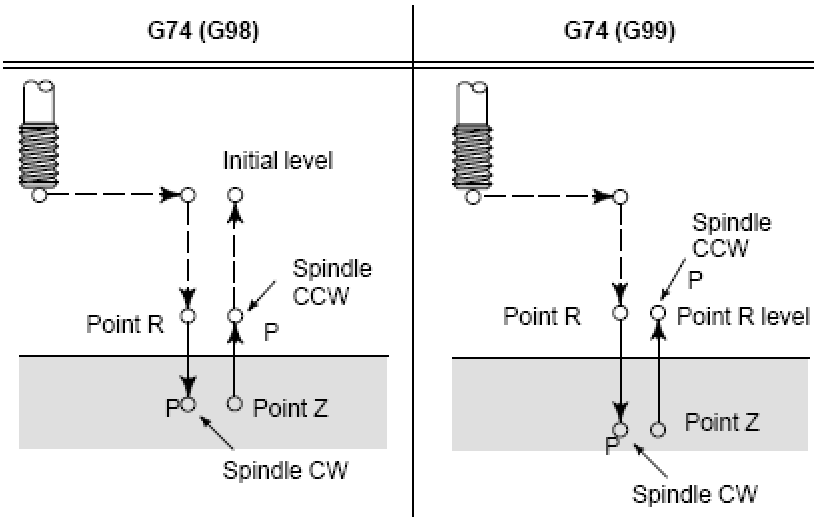


Fig.6.2-10

G76

➢ Precision boring circle (G76)

1. Format

G76 X__Y__Z__R__Q__P__F__K__

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

Q_: Offset of hole bottom

P_: Pause time (unit：ms)

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

Feed to the bottom of the hole and spindle pauses and return back quickly.

G80

➢Cancel the course of canned cycle(G80)

1. Format

G80；

2. Function

This order is used to delete the fixed cycle, and the machine tool return back to the usual state of manipulation. The needed data the hole , including R point , Z point and so on , are all omitted, but the ratio of the movement will still has its function.

Attention: In order to cancel the fixed cycle, except G80, the customer can also use the 01 group of G (G00,G01,G02,G03 and so on) , and any of them could play its role.

G81

➢Fixed point boring circle (G81)

1. Format

G81 X_Y_Z_R_F_K_;

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate)

R_: Starting point or raising point per time (absolute coordinate)

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

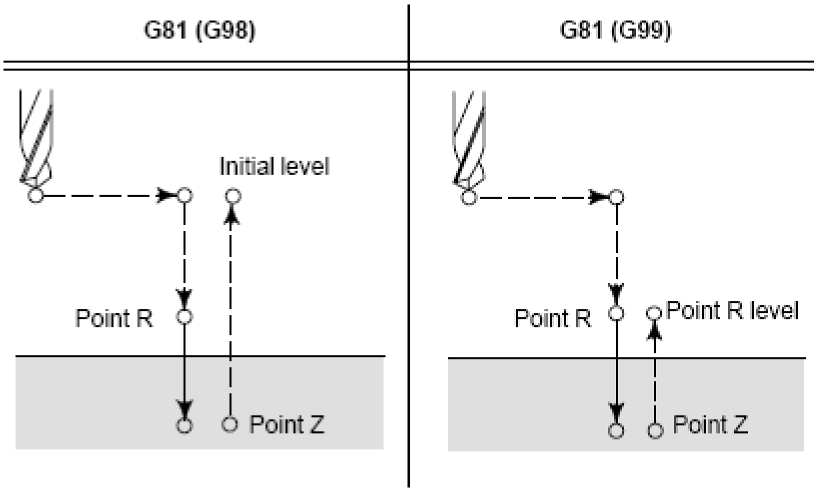


Fig.6.2-11

G81 command can be used in the machining of many holes.

3. Example

Fig.5.2-9

N005 G80 G90 G0 X0 Y0 M06 T1 ; change the aiguille of Ø20

N010 G55 ; call workpiece coordinate of G55

N020 M03 S1000

N030 G43 H1 Z50

N040 G98 G81 Z-30 R1 F200 ; drilling circle

N050 G80 G0 Z50 ; cancel fixed circle

N060 M05

N070 M30

G82

➢ Drilling cycle (G82)

1. Format

G82 X_Y_Z_R_P_F_K_;

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

P_: Pause time (unit： ms)

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

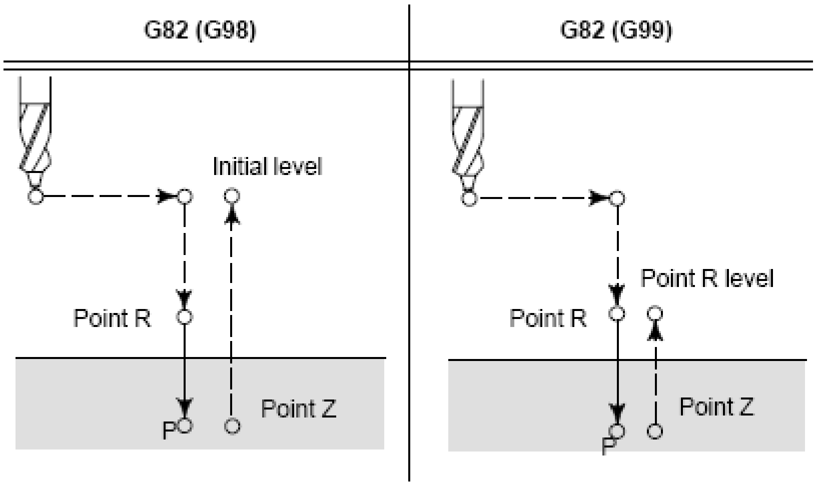


Fig.6.2-12

G82 drilling cycle, anti-boring cycle

3. Example

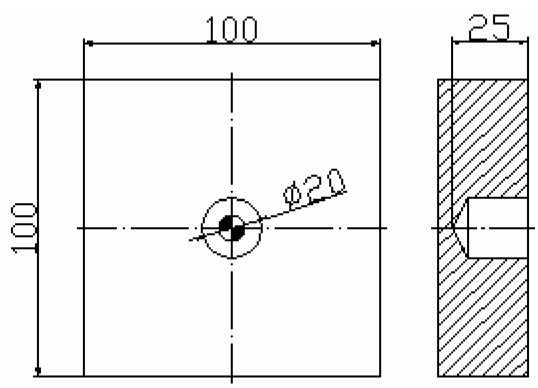


Fig.6.2-13

N005 G80 G90 G0 X0 Y0 M06 T1 ； change the aiguille of Ø20

N010 G55 ； call workpiece coordinate of G55

N020 M03 S1000

N030 G43 H1 Z50

N040 G98 G82 Z-30 R1 P2000 F200 ； drilling circle

N050 G80 G0 Z50 ； cancel fixed circle

N060 M05

N070 M30

G83

➢ Gun drilling cycle (G83)

1. Format

G83 X_Y_Z_R_Q_F_K_;

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

Q_: Offset of hole bottom

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

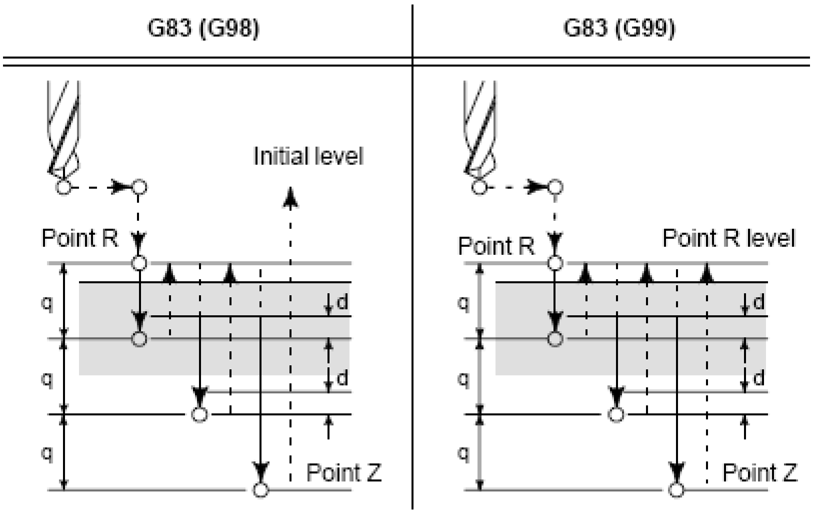


Fig.6.2-14

G83 Middle feeding and withdraw quickly

3. Example

As Fig.5.2-9 shows

N005 G80 G90 G0 X0 Y0 M06 T1 ; change the aiguille of Ø20

N010 G55 ; call workpiece coordinate of G55

N020 M03 S1000

N030 G43 H1 Z50

N040 G98 G83 Z-30 R1 Q2 F200 ; depth drilling circle, 2MM per time

N050 G80 G0 Z50 ; cancel fixed circle

N060 M05

N070 M30

G84

➢Tapping cycle (G84)

Do tapping this cycle, when get to the bottom of the hole , the spindle rotates reversely.

1. Format

G84 X_Y_Z_R_P_F_K_;

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

P_: Pause time (unit：ms)

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

The spindle do tapping anticlockwise, when get to the bottom of the hole , in order to return back, the spindle rotates reversely, and this process cause the whorl.

During the period of tapping the ratio of feeding is omitted, the feeding pause but the machine do not stop till the action of return is finished.

Before we appoint G84, we can use secondary function to make the spindle rotate.

When G84 and code M are specified in one block, execute the first action of orientation, meanwhile, the process of the R point we can add the increment.

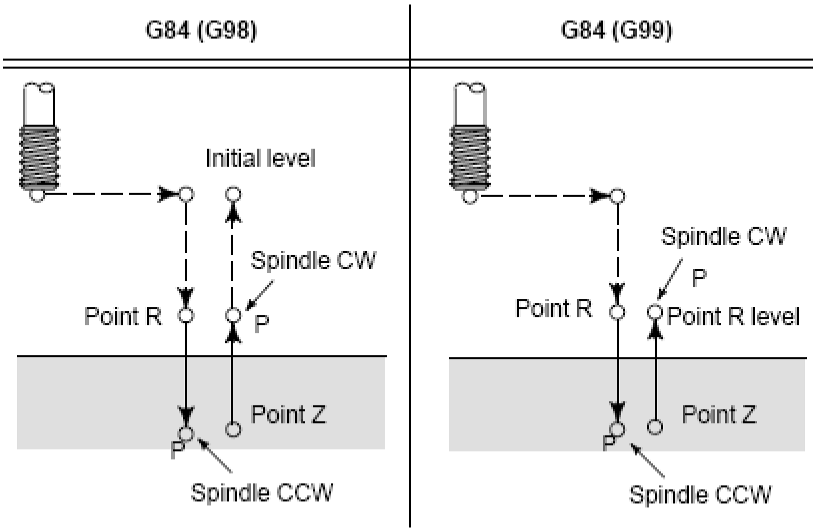


Fig.6.2-15

G84 The spindle feeding to the bottom of the hole rotates reversely and withdraw quickly.

3. Example

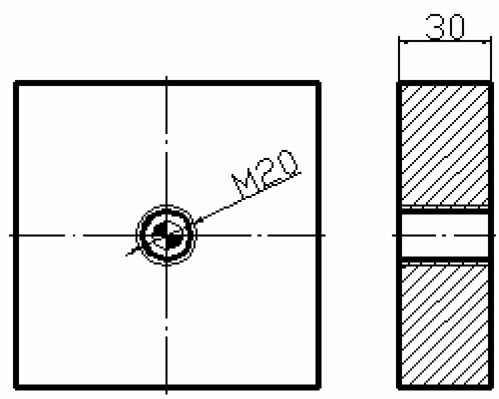


Fig.6.2-16

N005 G80 G90 G0 X0 Y0 M06 T1 ; change the aiguille of Ø20

N010 G55 ; call workpiece coordinate of G55

N020 M03 S800

N030 G43 H1 Z50 ; call length compensation

N040 G84 Z-30 R5 P2000 F2 ; tapping cycle

N050 G80 Z50 ; cancel fixed circle

N060 M05

N070 M30

G85

➢Boring cycle (G85)

1. Format

G85 X_Y_Z_R_F_K_;

X_ Y_: Depth of the bottom of hole(absolute coordinate)

R_: Starting point or raising point per time（absolute coordinate）

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

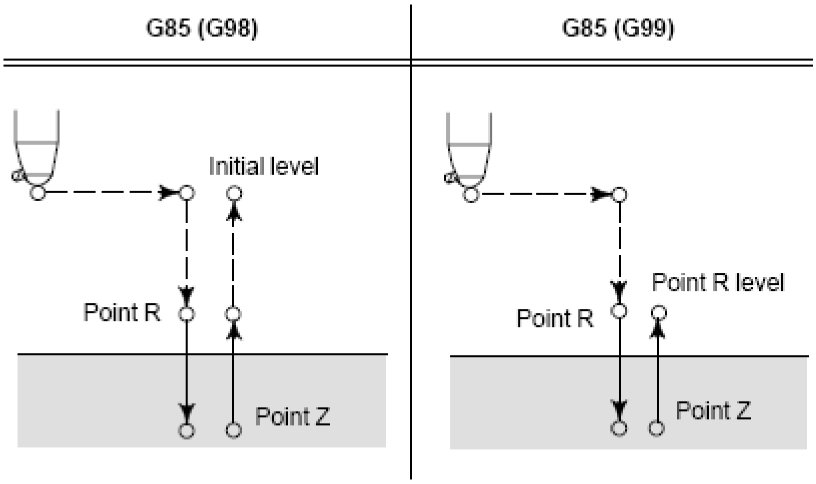


Fig.6.2-17

G85 Middle feeding to the bottom of hole and withdraw quickly

3. Example

As Fig.5.2-9 shows

N005 G80 G90 G0 X0 Y0 M06 T1 ; change the boring cutter of Ø20

N010 G55 ; call workpiece coordinate of G55

N020 M03 S1000

N030 G43 H1 Z50 ; call length compensation

N040 G85 Z-30 R1 F200 ; boring cycle

N050 G80 G0 Z50 ; cancel fixed circle

N060 M05

N070 M30

G86

➢ Boring cycle(G86)

1. Format

G86 X_Y_Z_R_F_K_；

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Functin

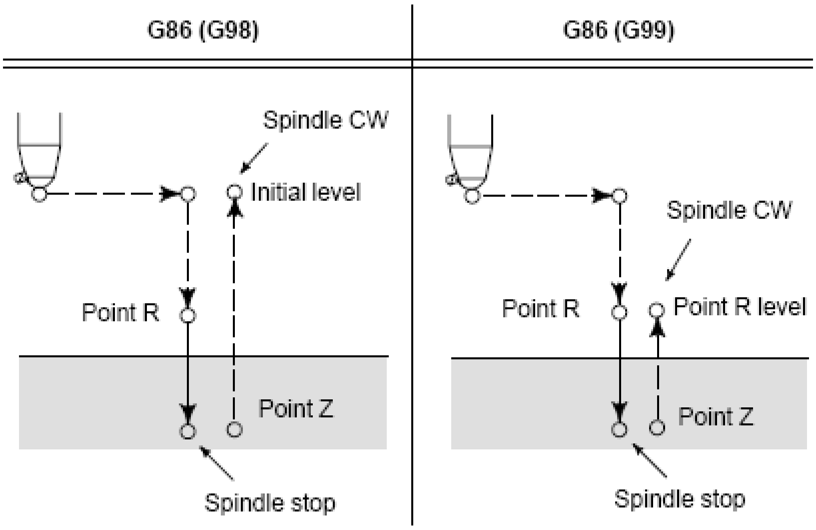


Fig.6.2-18

G86 The spindle feeding to the bottom of the hole rotates reversely and withdraw quickly.

3. Example

As Fig.5.2-9 showes

```
N005 G80 G90 G0 X0 Y0 M06 T1      ; change the boring cutter of Ø20
N010 G55                          ; call workpiece coordinate of G55
N020 M03 S1000
N030 G43 H1 Z50                   ; call length compensation
N040 G86 Z-30 R1 F200             ; boring cycle
N050 G80 G0 Z50                   ; cancel fixed circle
N060 M05
N070 M30
```

G87

➢Anti- boring cycle (G87)

1. Format

G87 X_Y_Z_R_Q_P_F_K_;

X_ Y: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

Q_: Offset of hole bottom

P_: Pause time (unit：ms)

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

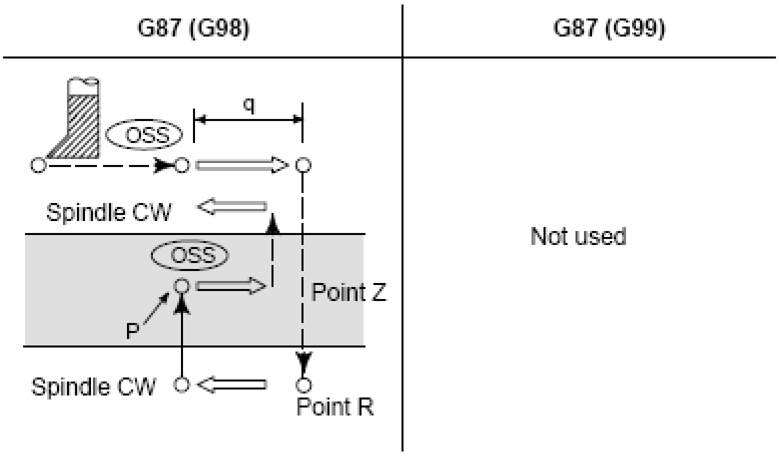


Fig.6.2-19

G87 Feed to bottom of hole and spindle rotates positive and withdraw quickly

3. Example

As Fig.5.2-9 shows

N005 G80 G90 G0 X0 Y0 M06 T1 ; change the boring cutter of Ø20

N010 G55 ; call workpiece coordinate of G55

N020 M03 S1000

N030 G43 H1 Z50 ; call length compensation

N040 G87 Z-30 R1 Q2 P2000 F200 ; anti-boring cycle

N050 G80 G0 Z50 ; cancel fixed circle

N060 M05

N070 M30

G88

➢Fixed point dring cycle (G88)

1. Format

G88 X_Y_Z_R_P_F_K_;

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

P_: Pause time (unit： ms)

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

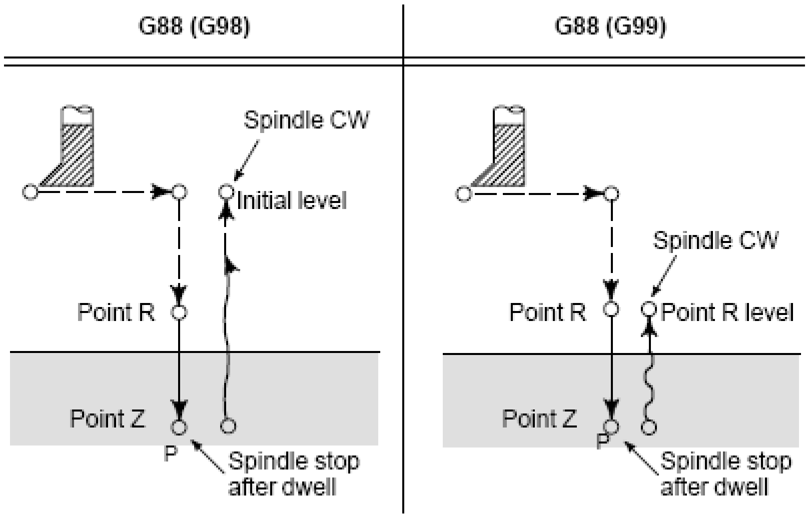


Fig.6.2-20

G88 Pause when feeding to bottom and spindle stops then withdraw quickly

G89

➢Boring cycle(G89)

1. Format

G89 X_Y_Z_R_P_F_K_；

X_ Y_: Data of hole site

Z_: Depth of the bottom of hole(absolute coordinate）

R_: Starting point or raising point per time（absolute coordinate）

P_: Pause time (unit： ms)

F_: Feeding rate of cutting

K_: Number of replication (if necessary)

2. Function

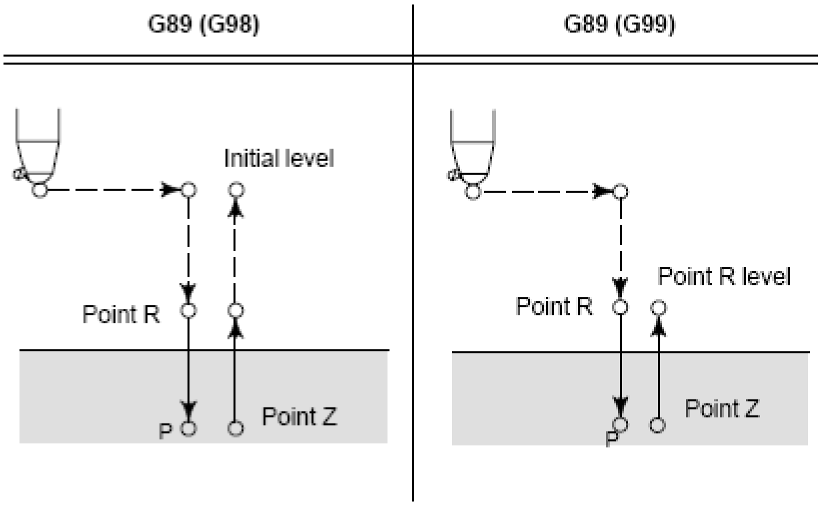


Fig.6.2-21

G89 Pause when feeding to bottom and spindle stops then withdraw quickly

3. Example

As Fig.5.2-9 shows

N005 G80 G90 G0 X0 Y0 M06 T1 ; change the boring cutter of Ø20

N010 G55 ; call workpiece coordinate of G55

N020 M03 S1000

N030 G43 H1 Z50 ; call length compensation

N040 G89 Z-30 R1 P2000 F200 ; boring cycle

N050 G80 G0 Z50 ; cancel fixed circle

N060 M05

N070 M30

G90/G91

➢ Absolute command / increment command (G90/G91)

This command set the value of X Y and Z in the program is absolute value or increment value, not consider their order before.The block containing G90 and the following block are assigned by absolute command; While block with G91 and its following block are assigned by increment command.

G98/G99 plane of return point

When the tool gets to the bottom of the hole, it can return back to the plane of R point and the initialized plane initialized by G98/G99. Generally speaking, G99 is used in the first drilling plane while G98 is used in the last drilling, even though we use G99 to drill, the plane of the initialized position would keep the same.

CHAPTER 7 FANUC PROGRAMMING OF LATHE

7.1COORDINATE SYSTEM

Program orgin

The orgins of program and coordinate system must be setted.Generaly, program orgin is setted on as the point convenient for programming.

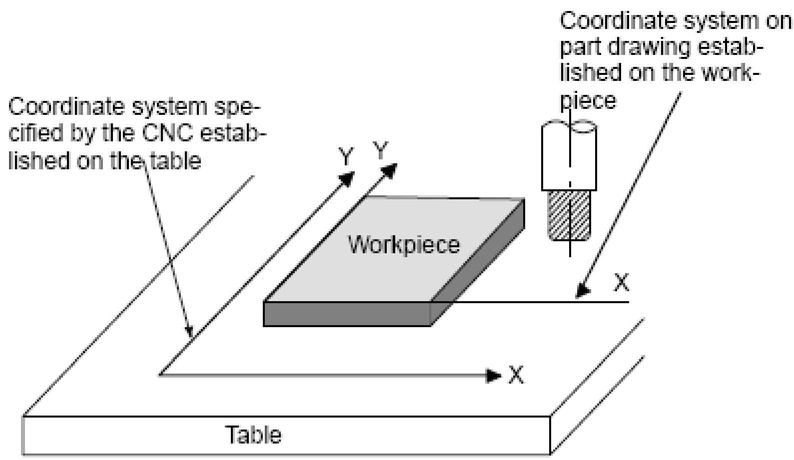


Fig.7.1-1

Setting orgin of coordinate system

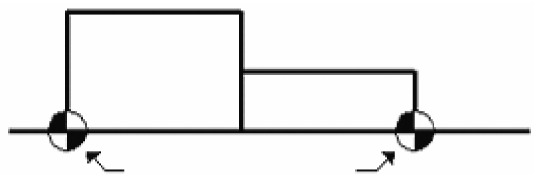


Fig.7.1-2 Example for setting of program orgin

Remaining ship distance

This function dose not belong to coordinate system, and it only displays the distance between aim position and current position after movement command is sent off.The command is finished only till remaining distance of each axes is zero.

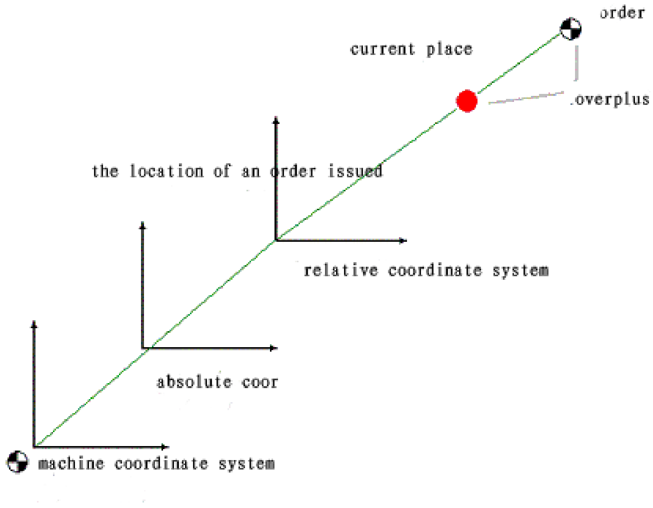


Fig.7.1-3

Setting workpiece coordinate system

Coordinate system must be setted on first for program editing, and the relationship of program orgin and start point of tool composes workpiece coordinate system; It is established by command G50.

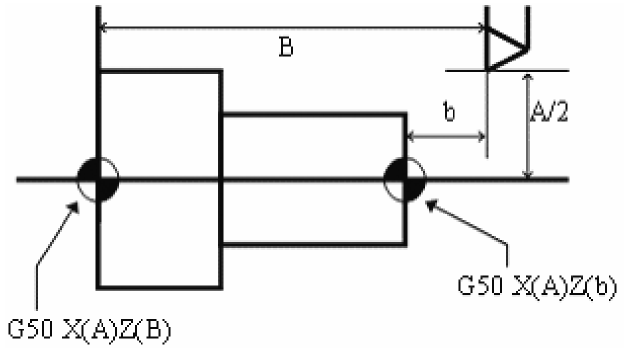
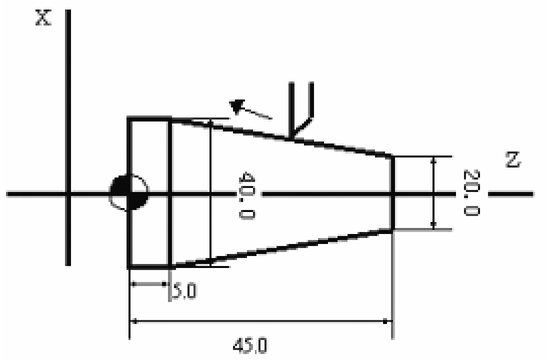


Fig.7.1-4

Absolute/relative coordinate system programming

NC lathe has two control shaft，and has two methods of programming: commanding methods of absolute coordinate and relative coordinate. Besides, these methods can be combined inti one command. The relative coordinate commands required by axis X and axis Z are U and V.



① absolute coordinate program---X40.Z5.;

② relative coordinate program ---U20.W-40.;

③ mixed coordinate program---X40.W-40.;

Fig.7.1-5

7.2 G CODE COMMAND

7.2.1 G CODE SET AND ITS MEANING

G Code	Group	Explanation
G00	01	Positioning
G01		Linear interpolation
G02		Circular interpolation/Helical interpolation CW
G03		Circular interpolation/Helical interpolation CCW
G04	00	Dwell
G09		Exact stop
G20	06	Input in inch
G21		Input in mm
G22	04	Stored stroke check function on
G23		Stored stroke check function off
G27	00	Reference position return check
G28		Return to reference position
G29		Return from reference position
G30		2nd reference position return
G32	01	Thread cutting
G40	07	Cutter compensation cancel/Three dimensional compensation cancel
G41		Cutter compensation left/Three dimensional compensation
G42		Cutter compensation right
G50	00	Scaling cancel
G52		Local coordinate system setting

G53		Machine coordinate system selection
G70	00	Finish maching cycle
G71		Inside and outside diameter rough cutting cycle
G72		Step rough cutting cycle
G73		Pattern repeating
G74		Peck dring cycle-Z axis
G75		Grooving in X axis
G76		Thread cutting cycle
G80	10	Canned cycle cancel
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G87		Back boring cycle
G88		Back tapping cycle
G89		Back boring cycle
G90	01	Cutting cycle ‘A’
G92		Thread cutting cycle
G94		Cutting cycle ‘B’
G96	12	Constant surface speed control
G97		Constant surface speed control cancel
G98	05	Feed per minute
G99		Feed per rotation

Table.7.2-1 G code set and its meaning

(Codes with the sign of “*”means they can be initialized when boot-strap)

7.2.2 G Code Explanation

G00

➢ Positioning(G00)

1. Format

G00 X_ Z_

The G00 command moves a tool to a appointed position(in the method of absolute coordinate mode), or to some distances(in incremental coordinate mode).

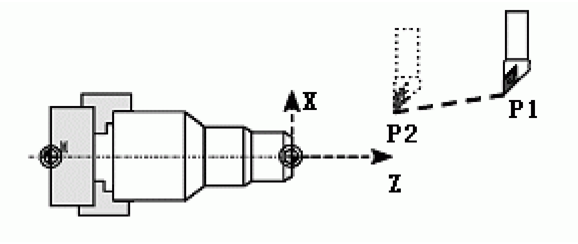


Fig.7.2-1

2. Nonlinear interpolation positioning

The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.

3. Linear interpolation positioning

The tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.

4. Example

N10 G00 X-100 Z-65

G01

➢ Linear Interpolation(G01)

1. Format

G01 X(U)_ Z(W)_ F_ ；

Linear interpolation makes tool move from current position to commang position by linear mode and command.

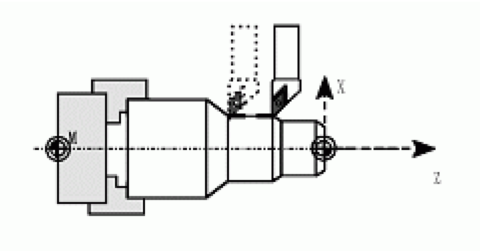


Fig.7.2-2

X, Z: The absolute coordinate value of the position required to move to

U, W: The incremental coordinate value of the position required to move to.

2. Example

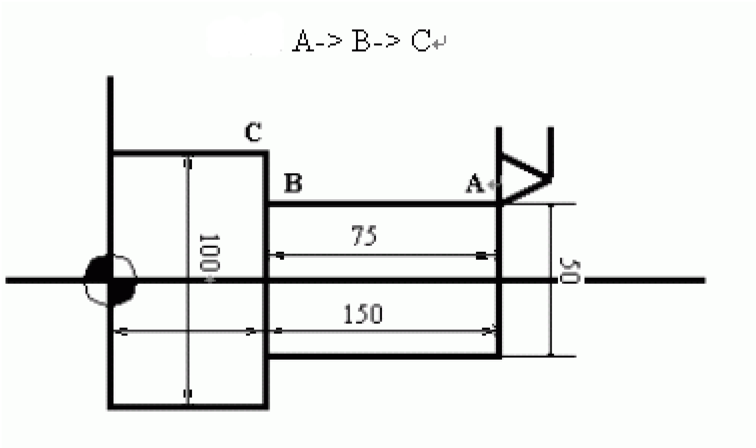


Fig.7.2-3

①

G01 X50. Z75. F0.2 ；absolute coordinate program

X100.；

②

G01 U0.0 W-75. F0.2 ；incremental coordinate program

U50.

G02/G03

➢ Circular interpolation(G02/G03)

When circular interpolation is being processed, tool must be on specified plane.Then determine the turning direction. Clockwise G02；Counterclockwise G03。

1. Format

G02(G03) X(U)__Z(W)__I__K__F__ ；
G02(G03) X(U)__Z(W)__R__F__ ；

X,Z – Specified end-point

U,W – Distance from start point to end point

I,K – Vector from start point to center point

R – Arc radius(less than 180 degree).

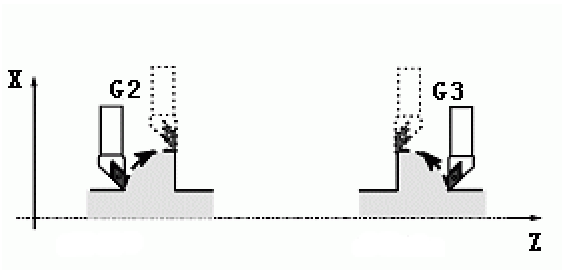


Fig.7.2-4

2. Example

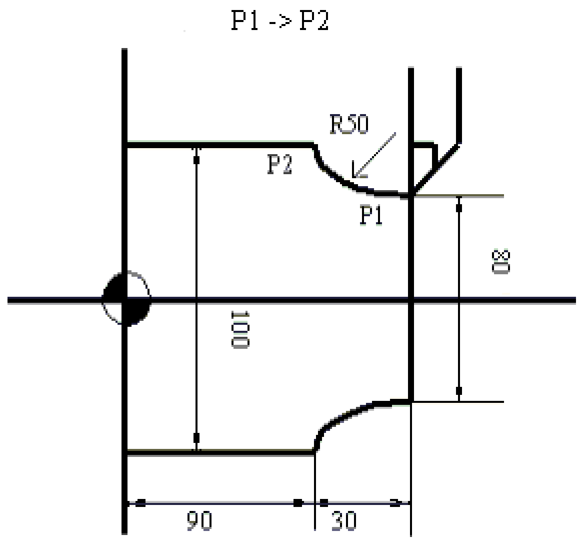


Fig.7.2-5

①

G02 X100. Z90. I50. K0. F0.2 ; absolute coordinate program

or G02 X100. Z90. R50. F0.2

②

G02 U40. W-30. I50. K0. F0.2 ; incremental coordinate program

or G02 U40. W-30. R50. F0.2

G04 Dwell

Format：G04X_; or G04U_; or G04P_;

X: Specified time(decimal point is permitted)

U: Specified time(decimal point is permitted)

P: Specified time(decimal point is unpermitted)

Explanation：G04 command dwell. The execution of the next block is delayed by the specified time, and Bit 1 (DWL) of parameter No. 3405 can specify dwell for each rotation in feed per rotation mode

G27 Reference position return check

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis goes on.

G28/G53 Reference position return

G28/G53 reference position return command is executed when the tool position is offsetted.

G30

➢ 2nd orgin return(G30)

The coordinate system can be setted by 2nd orgin function

1. Use parameter(a, b) to set coordinate value of start point of tool. “a” and “b” is the distance

from machine orgin to start point.

2. Use G30 to set coordinate system replacing G50 in programming.

3. Tool moves to 2nd orgin when this command is processed wherever tool is after 1nd orgin is executed.

4. Change tool at 2nd orgin.

G32

➢ Thread cutting(G32)

1. Format

G32 X(U)__Z(W)__F__ ；

F –lead of thread setting

There must be the function G97 with RPM symmetrical control, and some characters must be considered when edit program for thread cutting.

2. Example

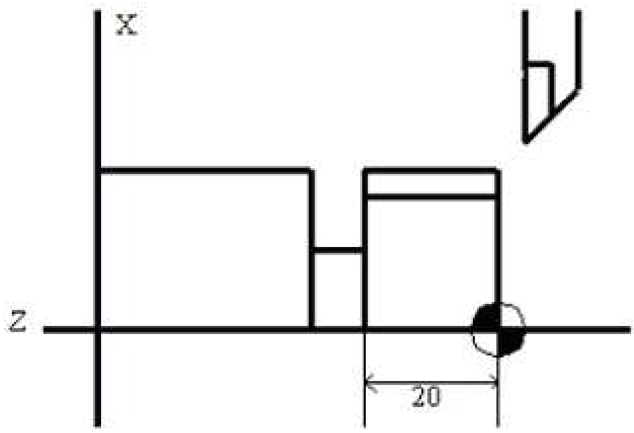


Fig.7.2-6

G00 X29.4

G32 Z-23. F2 ； 1 cutting cycle

G00 X32

Z4.

X29.

G32 Z-23. F2 ； 2 cutting cycle

G00 X32.

Z4.

G40/G41/G42

➢ Tool radius compensation (G40/G41/G42)

1. Format

G41 X_ Z_；   G42 X_ Z_；

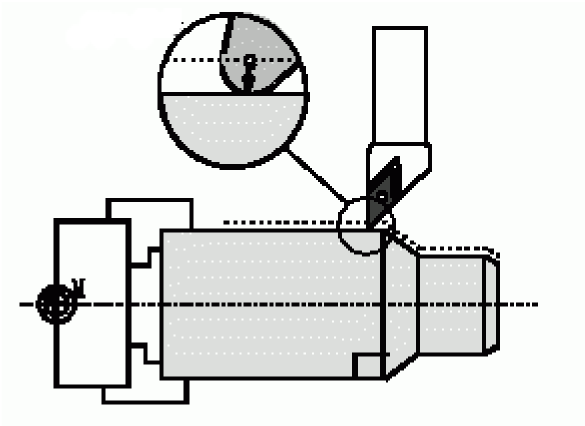


Fig.7.2-7

When the edge is a imaginary tool nose, cutting prosess which is executed according to the shape specified by program will not be error. But, real edge is composed by arc as the Fig.7.2.7 shows, and path of tool nose will make error in the case of circular interpolation.

2. Offset function

Command	Cutting position	Tool path
G40	Cancel	Tool moves according to program path
G41	Right	Tool offsets at the left side of path
G42	Left	Tool offsets at the right side of path

Table.7.2-2

Principle of compensation is lied on movement of the arc center of tool nose, and it is always mismatched with normal radius vector of cutting sutface. Therefore， the reference point is the center of tool nose. Generally, the reference point of compensation is the center of tool nose.

When the principle is used in cutter compensation, radius of tool nose should be measured by the reference points of X and Z.There are some type numbers used for radius compensation of imaginary tool nose(1-9).

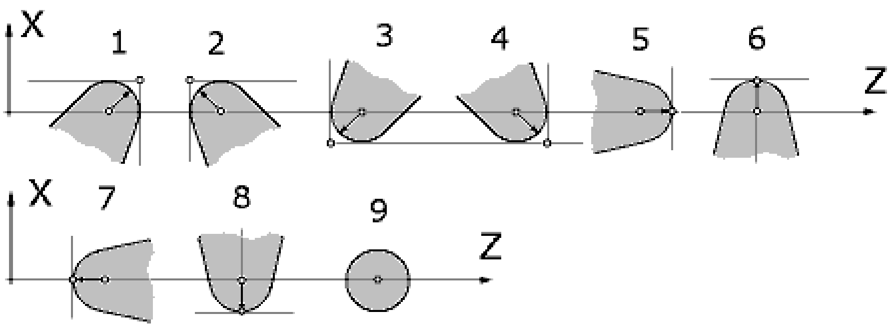


Fig.7.2-8

These contents should be inputted into cutter offset file in advance.

“nose radius offset” should be ordered or cancel by G00 or G01. Tool will not move correctly and lapse the executing path whatever if it is with circular interpolation.Therefore, nose radius offset should be finished before cutting starts; And can avoid over cutting. Contrarily, the cancel of offset is executed by moving command after cutting.

3. Example:

G41 X5 Z5 D1；

G02 X25 Z25 R25；

G40 G01 X10 Z10 D0；

G50 Workpiece coordinate system setting

Format：G50IP_

Illustration：When workpiece coordinate system is setted, the point on tool is in specified place.If IP_ is incremental command value, current tool position is matched with primary tool position adding specified incremental value after workpiece coordinate system is setted.If coordinate system is setted by G50 in offset, the position before offset is matched with the position specified by G50.

G54～G59

➢ Workpiece coordinate system selection(G54～G59)

1. Format

G54 X_ Z_；

2. Function

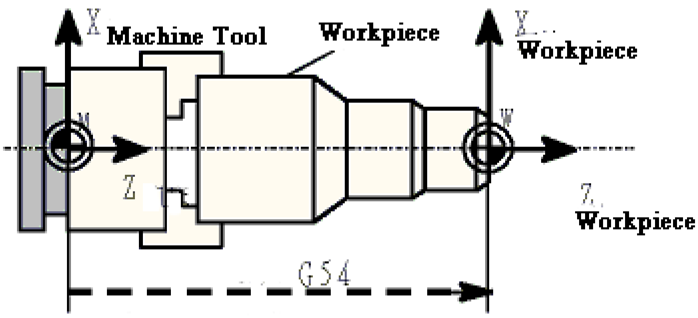


Fig.7.2-9

At most six workpiece coordinate systems can be setted by using G54～G59.

When the power is on and orgin is returned, system will select coordinate system automaticly 1 (G54).They keep the validity before“modality”change these coordinate.

G70

➢ Finish turning cycle(G70)

1. Format

G70 P(ns) Q(nf)

ns: First segment number of finish maching shap program nf: Last segment number of finish maching shap program

2. Function

Rough turning by G71、G72 or G73， then finish turning by G70.

G71

➢ Excircle rough turning canned cycle(G71)

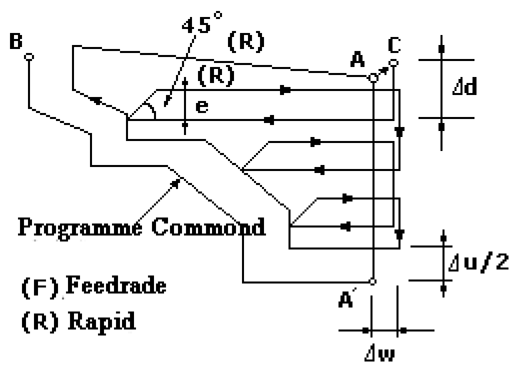


Fig.7.2-10

1. Format

G71U(△d)R(e) G71P(ns)Q(nf)U(△u)W(△w)F(f)S(s)T(t)

N(ns)……

………

F__ Program segments from ns to nf specify moving commands from A to B

S__

T__

N(nf)……

△d: Cutting depth(radius specifing)

Sign symbol is not specified. Cutting direction is determined by direction of AA',and it will not change before a value is specified.FANUC system parameter（NO.0717）specifing.

e: Travel of back off

This command is state command.It will not change before a value is specified. FANUC system parameter（NO.0718）specifing.

ns: First segment number of finish maching shap program

nf: Last segment number of finish maching shap program

△U: Distance and direction of finish machining obligated amount in X direction.（diameter/radius）

△W: Distance and direction of finish machining obligated amount in Z direction.

f,s,t: Any F, S or T function included in nsto nf is ignored in cycle.While in G71, F,S is valid.

2. Function

As the last Fig shows, finish maching shape from A to A’to B is determined byprogram,and finish machining obligated amount △u/2 and △w is obligated in specified area turned by △d(cutting depth).

G72

➢ Face cutting canned cycle(G72)

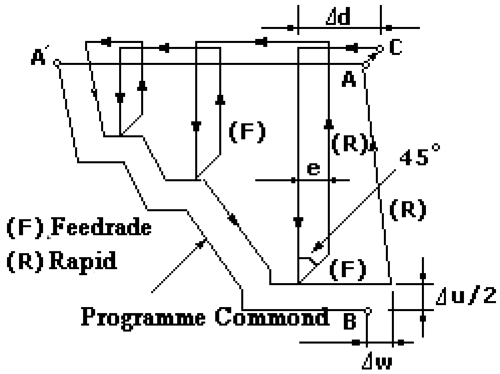


Fig.7.2-11

1. Format

G72W（△d）R(e)
G72P(ns)Q(nf)U(△u)W(△w)F(f)S(s)T(t)

The meanings of △d,e,ns,nf, △u, △w， f,s and t are the same with G71.

2. Function

As last Fig shows, this cycle except parallel with X is the same with G71.

G73

➢ Contour machining multiple system cycle(G73)

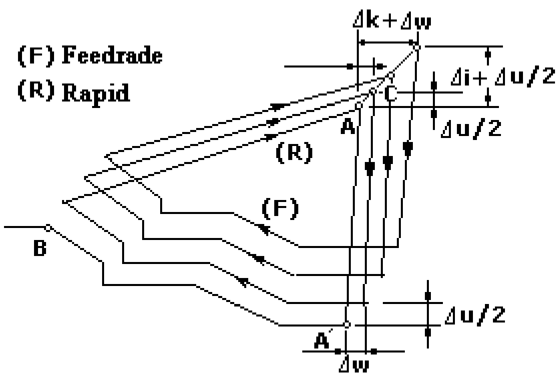


Fig.7.2-12

1. Format

G73U(△i)W(△k)R(d)
G73P(ns)Q(nf)U(△u)W(△w)F(f)S(s)T(t)

N(ns)………

…………

F__ Program segments from ns to nf specify moving commands from A to B

S__

T__

N(nf)………

△i: Distance of back off in X direction(radius specifing), FANUC system parameter（NO.0719）specifing.

△k: Distance of back off in Z direction(radius specifing), FANUC system parameter（NO.0720）specifing.

d: Times of division

This value is the same with repeat times of machining, FANUC system parameter（NO.0719）specifing.

ns: First segment number of finish maching shap program

nf: Last segment number of finish maching shap program

△U: Distance and direction of finish machining obligated amount in X direction.（diameter/radius）

△W: Distance and direction of finish machining obligated amount in Z direction.

f,s,t: Any F, S or T function included in nsto nf is ignored in cycle.While in G71, F,S is valid.

2. Function

This function is for repeat cutting of a canned format changing slowly.This cycle can cut a shaped workpiece machined by rough maching methods effectively.

G74

➢ Endface pecking drilling cycle(G74)

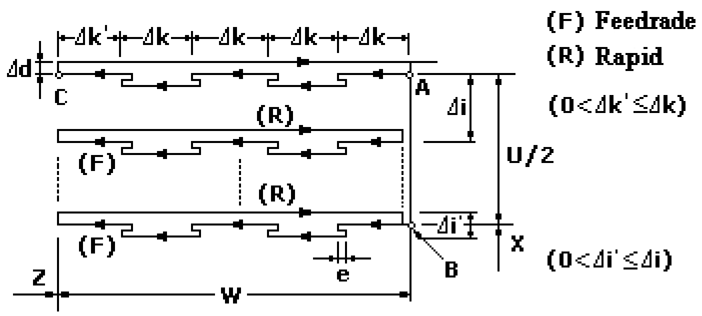


Fig.7.2-13

1. Format

G74 R(e);
G74 X(u) Z(w) P(△i) Q(△k) R(△d) F(f)

e: Quantity of back off

This command is state command.It will not change before a value is specified. FANUC system parameter（NO.0722）specifing.

x: X coordinate of B

u: Increment from A to B.

z: Z coordinate of C

w: Increment from A to C

△i: distance of moving in X direction(no symbol)

△k: distance of moving in Z direction(no symbol)

△d: Amount of back off when cutting bottom. Symbol of △d must be（+）.But, if X（U）and △I are ignored, direction of back off can be specified.

f: Feedrate

2. Function

As the last Fig shows, the cycle can process break cutting.If X（U）and P are ignored,just Z axis is processing for drilling.

G75

➢ External diameter / internal diameter pecking drilling cycle(G75)

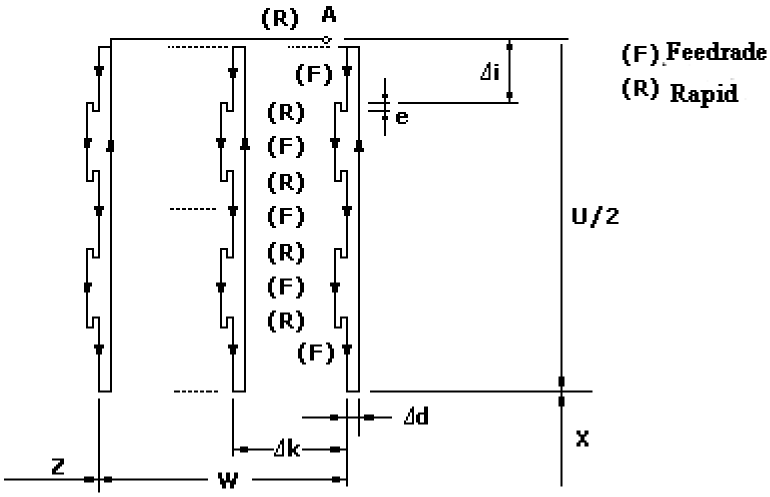


Fig.7.2-14

1. Format

G75 R(e)；

G75 X(u) Z(w) P(△i) Q(△k) R(△d) F(f)

2. Function

As last Fig shows，the cycle can process break cutting, and can cut channel in X axis and pecking drill.

G76

➢ Thread cutting cycle(G76)

1. Format

G76 P(m)(r)(a) Q(△dmin) R(d)

G76 X(u) Z(w) R(i) P(k) Q(△d) F(L)

m: Number of replication for finish maching（1 to 99）

This command is state command.It will not change before a value is specified. FANUC system parameter（NO.0723）specifing.

r: Quantity of chamfer

This command is state command.It will not change before a value is specified. FANUC system parameter（NO.0109）specifing.

a: Angle of tool nose：

You can select 80 degree、60 degree、55 degree、30 degree、29 degree、0 degree，and specify it with 2 digit.

This command is state command.It will not change before a value is specified. FANUC system parameter（NO.0724）specifing.For example：P（02/m、12/r、60/a）

△dmin: Minimum of cutting depth expressed by radius.

This command is state command.It will not change before a value is specified. FANUC system parameter（NO.0726）specifing.

d: Allowance for finish

i: Semidiameter of threaded portion

If i=0,it is seen as normal linear thread cutting.

k: Height of thread expressed by radius value.

△d: Cutting depth for fist time（radius value）

L: Lead of thread（as G32）

2. Function

Thread cutting cycle.

G80-G89 drilling canned cycle

Generally, drilling needs several blocks, and if you want to use canned cycle just one command is used to predigest proframming.

Illustration：

Drilling G code specifys fixation axis and drilling axis as the following shows. Axis C and X or Z is used as fixation axis,while axis X or Z not used as fixation axis is used as drilling axis.

Drilling mode:

G83 and G85/G87 and G89 is mode G code.When it is valid, its state is drilling.When drilling mode and drilling data is specified, it keeps its state.When canned cycle starts,necessary drilling data is appointed,and when canned cycle is executed just data modifying is appointed.

G90

➢ Inside and outside diameter cutting cycle(G90)

1. Format

Linear cutting cycle：

G90 X(U)___Z(W)___F___ ；

Press swith to login single block.As Fig shows, 1→2→3→4 cycle.The symble (+/-) of U and W change according to direction changing of 1 and 2 in incremental coordinate program.

Cone cutting cycle：

G90 X(U)___Z(W)___R___ F___ ；

"R" value of cone must be appointed.The function is similary with linear cutting cycle.

2. Function

Lathe turning cycle

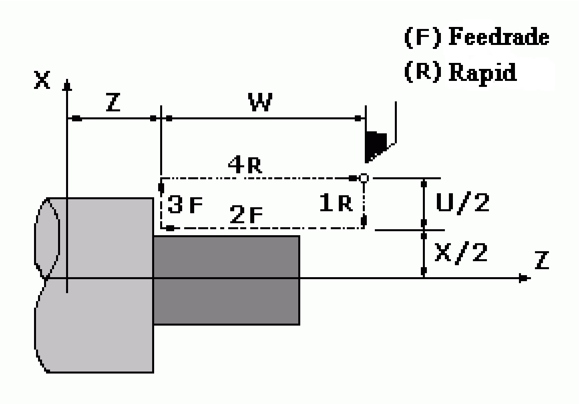


Fig.7.2-15

1. U<0, W<0, R<0

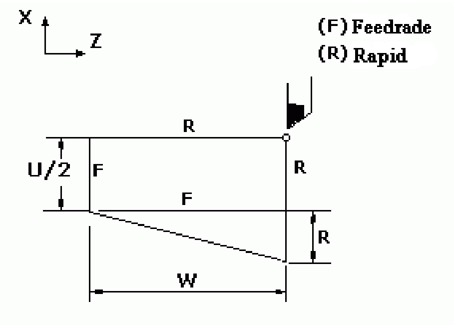


Fig.7.2-16

2. U>0, W<0, R>0

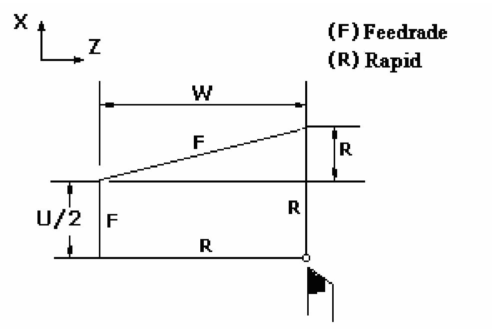


Fig.7.2-17

3. U<0, W<0, R>0

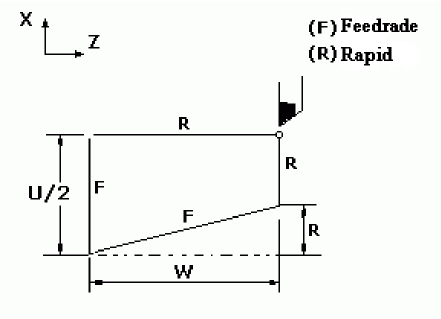


Fig.7.2-18

4. U>0, W<0, R<0

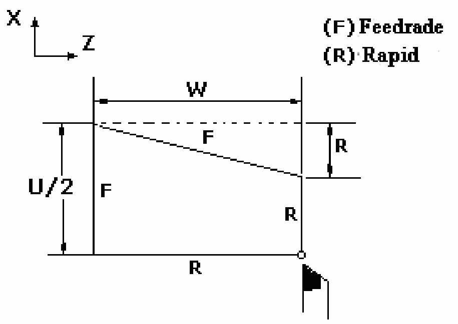


Fig.7.2-19

G92

➢ Thread cutting cycle(G92)

1. Format

Straight thread cutting cycle：

G92 X(U)___Z(W)___F___ ；

Thread range and spindle steady control RPM(G97) is similary with G32(thread cutting). In this cycle, back off as [Fig. 9-9] shows；Size of chamfer is setted 0.1L units in 0.1L～12.7L according to assigned parameter.

Taper thread cutting cycle：

G92 X(U)___Z(W)___R___F___ ；

2. Function

Thread cutting cycle

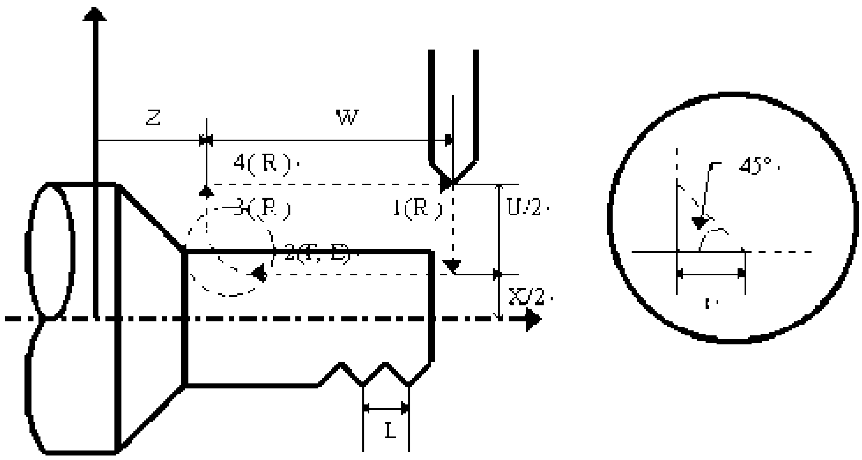


Fig.7.2-20

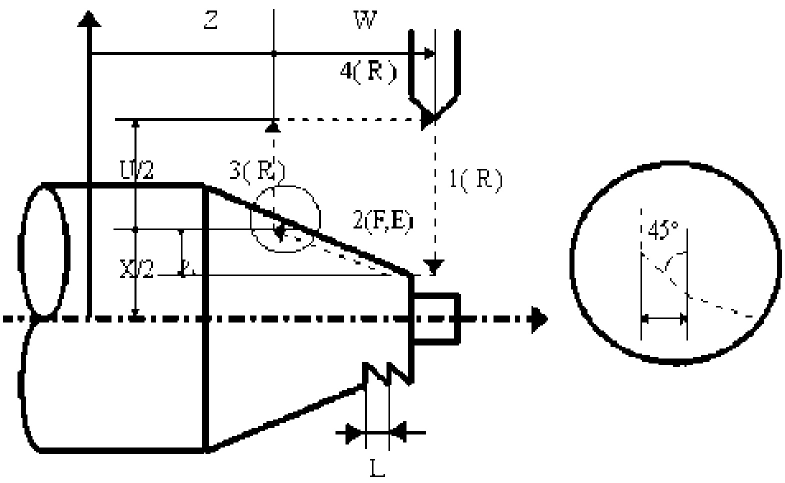


Fig.7.2-21

G94

➢ Step cutting cycle(G94)

1. Format

Flat step cutting cycle：

G94 X(U)___Z(W)___F___ ；

Wimble step cutting cycle：

G94 X(U)___Z(W)___R___ F___ ；

2. Function

Step cutting

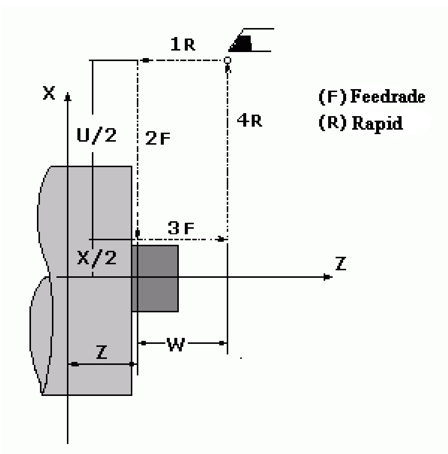


Fig.7.2-22

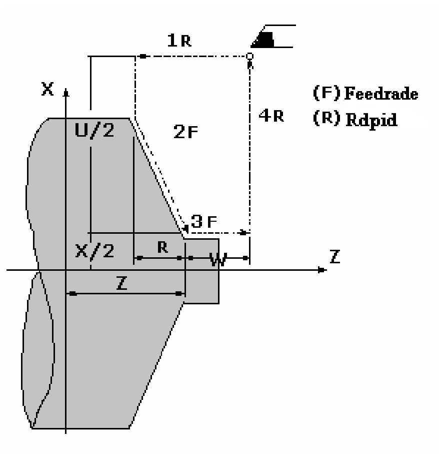


Fig.7.2-23

G96/G97

➢ Linear rate control(G96/G97)

Spindle of NC machine is divided into low speed and high speed area；Rate can be changed freely in each area.

G96 is used to execute constant linear rate control, and control changing of diameter of work to keep constant cutting rate by changing rotate speed only.It is used with G50.

G97 is used to cancel constant linear rate control, and control the stasdy of rotate speed only.

G98/G99

➢ Feedrate per minute/feedrate per minute setting(G98/G99)

Feedrate can be specified by G98（mm/m），or by G99（mm/r）.G99 is always used in NC lathe maching.

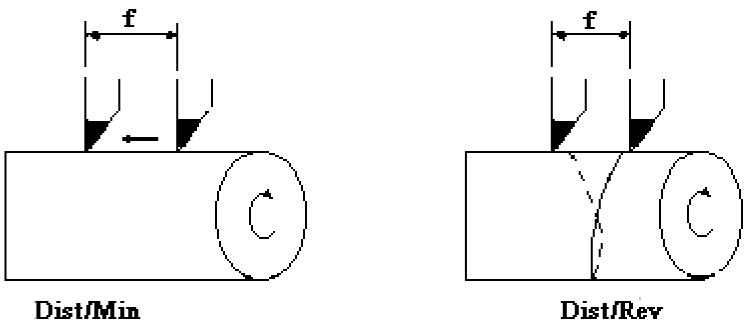


Fig.7.2-24

Moving rate per minute(mm/m) = rate of displacement per round(mm/r) x spindle speed

7.3 AUXILIARY FUNCTION（M FUNCTION）

Code	**Function**
M00	Program stop
M01	Optional stop
M02	End of program
M30	End of program reset
M03	Spindle corotating
M04	Spindle reversing
M05	Spindle stop
M08	Cutting solution open
M09	Cutting solution close
M40	Spindle gear at middle
M41	Spindle low gear override
M42	Spindle high gear override
M68	Hydraulic chuck clamping

M69	Hydraulic chuck clamping loosening
M78	Tailstock advancing
M79	Tailstoc reversing
M94	Mirrorimage cancel
M95	Mirrorimage of X axis
M98	Calling of subprogram
M99	End of subprogram

Table.7.3-1 Auxiliary function(M function)

7.4 PRESETTING CUTTER OF LATHE

Operation steps：

First、Methods for setting workpiece orgin of FANUC 0-TDⅡ NC lathe：

1、 Test cutting for presetting directly

(1)firstly use billmpse tool to cut one excircle, measure excircle diameter, press MENU OFSET →[GEOM]input “MX out round diameter value”，click INPUT key，and then inputs to cutting tool geometry shape。

(2)again use billmpse tool to cut excircle end surface，press MENU OFSET →[GEOM]input“MZ 0”，press INPUT key，and that inputs to cutting tool geometry shape。

2、 use G50 to set workpiece zero point

(1) use billmpse tool to cut one excircle，select[REL]、press 1 U、press CAN key to reset “zero”，after measure out round diameter，back off along Z positive direction，select MDI mode, input G01 U..F0.3cut surface to center。

(2) select MDI mode，input G50 X0 Z0，run CYCLE START key，set current to zero。

(3) select MDI mode，input G0 X150 Z150 ，run CYCLE START key，feed machining。

(4) program begin：G50 X150 Z150 ……。

(5) Attention：use G50 X150 Z150，program beginning and ending must be in accord with X150 Z150，thus can promise repeated machining not to mistake.

(6) use second reference point G30，thus can promise repeated machining not to mistake.，program beginning

G30 U0 W0

G50 X150 Z150

(7) In FANUC system, set second reference point in the parameters， machine presetting finishes(X150 Z150),press mouse right-key to appear dialog box.

3、 set workpiece zero point

(1) In FANUC0-TD system such as MENU OFSET,there is a workpiece moving interface，can input zero point excursion value.

(2) use billmpse tool to test cut workpiece end face firstly，now the position of X、Z coordinate as follows： X-260 Z-395， directly input to excursion value.

(3) select HOME， then press +X +Z， here workpiece zero point coordinate system is founded.

(4) Attention：hold on this zero point，so long as reset to excursion value Z0，this can be deleted.

4、 G54～G59 set workpiece zero point

(1) use out-cut try to cut one out round， then measure out round diameter, then back off along Z positive direction. Cut face to center。

(2) input current X and Z coordinate to G54～G59 directly. Program directly usesuch as:G54 X50 Z50…….

(3) Attention: use G53 delect G54～G59workpiece coordinate.

Second、Methods of FANUC 0iTset zero point：

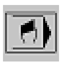
Operation step：

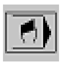
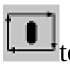
1、Test cutting for presetting directly

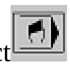
(1) Use billmpse tool to test cut a excircle， then measure the diameter ， press OFSET SET →[OFFSET]→[GEOM]input "out round diameter value"， press [MEASUR] key ， cutting tool"X"repair value is inputted into geometry shape automaticly.

(2) Use billmpse tool to test cut excircle endface， then press OFSET SET →[OFFSET]→[GEOM]input "Z 0"， press[MEASUR]， workpiece "Z"repair value is inputted into geometry shape automaticly。

2、 Use G50 to set workpiece zero point

(1) Use billmpse tool to test cut a section of excircle， press [REL] then press SHIFT → X U， now "U"is shining。Press ORIGIN to set "zero"， test out round ， select  "MDI"mode， input G01 U-××(××is test diameter)F0.3,cut face to center。

(2) select  MDI mode， input G50 X0 Z0， set to zero。

(3) select MDI mode， input G0 X150 Z150 ， make tool out of workpiece。

(4) Begin with：G50 X150 Z150 ……。

(5) Attention：use G50 X150 Z150，program beginning and end must consist.

(6) use second reference point G30，program begin with

G30 U0 W0

G50 X150 Z150

(7) In FANUC system，set second reference pointing the parameters. After presetting(X150 Z150), press mouse right-key to appear dialog box，press mouse left key to make sure。

3、 Workpiece orgin setting of workpiece moving

(1) in FANUC0 system such as OFSET SET， there is a interface of workpiece moving，and you can input zero point excursion value。

(2) Use billmpse tool to test cut workpiece face, then the position of X、Z is: X-260 Z-395, direct inputs into excursion value。

(3) select ， along X、Z back to reference point。

(4) attention：hold on zero point，so long as reset excursion value Z0，it can be cleared。

4、 G54～G59 sets workpiece zero point

(1) Use billmpse tool to test cut a excircle, press OFSET SET → → [COORD], if selectG55, input X0、Z0,then press [MEASUR]， program such as:G55 X60 Z50……。

(2) Attention: use G53 direction to clear G54～G59direction。

7.5 EXAMPLE

G90 Inside and outside diameter cutting cycle

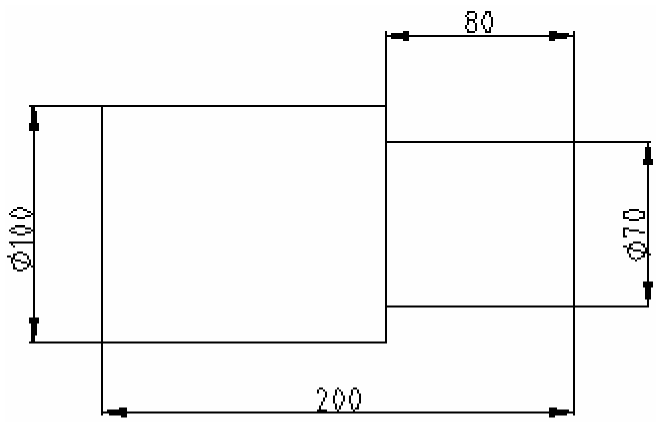


Fig.7.5-1

T0101 ；cutter compensation

M03 S1000

G0 X105 Z5

G90 X90 Z-80 F0.3 ；calling Inside and outside diameter cutting cycle for rough turnning

X85 ; repeat calling cutting cycle

X80

X75

X70 ; size of cutting

G0 X100 Z100

T0100 ; cutter compensation cancel

M05

M30

G92 Thread cutting cycle

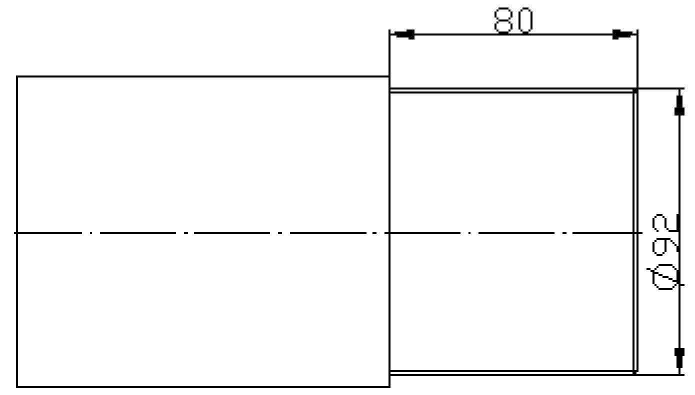


Fig.7.5-2

T0101 ; cutter compensation

M03 S100

G0 X102 Z10

G92 X98 Z-80 F0.3 ; calling Thread cutting cycle

X96 ; repeat calling Thread cutting cycle

X94

X92 ; size of thread cutting

G0 X200 Z100

T0100 ; cutter compensation cancel

M05

M30

G94 step cutting cycle

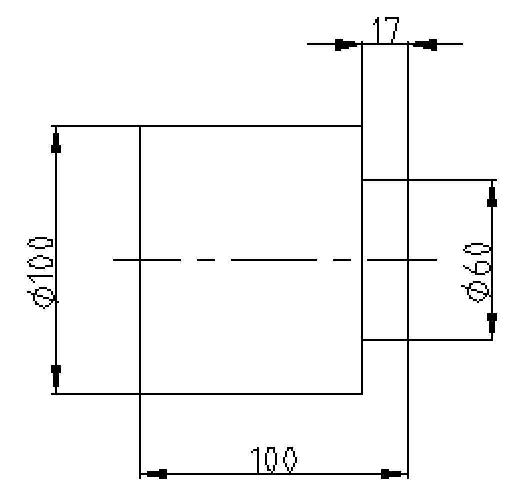


Fig.7.5-3

T0101 ; cutter compensation

M03 S1000

G0 X105 Z5

G94 X60 Z-5 F0.3 ; calling step cutting cycle

Z-9 ; repeat calling step cutting cycle

Z-13

Z-17 ; Size of cutting

G0 X100 Z100

T0100 ; cutter compensation cancel

M05

M30

G70 Finish maching cycle

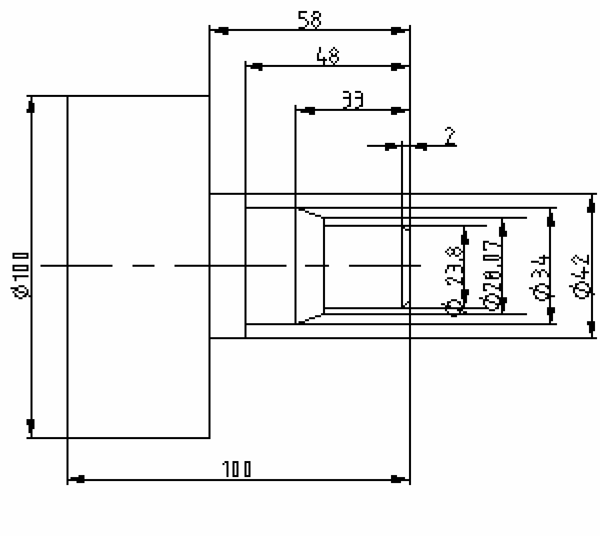


Fig.7.5-4

N010 T0101 ; cutter compensation

N020 M3 S800

N030 G0 X45 Z2

N040 G71 U2 R1 ; calling rough turning cycle format

N050 G71 P060 Q130 U0.25 W0.1 F0.25 ; calling program N number

N060 G0 X15.8

N070 G1 X23.8 Z-2

N080 Z-25

N090 X28

N100 X34 Z-33

N110 Z-48

N120 X42

N130 Z-58

N140 G0 X100 Z100

N150 X45 Z3

N160 G70 P060 Q130 ; calling finish machining cycle

N170 G0 X100 Z100

N180 T0200 ; back off

N190 M05

N200 M30

G72 Face cutting canned cycle

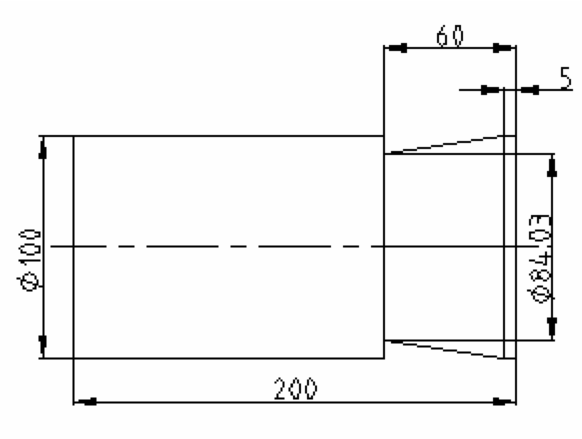


Fig.7.5-5

N10 T0101 M03 S1000 ; cutter compensation

N20 G0 X102 Z2

N30 G72 W7.0 R1.0 ; calling face cutting canned cycle format

N40 G72 P50 Q100 U4.0 W2.0 F0.3 S550 ; calling program N number

N50 G0 X110 Z10

N60 G01 X100 W-12 F0.15

N70 W-10

N80 X95 W-10

N90 W-20

N100 X80 W-22

N110 G0 X100 Z100

N120 T0100 ； cutter compensation cancel

N130 M05

N140 M30

G73 Contour machining multiple system cycle

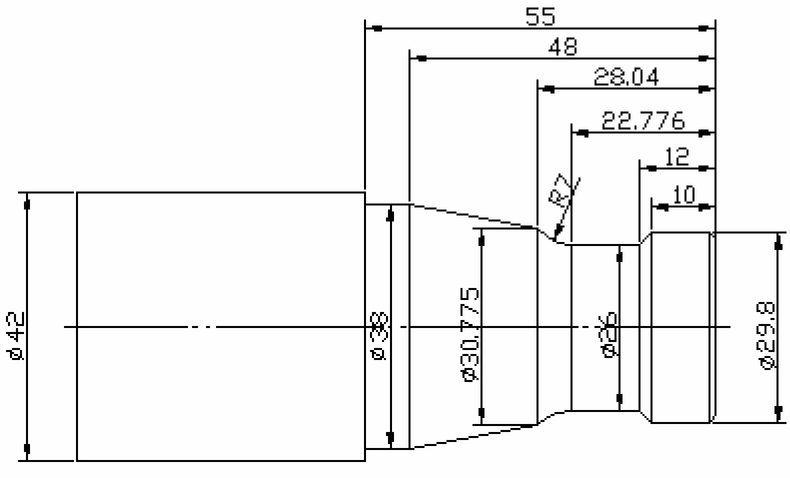


Fig.7.5-6

N10 G97 G99 S1200 M03 T0101 ； 1# cutter compensation

N20 G0 X44 Z-1 ； approaching workpiece

N30 G01 X-1 F0.05 ； endface turning

N40 Z2

N50 G0 X40 Z2

N55 G73 U7 W0 R7 ； calling contour machining multiple system cycle

N60 G73 P70 Q160 U0.6 W0.3 F0.1

N70 G0 X27.8 Z2 S1500 M03

N80 G01 Z0 F0.05

N90 X29.8 Z-1

N100 Z-10

N110 X26 Z-12

N120 Z-22.776

N130 G02 X30.775 Z-28.041 R7

N140 G01 X38 Z-48

N150 Z-55

N160 X42

N170 G0 X80 Z1

N180 G70 P70 Q160 ; calling finish maching cycle

N190 G0 X200 Z200

N200 M05

N210 T0100 ; cutter compensation cancel

N220 M30

G74 Endface pecking drilling cycle

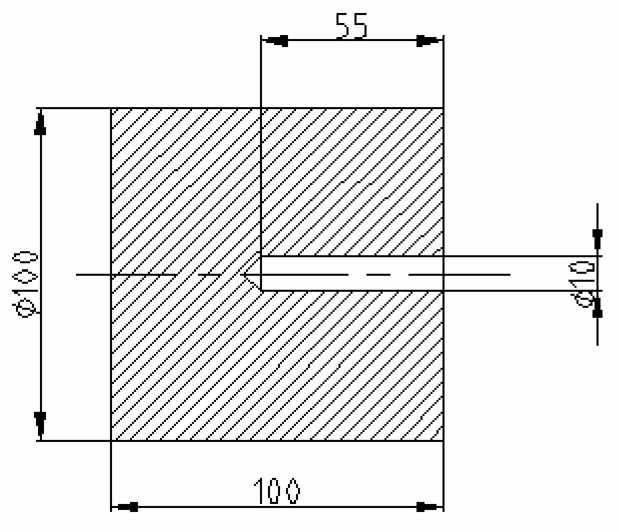


Fig.7.5-7

T0101 ; 1# cutter compensation

M3 S800

G0 X0 Z2

G74 R1 ; calling endface pecking drilling cycle format

G74 Z-60 Q3000 F0.1 ; depth of drilling

G0 X100 Z100

T0100 ; cutter compensation cancel

M05

M30

G75 outside diameter/internal diameter pecking drilling cycle

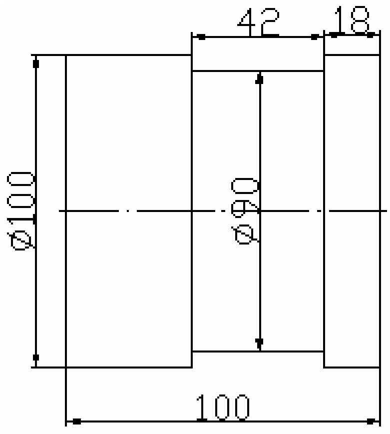


Fig.7.5-8

T0101 ; cutter compensation

M3 S800

G0 X105 Z2

X105 Z-22

G75 R2 ; calling drilling cycle format

G75 X90 Z-60 P3000 Q3000 R0 F0.1

G0 X100 Z100

T0100; cutter compensation cancel

M05

M30

G76 Thread cutting cycle

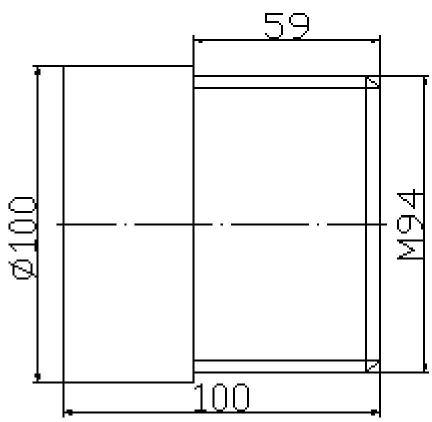


Fig.7.5-9

T0101 ; cutter compensation

M03 S800

G0 X105 Z2

G76 P010060 Q100 R0.1 ; calling thread cutting cycle

G76 X94 Z-59 P1200 Q400 F2

G0 X110 Z110

T0100 ; cutter compensation cancel

M05

M30

G90 cone cutting cycle

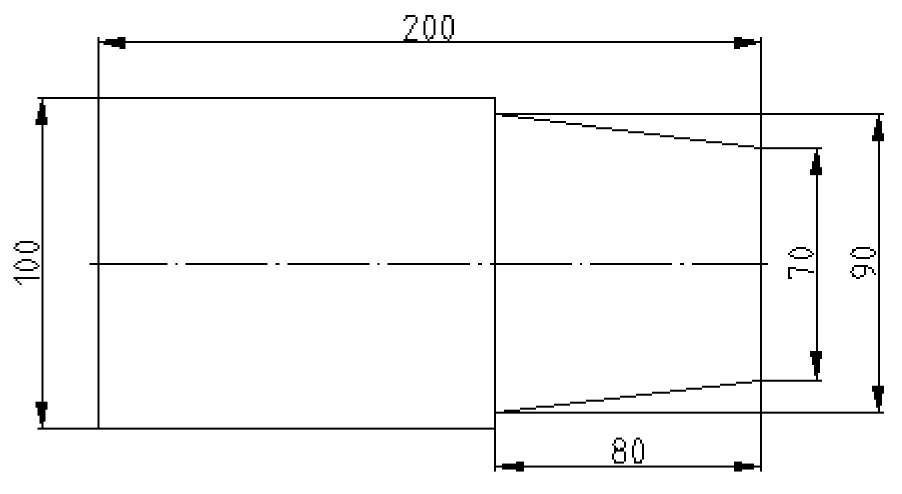


Fig.7.5-10

T0101 G0 X200 Z200 ; cutter compensation

M03 S1000 ; rotate speed 1000

G0 X105 Z5

G90 X90 Z-80 R-10 F0.3 ; calling cone cutting cycle

U-10

G0 X100 Z100

T0100 ; cutter compensation cancel

M05

M30

M98/M99 calling subprogram

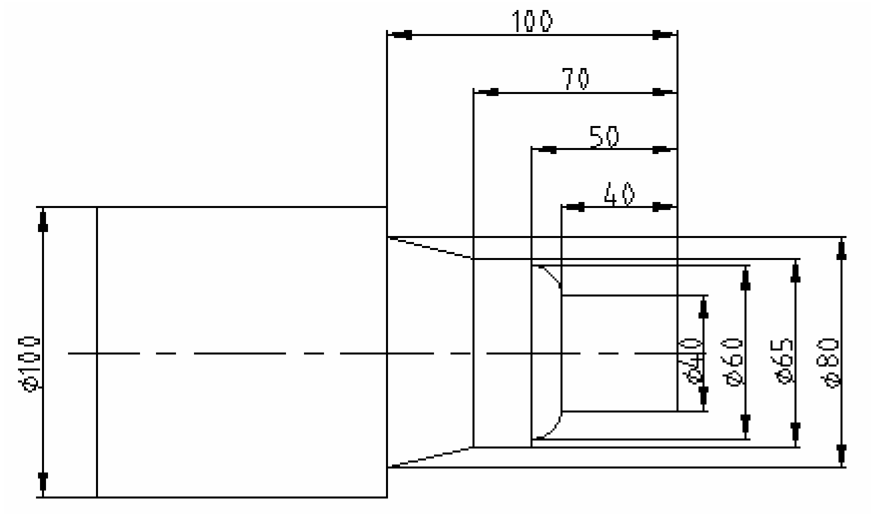


Fig.7.5-11

Mainprogram:

O0012

N010 M03 S1000

N020 T0101 ; 1# cutter compensation

N030 G00 X40 Z2

N040 M98 P20090 ; calling secondary subroutine name O 0090

N050 G00 X120 Z80

N060 M05

N070 M30

%

Subprogram:

O0090

N010 G1 Z-40 F0.3 ; subprogram

N020 G3 X60 Z-50 R10

N030 G1 X65

N040 Z-70

N050 X80 Z-100

N060 M99 ; return to mainprogram

%

Integrative example

T1: Excircle roughing turning tool T2: Excircle smoothing-tool T3: Thread tool T4: Aiguille

T5: Hole boring cutter

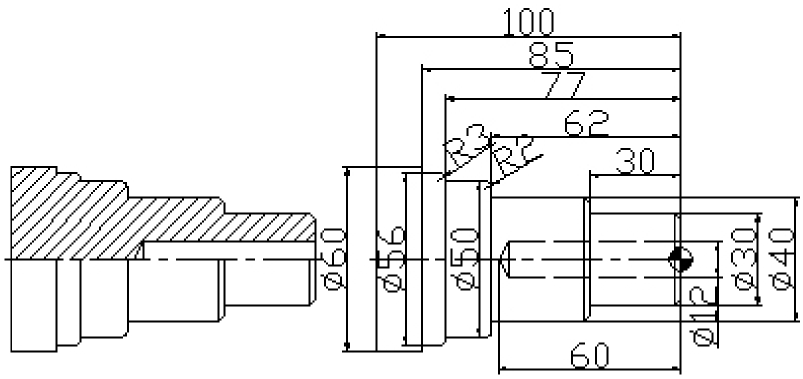


Fig.7.5-12

Program:

N010 G30 U0. W0. ; Return to 2nd reference point

N015 G50 X0. Z0. T0100 ; Establishing workpiece coordinate system, changing tool T01

N020 G96 S1500 M03 ; Spindle turning, constant linear rate

N025 G00 X60 Z0. T0101 ; calling T01 cutter compensation

N030 G01 X-1. F0.5

```
N035 G00 X61. Z3.
N040 G71 U2. R0.5                        ； Rough cutting cycle
N045 G71 P50 Q115 U0.4 W0.2 F0.4         ； Rough cutting cycle
N050 G00 X20.                            ； subprogram
N055 G01 Z0.                             ； subprogram
N060 X22.                                ； subprogram
N065 Z-2. X30.                           ； subprogram
N070 Z-30. X30.                          ； subprogram
N075 Z-30. X36.                          ； subprogram
N080 Z-32. X40.                          ； subprogram
N085 Z-62. X40.                          ； subprogram
N090 Z-62. X46.                          ； subprogram
N095 G03 Z-64. X50. K-2. I0.             ； subprogram
N100 G01 Z-77. X50.                      ； subprogram
N105 G03 Z-80. X56. K-3. I0.             ； subprogram
N110 G01 Z-85. X56.                      ； subprogram
N115 Z-85. X57.                          ； subprogram
N120 G00 X100. Z30.
N125 X150. Z150. T0100                   ； back off and bring tool compensation down
N130 G00 X61. Z30. T0202                 ； changing tool T2
N135 G00 Z10.
N140 G70 P50 Q115                        ； Finish cutting cycle
N145 G40 G00 Z30.
N150 X150. Z150. T0200                   ； back off and bring tool compensation down
N151 G0 X0 Z170. T0404                   ； changing tool T4
N152 G0 Z1.
N153 G01 Z-60. F100
N154 G0 Z170. T0400                      ； back off and bring tool compensation down
N155 T0505                               ； changing tool T5
N156 G0 Z1.
N157 G01 Z-50. F100
N158 G0 Z170 T0500
N159 G97 S500 M03                        ； costant rotation rate
N160 G00 X61.Z3. T0303                   ； changing tool T3
N165 X42. Z-32.
N170 G76 P010060                         ； thread cutting cycle
N175 G76 X37.835 Z-57. P1083 Q300 F2.0   ； thread cutting cycle
N180 G00 X61. Z3
```

N185 X150. Z150. T0300 ; back off and bring tool compensation down

N190 M05 ; spindle stops

N195 M30 ; program stops

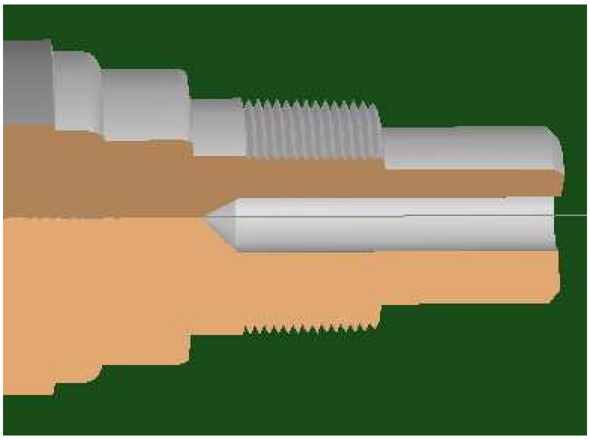


Fig. 7.5-13

CHAPTER 8 CUSTOM MACRO

8.1 VARIABLE

Normal workpiece programme specify G code and ship distance with numerical value directly. Such as GO1 and X100.0. Numerical value is specifyed directly or by variable when use custom macro. While when you use variable, numerical value can be modified by program or the operation of MDI panel.

＃1＝＃2＋100

G01 X#1 F300

Illustration：

- Variable expression

Variable name can be used by computer except custom macro. Variable is specified by variable symbol（＃）and variable number after it.

For example：＃1

Expression can be used to specify variable number. Here, expression must be enclosed in bracket.

For example：＃[#1+#2-12]

- Variable type

There are four variable types according to variable number

variable number	variable type	function
#0	dummy variable	This variable is always vacant, and no value can be assigned to it
#1-#33	local variable	local variable only can be used in macro to store data, such as operating result. When power cut, local variable is initialized to vacant. Independent variable assign local variable when call macro.
#100-#199 #500-#999	Common variable	Common variables have the same sensein different macro. When power cut,variable #100-#199 are initialized to vacant.Data of #500-#999 will be saved,even if power cut .
#1000---	system variable	system variable is used to read and write changement of datas when CN program is running.For example, current position and compensation value of tool.

- Range of variable value

Local variable common variable can have the value 0 or the value:

$-10^{47}$to-10^{-29}or$-10^{-2}$to-10^{47}

P/S warning NO.111 will accur if computing result overtop effective range.

- Omit of radix point

Radix point can be omitted when variable value is defined in program.

For example：When you define ＃1＝123；Actual value of variable ＃1 is 123.000.

- Citation of variable

When you specify variable with expression，put expression in bracket.

For example：G01X[#1+#2]F#3;

The cited variable is rounded automatically according to the least setting unit of address.

For example： When G00X# is executed at 1/1000mm， 123456 is assigned tovariable #1 by CNC.So its actual instruction value is G00X12346.

If you want to change the symbol of cited variable value, put（－）before ＃

For example：G00X－#1

When undefined variable is cited, variable and its address are ignored.

For example：When the value of variable ＃1 is 0 and the value of ＃2 is vacant,the result of G00X#1 Y#2 executing is G00X0.

- Common variable of double trails（double trails control）.

For double trails control, System provides separate macrovirable for every trail.But some common variable can use both of the trails at one time according to the setting of parameter N0.6036 and 6037.

- Undefined variable

When variable value is undefined, the variable becomes vacant variable. Variable ＃0 is aways vacant variable,and it is for reading only.

（a） Quotation

When a undefined variable is quotated, the address itself is ignored too.

when＃1=<vacant>	when＃1＝0
G90 X100 Y#1 ↓ G90 X100	G90 X100 Y#1 ↓ G90 X100 Y0

(b) Operation

<Vacant> is the same with 0 except the condition that you assign <vacant>.

when＃1=<empyt>	when＃1＝0
＃2＝＃1 ↓ ＃2＝< empyt >	＃2＝＃1 ↓ ＃2＝0

＃2＝＃*5 ↓ ＃2＝0	＃2＝＃*5 ↓ ＃2＝0
＃2＝＃1+＃1 ↓ ＃2＝0	＃2＝＃1+＃1 ↓ ＃2＝0

(c) Conditional expression

< vacant > in EQ and NE is different with 0.

when＃1=< vacant >	when＃1＝0
＃1EQ＃0 ↓ true	＃1EQ＃0 ↓ error
＃1 NE ＃0 ↓ true	＃1 NE ＃0 ↓ error
＃1 GE ＃0 ↓ true	＃1 GE ＃0 ↓ error
＃1 GT ＃0 ↓ error	＃1 GT ＃0 ↓ error

- Restriction

Program number，sequence number and jump number of optional block can’t use variable.

For example：Variable is unusable in such situation:

0＃1;

/＃2G00X100.0;

N#3Y200.0;

8.2 ARITHMETIC AND LOGIC OPERATION

The operations listed in the following list can be executed in variable.The expression at the right of operators can include constant and the variable composed with functions or operators.Variable ＃j and #k can be assigned with constant.And the left variable can be assigned with expression too.

function	format	remark
definition	#i=#j	

sine antisine cosine arccosine tangent arc tangent	#i=SIN[#j]; #i=ASIN[#j]; #i=COS[#j]; #i=ACOS[#j]; #i=TAN[#j]; #i=ATAN[#j];	Angle is expressed with degree. 90°30' for 90.5 degree
square root absolute value rounding Superior rounding Inferior rounding natural logarithm exponential function	#i=SQRT[#j]; #i=ABS[#j]; #i=ROUNND[#j]; #i=FIX[#j]; #i=FUP[#j]; #i=LN[#j]; #i=EXP[#j];	
or exclusive-or and	#i-#jOR#k; #i-#jXOR#k; #i-#jAND#k;	logical operation is executed by scale of two one bit by one bit
Turn BCD to BIN Turn BIN to BCD	#i=BIN[#j]; #i=BCD[#j];	It is used for handshaking with.

Illustration：

- Angle unit

  Angle unit of functions: SIN, COS, ASIN, ACOS, TAN and ATAN is degree. 90°30'for 90.5 degree.

- ARCSIN # i= ASIN[#j]

（1）Range of value to assign：

When the bit of parameter（NO.6004#0）NAT is setted 0:270°～90°

When the bit of parameter（NO.6004#0）NAT is setted 1:－90°～90°

（2）P/S warning NO.111 will accur if #j overtops the range:-1 to 1.

（3）Variable #j can be replaced with constant.

- ARCCOS ＃i＝ACOS[#j]

(1) Range of value to assign：180°～0°

(2) P/S warning NO.111 will accur if #j overtops the range:-1 to 1.

(3) Variable #j can be replaced with constant.

- ARCTAN #i= ATAN[#j]/ [#k]

(1) Specified length of the two sides,and split them by (/).

(2) Range of value to assign：

When the bit of parameter（NO.6004#0）NAT is setted 0:0°～ 360°

When the bit of parameter（NO.6004#0）NAT is setted 1:-180°～ 180°

(3) Variable #j can be replaced with constant.

- Natural logarithm #i=LN[#j];

(1) Attention: relative errormay be more than 10^{-8}.

(2) When antilog arithm (#j) is 0 or less than 0, P/S warning NO.111 will accur

(3) Variable #j can be replaced with constant.

- EXP function #i=EXP[#j]

(1) Attention: relative errormay be more than 10^{-8}.。

(2) Overflow and P/S warning NO.111 will accur when result of operation overtops $3.65X10^{47}$(j is about 110).

(3) Variable #j can be replaced with constant.

- ROUND function

(1) When ROUND function is included in arithmetic operation(or logical operation)commond IF or WHILE,ROUND function rounds at first decimal place. when execute #1= ROUND[#2](#2=1.2345),the value of variable 1 is 1.0.

(2) When ROUND function is used in address of NC sentence,ROUND function will specify a value to round according to least setting value of function address.

- Superior rounding and inferior rounding

When CNC processes numerical operation,if resulted integer absolute value is more than the absolute value of original number it is superior rounding,if less it is inferior rounding.and be careful for processing negative.

- Abbreviation of arithmetic and logic operation command

Two characters before funcyion name can be used to specify thefunction.

ROUND→RO

FLX→FI

- Order operation

(1) Function

(2) Multiplication and division operation

(3) Addition and subtract operation

- Parentheses nesting

Parentheses is used to change operational order,and it has 5 levels including the parentheses in function. P/S warning NO.118 will accur when it is over the 5th level.

LiMiT

- Bracket([,])is used to enclose expression. Attention: parentheses is for remark.
- Operational error Error may occur in operation.

Note: 1. Relative error lies on operation result.

2. Adopt the one whose error is less.

3. Absolute error is constant not operation result.

4. Function TAN execute SIN/COS.

5. If operation result of SIN/COS or TAN is less than $10X10^{-8}$ or is not 0 because of limit of operational precision,set parameter NO.6004# 1,and the operation result can be seen as 0.

(1) Precision of variable value is about a 8 bit decimal numeral. Anticipant result won't be got when process mortal data in addition/substract.

(2) Error may occur in using conditional expression EQ,NE,GE,GT.LE and LT.

(3) Be careful for using Inferior rounding command.

● **DIVISOR** P/S warning NO.112 will occur when specify 0 as divisor in division or TAN[90].

8.3 MACRO SENTENCE AND NC STATEMENT

Following blocks are macro statements:

◆ Blocks including arithmetic or logical operation (=).

◆ Program blocks including control statements

◆ Program blocks including call instruction of macro.

All program blocks except macro are NC statements.

Illustration:

● The differences with NC statements

(1) Machine won't stop even if it is at single code block running mode.But when parameter N0.6000#5SBM is setted as 1,machine will stop at single block running mode.

(2) Macro isn't processed as unshift block in tool radius compensation mode.

● The NC statement hasing the same characters with macro statement.

(1) It has subroutine call instruction,but doesn't eliminate 0.The characters of NC statement except N or L address command NC statement is the same with macro's.

(2) The characters of NC statement except ONP or L address command NC statement is the same with macro's.

8.4 TRANSFER AND CIRCLE

In program, GOTO statement and IF statement can control the flow of control.There are three kinds of transfer and circle for using.

Transfer and circle----------GOTO statement(unconditional transfer)

↓→IF statement (conditioned transfer)

WHILE statement（when…circle）

8.4.1 UNCONDITIONAL TRANSFER (GOTO STATEMENT)

Move to the block marked sequence number n.When specify the sequence number out of 1 to 99999,P/S warning NO.128 occurs. Sequence number can be specifyed by standout mode.

GOTOn;n: Sequence number (1 to 99999)

8.4.2 CONDITIONAL TRANFER(IF) STATEMENT

Specify conditional expression behind IF.

IF[<conditional expression >]GOTOn   If specifyed conditional expression is satisfied, move to the program segement marked sequence number n.If not,execue next program segement.

IF[<conditional expression >]THEN   If conditional expression is satisfied,execut macro statement preselected.Only one macro statement is executed.

Illustration:

- **CONDITIONAL EXPRESSION**

Conditional expression must include operators. Operator is inserted between two variables or constants, and enclosed with ([ ]).Variable can be replaced with expression.

- **OPERATOR**

Operator is composed with two letters,and used to compare magnitude of two value .Note: symbol NO can't be used.

operator	meanings
EQ	equal
NE	unequal
GT	greater than
GE	Less than or Equal to
LT	less than
LE	Less than or Equal to

Classical program:

The following program is used to comput the summation from 1 to 10.

```
09500;
#1=0;   store the initial value of sum variable
#2=1;   initial value of   summand   variable
N1 IF[#2 GT 10]GOTO 2;   move to N2 when summand is more than 10
#1=#1+#2;   compute sum
#2=#2+#1;   next summand
GOTOA1;     move to N1
N2 M30 ;    stop
```

8.4.3 CIRCLE(WHILE STATEMENT)

A conditional expression ia specifyed after WHILE.When the specifyed condition is satisfied,execute program from D0 to END.Otherwise goto the block after END.

Illustration:

When the specifyed condition is satisfied,execute program from D0 to END.Otherwise goto the block after END.This instruction format applys IF locution.The number after D0 and END is the sign of program execution range,and the value of sign is: 1,2,3.If the value out of 1,2,3 is used, P/S warning NO.126 will occur.

- **NEST**

The signs in DO-END circle can be used for many times according to requisition.But when there is crossed recirculation (superposition in DO), P/S warning NO.126 will occur.

Illustration:

◆Endless loop　　when DO is specifyed but WHILE is not, endless loop will occur from DO to END.

◆Handling time　　Sequence number is retrieved when there is label transfer statement in GOTO statement. Retrospective retrieval is slower than forward search.WHILE statement can reduce the time of processing.

◆Undefined variable　　<vacant> and 0 has different effection in using conditional expression of EQ or NE. <vacant> is regarded as 0 in other conditional expression.

◆Classical program

The following program is used to comput the summation from 1 to 10.

```
00001;
#1=0;
#2=1;
WHILE[#2LE10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

8.5 MACRO CALL

Call macro using following methods:

Macro call ------------ modeless call(G65)
----------- mode call(G66,G67)
------------ macro call by G code
------------ macro call by G code
------------ subprogram call by M code
------------ subprogram call by T code

LMT

- The differences between macro call and subprogram call

Macro call(G65) is different from subprogram call(G68),as the following words show:

(1) Independent variable can be specifyed by (G65), not by M98.

(2) When block M98 include another NC file,call subroutine after command is executed. By

contraries,G65 call macroprogra unconditionally.

(3) When block M98 include another NC file,machine stops in single block mode. By contraries,G65 dosn’t stop.

(4) Change the level of local variable with G65.M98 doesn’t change the level of local variable.

8.5.1 MODELESS CALL(G65)

Custom macro specifyed by address P is called, when G65 is specifyed.Anddatacan be transferred into custom macro body.

Illustration:

Call:(1) Program number specifyed by address P is used after G65.

(2) When repeat is asked, specify the number of replication from 1 to 9999after address L.When value of L elided, regard L as 1.

(3) When use independent variable specifying, its value is assigned to corresponding local variable.

●Independent variable specifying

There are two modes of independent variable specifying. Independent variable specifying I uses letters expect G, L, O, N and P, and each letter is used for once. Independent variable specifying II uses letters A,B,C and I,J and K(I is 1-10).Type specifyed by independent variable is changed automatically according to the used letters.

Independent variable specifying I

address	Variable number	address	Variable number	address	Variable number
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

(1) Address G,L,N,Q and P can’t be used in.

(2) Address unnecessary to specify can be elided.The corresponding local variable is setted as vacant.

(3) Address is unnecessary to be specifyed by alphabetic sequence.And it must accord the format of word address.But I,J and K need to be specifyed by alphabetic sequence.

Independent variable specifying II

Independent variable specifying II use each A,B,and C just once,whle I,J,K 10 times.

Independent variable specifying II is used to transfer variables such as 3D coordinate value.

address	Variable number	address	Variable number	address	Variable number
A	#1	K3	#12	J7	#23
B	#2	I4	#13	K7	#24
C	#3	J4	#14	I8	#25
I1	#4	K4	#15	J8	#26
J1	#5	I5	#16	K8	#27
K1	#6	J5	#17	I9	#28
I2	#7	K5	#18	J9	#29
J2	#8	I6	#19	K9	#30
K2	#9	J6	#20	I10	#31
L3	#10	K6	#21	J10	#32
J3	#11	I7	#22	K10	#33

Suffix of I,J,K is used to confirm the independent variable specifying sequence,and is not writed in actual pargramming.

Limitation:

- Format: G65 must be specifyed before every independent variable.
- Commixture of Independent variable specifying I, II CNC interior identify independent variable specifying I & II automatically. If independent variable specifying I&II mix,the later one specifyed is effective.
- Position of decimal The unit of argument data without decimal is the least setting unit of each address. Transfering of value of independent variable is based on variation of acutal system configuration of machine.Using decimally can improve program compatibility.
- Nesting call：Call can be nested for 4 levels , and so dose modeless call（G95）and mode call (G66). Subprogram call(M98) is not.
- Levels of local variable :

  1. Nesting of local variable is from 0 to 4 level.
  2. Main program is 0 level.
  3. Level of local variable add 1 once macro is called. Local variable of last level is stored in CNC.
  4. When M99 is executed in macro, control return to called program.Here, level of local variable subtract 1,and local variable value stored in program call is recoverd.

- Typical program: Weave a macro to machine hole of wheel. Circumferential radius is I. Initiative angle is A,and space is B,and drill number is H. Centre of circle is(X,Y).Command can be specifyed by absolute value or increment.When drill clockwise,B is apointted negative.

- Format of call:G95 P9100 Xx Yy Zz Rr Li Aa Bb Hh ;

  X: coordinate X of circle center (specifying of absolute value or increment value)(#24)

  Y: coordinate Y of circle center (specifying of absolute value or increment value)(#25)

  Z: deepness of hole (#26)

  R: fast approaching point (#18)

  F: rate of cutting feeding (#9)

  I: radius of circle (#4)

  A: angle of first hole (#1)

  B: increment angle(negative for clockwise)(#2)

  H: hole number(#11)

- Program of macro call:

```
O0002;
G90   G92 X0 Y0 Z100,0;
G65   p9100 X100 Y50.0   R30.0 Z50.0 500 I100.0 A0 B45.0 H5;
M30;
```

- Macro

```
09100;
#3=#4003; store G codes of 03 group
G81   Z#26   R#18 F#9   K0;(note) drill circle
                              note: L0 is useable too
IF[#3 EQ 90] GOTO 1;transfer to N1 in G90 mode
#24=#5001+#24; compute coordinate Y of circle center
#25=#5001+#25; compute coordinate Y of circle center
N1 WHILE[#11 GT O] DO 1; till residual hole is 0
#5=#24+#4*COS[#1]; compute hole site in X axes
#6=#25+#4*SIN[+1]; compute hole site in X axes
G90   X#5 Y#6 ; drill after move to target location
#1=#1+#2; renew angle
#11=#11-1; hole number-1
END 1;
G#3 G80 ; return to original G code
M99
```

8.5.2 MODE CALL(G66)

Once G66 is send out, specify mode call. That is to call macroafter block specifyed to move along moving axes. G97 is to cancel mode call.

Illustration:

- call

  1. Use the mode call program number specifyed by address P after G66.

  2. When repeat is asked, specify the number of replication from 1 to 9999after address L.

  3. It's same with modeless call(G65) that data specifyed by independent variable is transferred to macro body.

- Cancel Latter block won't execute mode call of macrowhen G97 code is specifyed.
- Nesting call: Call can be nested for 4 levels , and so dose modeless call（G95）and mode call (G66). Subprogram call(M98) is not.

  Limitation:

  1. Multi-macro cann't be called in G66 block.
  2. G66 must be specifyed before independent variable.
  3. Macro can not be called in some segements such as those only having auxiliary function but no moving order
  4. Local variable (independent variable) is only specifyed in G66.Note: Local variable will not be setted every time mode call is executed.

- Typical program Weave operation of fixed cycle G81 by macro.Processing program uses mode call. All datas of drill specifyed by absolute value are used for facilitating program.
- Format of call G65 P9110 Xx Yy Zz Rr Ff Ll;

  X: coordinate X of hole (specifyed by absolute value)(#24)

  Y: coordinate Y of hole (specifyed by absolute value)(#25)

  Z: coordinate Z (specifyed by absolute value) (#26)

  R: coordinate R (specifyed by absolute value) (#18)

  F: rate of feeding (#9)

  L: number of replication

- Program of macro call

```
00001;
G28 G91 X0 Y0 ZO;
G92 X0  Y0 Z50.0;
G00 G90 X100.0 Y50.0;
G66  P9110  Z-20.0 R.0 F500;
G90 X20.0 Y20.0;
X50.0;
X0.0 Y80.0;
G67;
M30;
```

- Macro(called program)

```
09110;
 #1=#4001; store G00/G01
 #2=#4003; store G90/G91
```

```
    #3=#4109; store rate of cutting feeding
    #5=#5003; store coordinate Z of drill starting
    G00 G90   Z#18; fix at point R
    G01   Z#26   F#9 ; cutting feed to point Z
    IF[#4010 EQ 98]GOTO1;return to point 1
    G00 Z#18 ; fix at point R
    GOTO 2;
N1   G00   Z#5 ;  fix at point 1
N2   G#1   G#3   F#4; renew mode info.
M99;
```

8.5.3 MACRO CALL BY G CODE

Set G code of macro call in parameter, and call macro using the same method with modeless call(G65).

Illustration: Set G code of custom macro(09010 to 09019) call in parameter (NO.6050 to NO.6059),and the call mede is the same with G65.For example: Parameter setting.It make G81 call 09010,and call processing circle programmed by custom macro without reworking processing program.

- Corresponding relation between program number parameter number

program number	parameter number
09010	6050
09011	6051
09012	6052
09013	6053
09014	6054
09015	6055
09016	6056
09017	6057
09018	6058
09019	6059

- Repeat   Address L can specify the repeat times from 1 to 9999.
- Independent variable specifying   Two kinds of independent variable specifying are all effective: Independent variable specifying I&II. Specified type of independent variable is setted according to the used address automatically.
- Nesting of macro call using G code   Multi-macro can not be called by just one G code in the program called by G code. G code in this program is processed to normal G code.Multi-macros are not called by just one G code in program in which M or T code regarded as subprogram is called.G code is processed to normal G code in this program.

8.5.4 MACRO CALL BY M CODE

Use the same method with modeless call(G65) to set M code for call macro in setup of parameter.

Illustration: Set M(from 1 to 99999999)code of custom macro(from 09021 to 09029) in parameter(NO.6080 to NO.6089).Use the same way with G65 to call custom macro.

- Relationship between program number and parameter number

program number	parameter number
09020	6080
0902	6081
09022	6082
0902	6083
09024	6084
09025	6085
09026	6086
09027	6087
09028	6088
09029	6089

- Repeat Address L can specify the repeat times from 1 to 9999.
- Independent variable specifying Two kinds of independent variable specifying are all effective: Independent variable specifying I&II. Specified type of independent variable is setted according to the used address automatically.
- Limitation: 1.M code call macro must be specifyed at the front of program segement.

  2. Multi-macros cann't be called by just one G code in macro called by G code or in subprogram called by M or T code. M code in this program or in this macro is processed to normal M code.

8.5.5 SUBPROGRAM CALL BY M CODE

Use the same method with subprogram call(M98) to set M code for call subprogram(macro) in setup of parameter.

Illustration: Set M code(from 1 to 99999999) call subprogram in parameter (NO.6071 to NO.6079),and use the same way with M98 to call corresponding custom macro(from 09001 to 09009).

- Relationship between program number and parameter number

program number	parameter number

09001	6071
09002	6072
09003	6073
09004	6074
09005	6075
09006	6076
09007	6077
09008	6078
09009	6079

- Repeat Address L can specify the repeat times from 1 to 9999.
- Independent variable specifying Independent variable specifying is forbided.
- M code M code called in macro is processed to normal M code.
- Limitation: Multi-macros aren’t called by just one G code in macro called by G code or in subprogram called by M or T code.

8.5.6 SUBPROGRAM CALL BY T CODE

Set T code of subprogram called,and call macro every time T code isspecified in processing program.

说明:

- Call: Set the 5th level of parameter NO.6001 TCS=1. When T code is specifyed in processing program, macro 09000 can be called.T code specifyed in processing program is assigned to common variable #149.
- Limitation: Multi-macros can not be called by just one M code in macro called by G code or in subprogram called by M or T code. T code in this program or in this macro is processed to normal T code.

8.5.7 TYPICAL PROGRAM

Use the function for M code call subprogram to call macro for measuring accumulative hours of use of tool.

Condition: 1.Measure accumulative hours of use of each tool from T01 to T05.The tools whose tool number is over T05 won’t be measured.

2.The following variable is used to store tool number and measure time.

#501	accumulative hours of use of tool number 1
#502	accumulative hours of use of tool number 2
#503	accumulative hours of use of tool number 3

#504	accumulative hours of use of tool number 4
#505	accumulative hours of use of tool number5

3. When M03 is specified, start to compute used time,and when M05 is specified stop computing. During circle start light is on, use system variable #3002 to measure this time.And during feeding pause＆single block stop, time is not computed, but time of toochange and workbench change is computed.

Checking

- Parameter setting set 3 in parameter NO.6071,05 in NO.6072.
- Setup of variable value Set 0 from variable #501 to #505.
- Program call macro

```
00001;
T01  M06;
M03;
M05; change#501
T02  M06;
M03;
M05; change#503
T05 M06;
M03;
M05; change#504
T05 M06;
M03;
M05; change#505
M30;
```

- Macro(the called program)

```
09001 (M03); start macro for computing
N01;
IF[#4120 EQ 0]GOTO9; no specifyed tool
IF[#4120 GT 5]GOTO9; over range of tool number
#3002=0; clear calculator
N9  M03; spindle rotate along positive
M99;
09002(M05); stop macro for computing
M01;
IF[#4120 EQ 0]GOTO9; no specifyed tool
IF[#4120 GT 5]GOTO9; over range of tool number
```

#[500+#120]=#3002+#[500+4120]; compute accumulative time

N9 M05; spindle stop

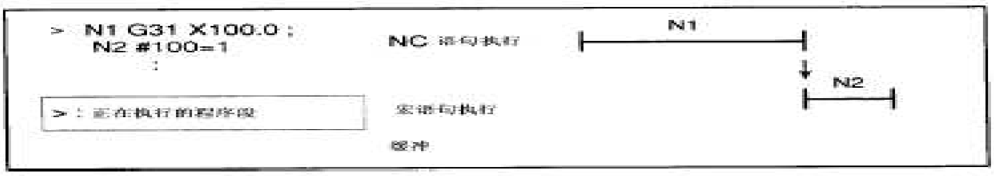
M99;

8.6 PROCESSING OF MACRO STATEMENT

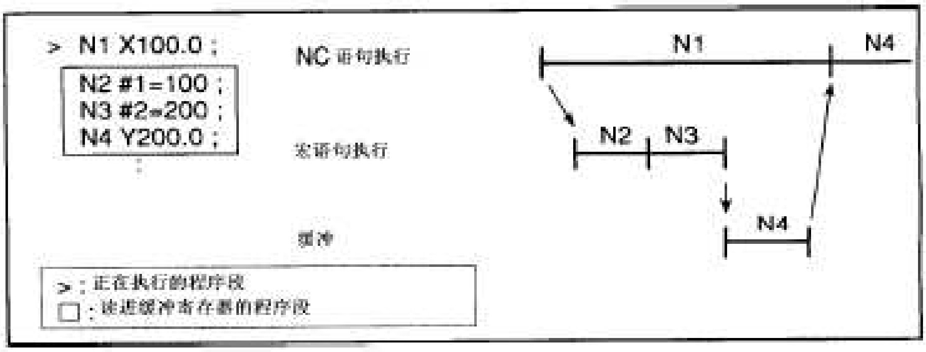
CNC will preread next to be executed NC statement for smoothness maching, and this operation is called buffering.In tool radius compensation mode(G41,G42), NC will preread 2 or 3 program statements for finding crossing point, and arithmetic expression and macro statement of conditional jump will be processed at once after they are read into buffer register.The program statements include M00,M01,M02 or M30;include statements of forbidden buffering setted by prrameter from NO.3411 to NO.3420.

Illustration:

- When the next statement isn't buffered(unbuffered M code,G31 and so on)

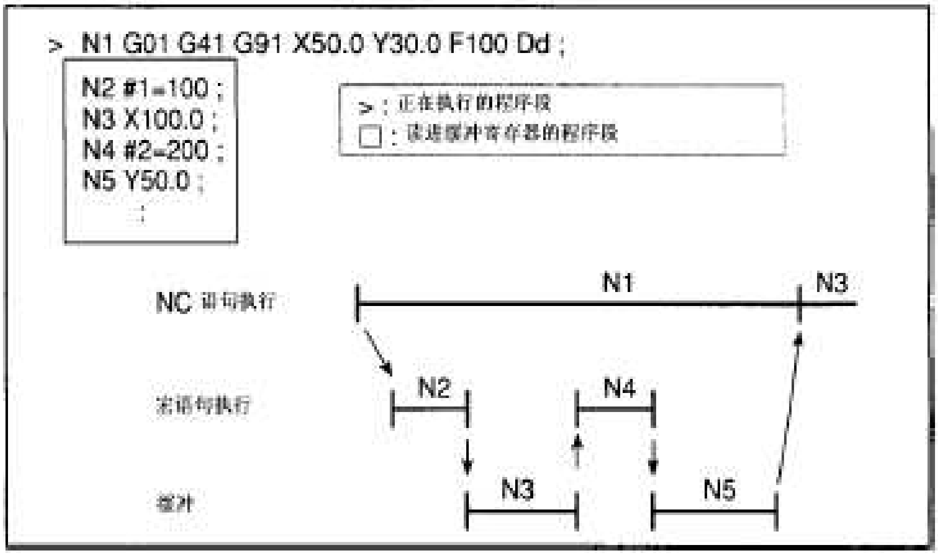


- Buffer next statement in the modes except tool radius compensation mode (G41,G42).



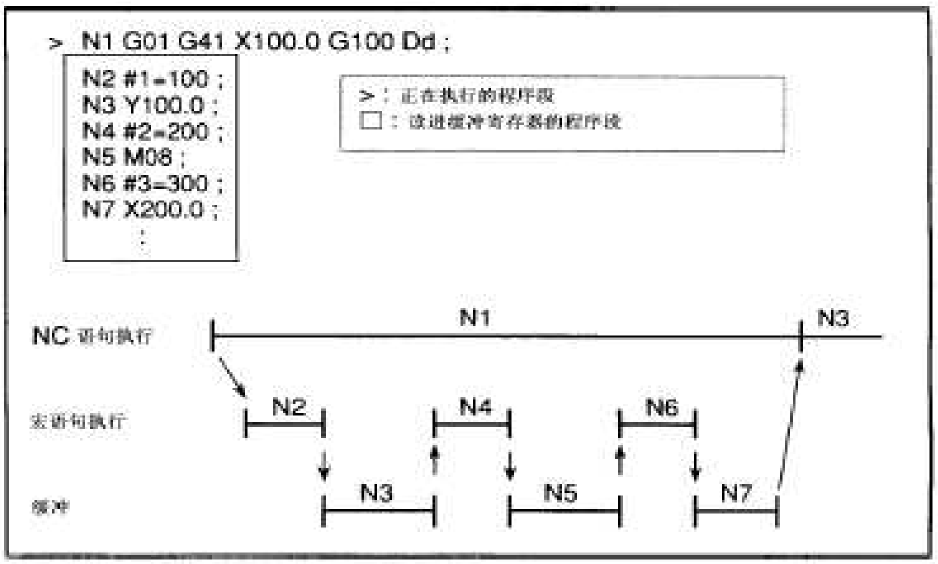
When N1 is executed,next NC statement(N4) is read into buffer.The macros between N1 and N4 are processed during N1 is processed.

- Buffer next statement in tool radius compensation mode (G41,G42).



When N1 is executed, the NC statements in next two statements(till N5) are read into buffer register. The macro statements(N2,N4) between N1 and N5 are processed during N1 is processed.

- When next block includes unmoved segement in tool radius compensation mode C(G41,G42).



When N1 is executed, the NC statements in next two statements(till N5) are read into buffer register.But crossing point can not be computed because N5 is not moving segement.Here, the NC statements in next three statements(till N7) are read in. The macro statements(N2,N4 and N6) between N1 and N7 are processed during N1 is processed.

8.7 STORAGE OF CUSTOM MACRO

Custom macro is similar with subprogram, and it can use the same way to store and programme.The amount of storage is determined by total volume of macro and subprogram.

8.8 LIMITATION

- MDI running   Call command of macro can be specifyed in MDI mode.But during Auto-running,macro call cann't shift to MDI mode.
- Sequence number retrieving

  In single program segement mode, when programisexecuted, the segement can stop.

  In the program segements including macro call command(G65,G66 or G67),it can not stop even in single program segement mode.When set 5^{th} bit of parameter of SBM() NO.6000 1,the segements including arithmetic operation command and control command can stop.

  Single program segement running is used to debug custom macro.Note that,In tool radius compensation mode C,When single program segement stop occur the statement is not regarded as including moving program segement.And in some conditions,right consideration can not be executed.
- Optionalprogram segement jumping   The symble appeared in < expression > isregarded as division operator;not regarded as optional program segement to jump over code.
- Running in EDIT mode   Set parameters NE8 and NEP 1,and custom macro and subprogram numbered from 8000 to 8999 and 8000 to 8999 can be protected.When memorizer is cleared,all data including macro in memorizer are cleared.
- Reset

  When reseted, local variable and public variable from #100 to #149 is cleared to zero.Set CLV and CCV,they are selective to be cleared.System variable from #1000 to #1333 isn't be cleared.

  Reset operation clear any call status and DO status of custom macro and subprogram, and return to main program.
- Display of program restart   The same as M98, the M and T code of called subprogram don't display.
- Feeding pause   during using macro ,when feeding stop is eddective,when reset or alarm,machine stops.
- Constant value used in <espression>   +0.0000001 to +99999999

  -99999999 to -0.0000001

  Effective value is 8bit(algorism),if beyond this range,appears P/S warning NO.003.

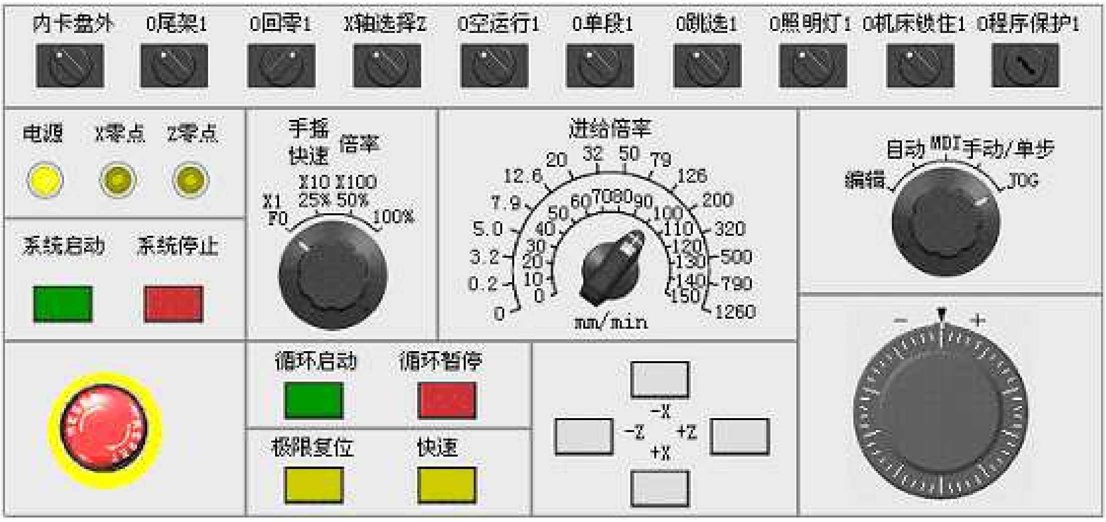
APPENDIX

Nanjing Swan Software Technology Company trys to satisfy consumers.It aimed at so many machines intergrates many panels of different companys and the new Swan panel into NC simulation software. And it is easy to change panel for different panel in this software.

Operational approach：Click to execute Swan software， then select NC lathe or mill.The linking interface shows on Low Right side , then click pulldown list at the corner of Low Right to select panel of required machine.

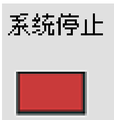
INTRUCTION OF OPERATION PANEL：

1、PANEL OF DALIAN MACHINE

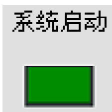


Emergency stop

power/X zero/Z zero display

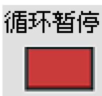


NC system stop

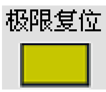


NC system start

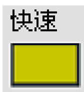
program circle strat



program circle stop

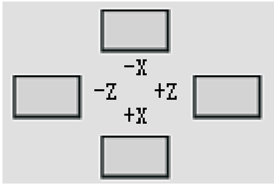


Limit reset

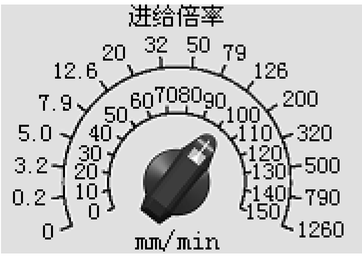


Axes direction quick moving

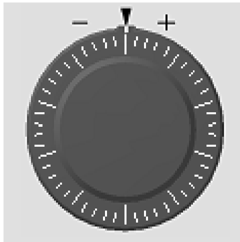
handweel ratio/quickmove speed



Axes direction moving



feeding ratio adjustment

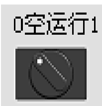


handwheel

mode selection：edit，auto，MDI，handwheel or single step，JOG

orgin returning to 0 selection

axis direction moving selection



dry running selection

single segment selection

skip swith

machine locking swith

program protection switch

2、PANEL OF JINAN MACHINE

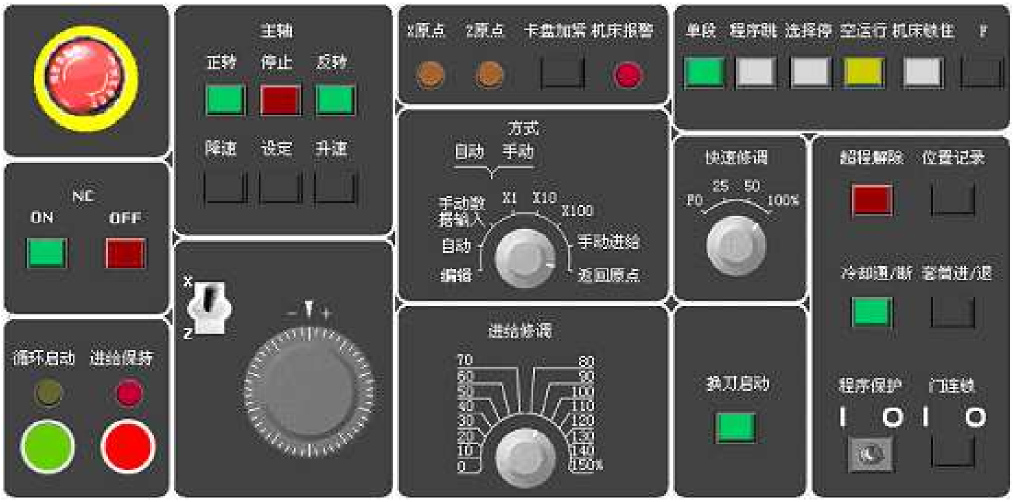
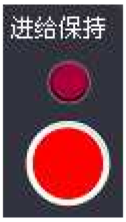


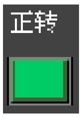
Fig.1

loop start

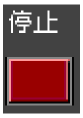


feed keeping

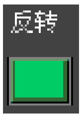

emergent stop



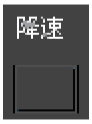
spindle corotation



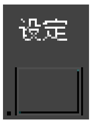
spindle stop



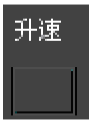
spindle reversion



spindle deceleration

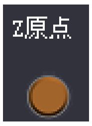


spindle setting

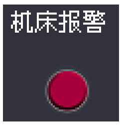


raising speed of spindle

display returningto X orgin

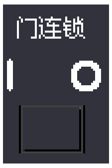


display returningto Z orgin

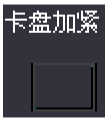


display alarm

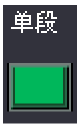
program protection



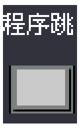
machine gate locking



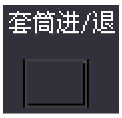
chuck clamping



single segment

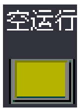


program skip

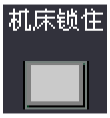


sleeve enter/back

off

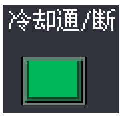


dry run

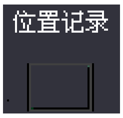


Machine locking

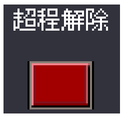
chang-tool run



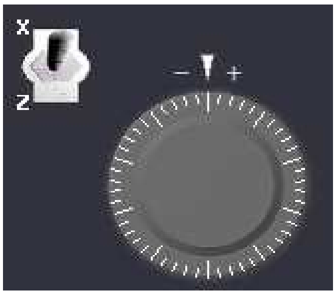
cooling on/off



position record



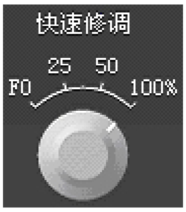
outline unlocked



handwheel control

feeding adjustment

edit/auto/input by hand/feeding ratio/feeding by hand/return to origin



quick adjustment

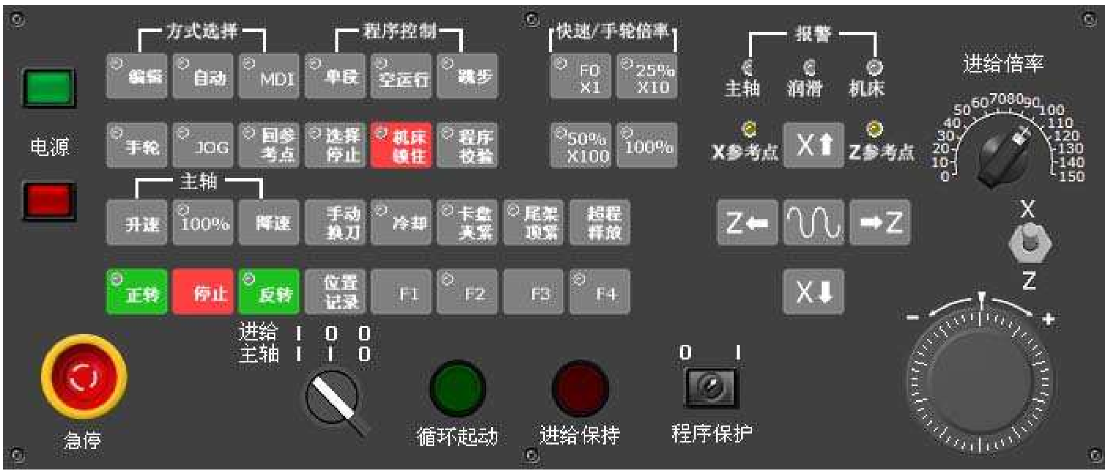


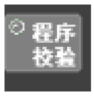
Fig.2

emergency

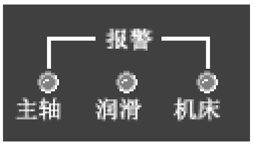

recycle

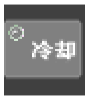

feeding hold

program protection


way selection


program test

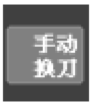
quick/handweel ratio


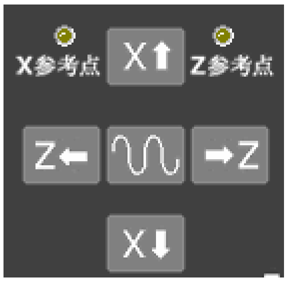
warning program


cooling

way selection

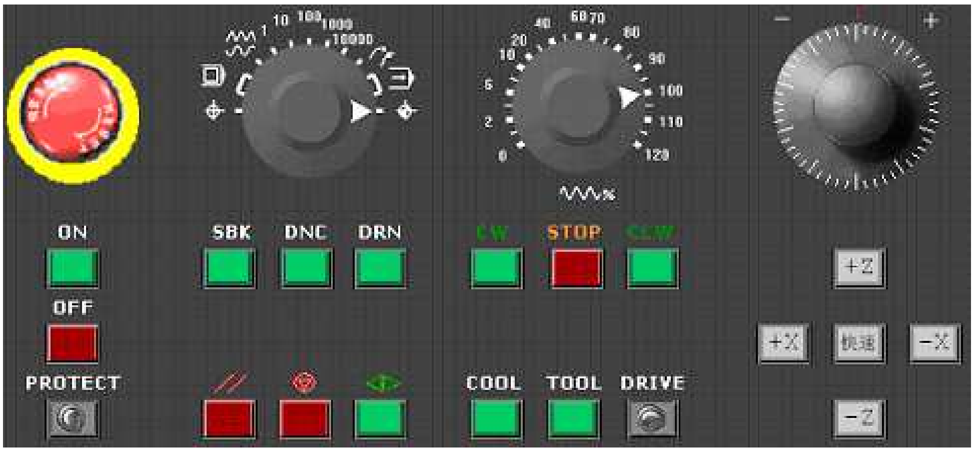

program control

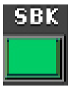
feeding ratio


chang tool by hand


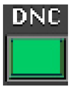
reference point

3、PANEL OF SECOND NANJING MACHINE

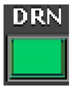




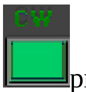
signal run



online operation



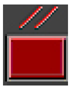
free run

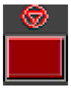


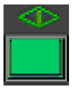
principal axis positive rotate

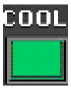
program input protion switch

principal axis negative rotate

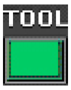

program reset


program stop


program run



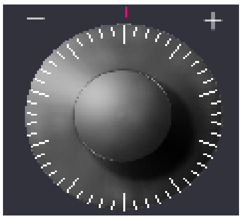
cooling fluid switch



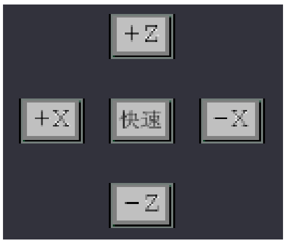
knife rest selectionswitch

machine drive


emergency switch



handwheel

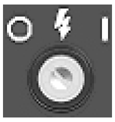


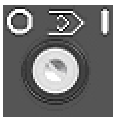
coordinate axis selection/quick feeding

feeding ratio adjustment

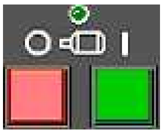
EDIT/MDI/JOG（hand）/MDI feeding/ AUTO（auto recycle）/return to reference point

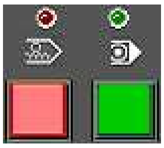
4、PANEL OF NANJING MACHINE

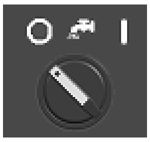

power on/off

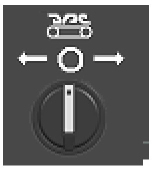

program protection switch

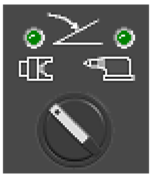
NCsystem power switch

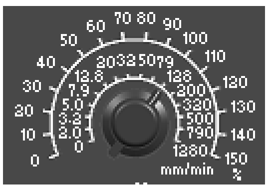

principle axis run/stop


program run/stop

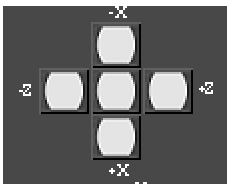

cutting fluid switch


slan transportation switch


chuck act-nip/final act-nip



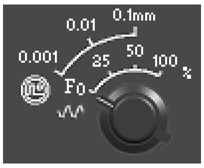
feeding ratio adjustment



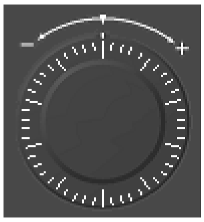
axis move selectionswitch


emergency switch

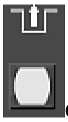
axis selection

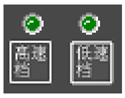


inch-move unit/quick move

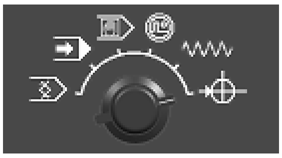

handwheel


signal step

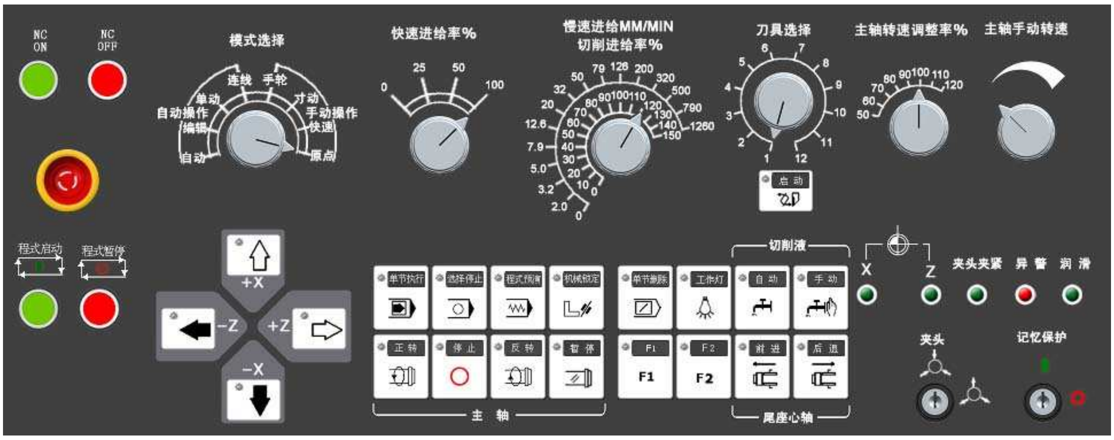

chuck clamp

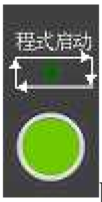


quick/slow display

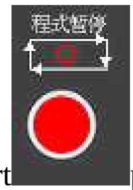

mode selection: edit, recycle, MDA, handwheel, hand, return to origin

5、PANEL OF YOUJIA MACHINE

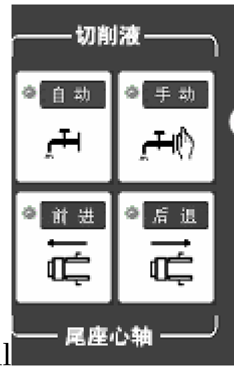




program start



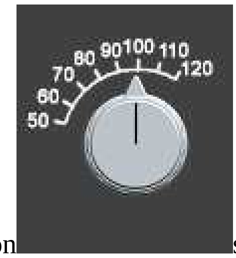
program dwell



mandril of tailstock/cutting solution

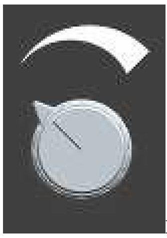
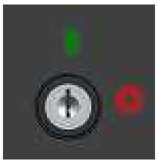
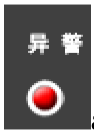
mode selection 

feedrate

tool selection 

speed droop of spindle running


collet


manual speed of spindle 
memory protection 

anomaly warning

6、PANEL OF BAOJI MACHINE

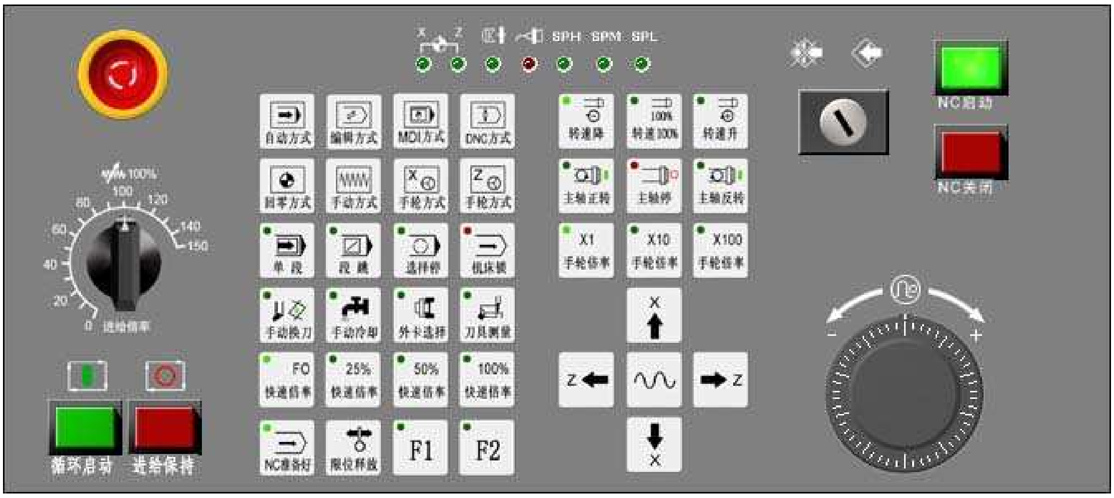
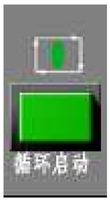
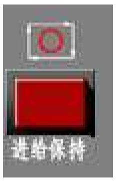


Fig 1

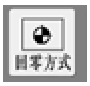
Multiple of feeding speed



Recycle

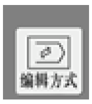


Feeding keeping



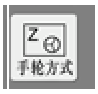
Home mode

Auto mode

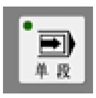


Edit mode

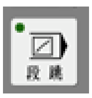
Handwheel mode of X



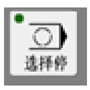
Handwheel mode of Z



Segment run

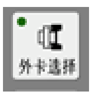


Segment skip



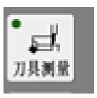
Opt stop

Machine lock



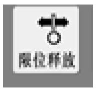
Out card selection

Cooling manually



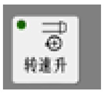
Tool measurement

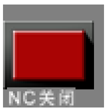
Multiple of fast moving



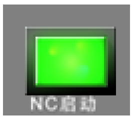
Release limitation of position 

Spindle speed decrease



Spindle speed increase 

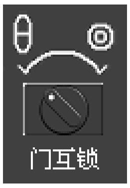
Nc stop 

Spindle normal rotating /stop/abnormal 

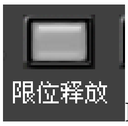
NC start

Mode option 

Emergent stop

Multiple of spindle's speed 

Door locking

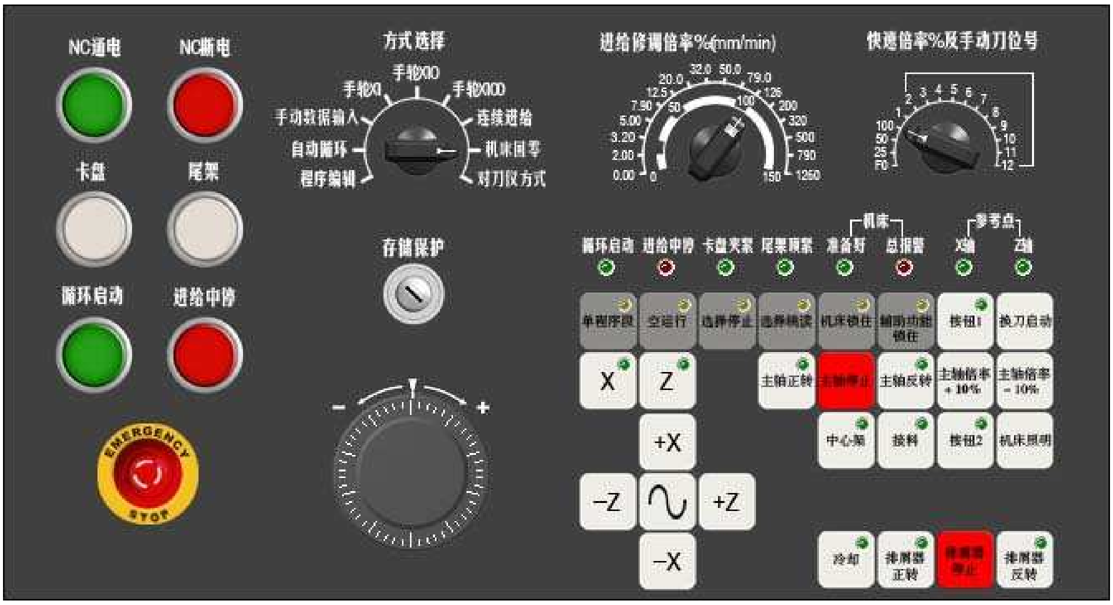


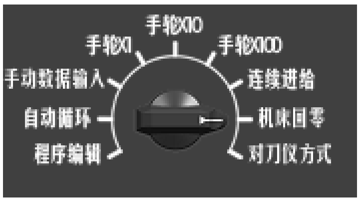
Release position limitation 

Dry run 

Segment skip Swit

7、PANEL OF GREAT WALL MACHINE

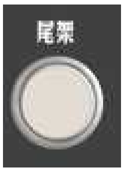




Mode option

Save protection

Chuck

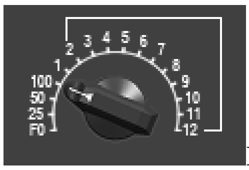


Tail-stock

Recycle

Feeding stop

Multiple of feeding speed adjusting



Multiple of Fast moving and tool number

8、PANEL OF SHENYANG MACHINE

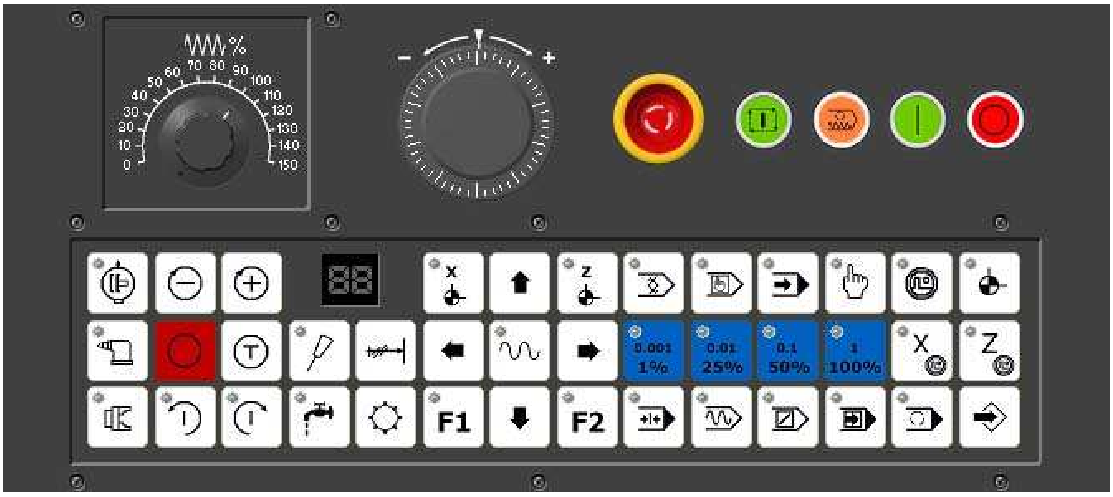


Fig 1

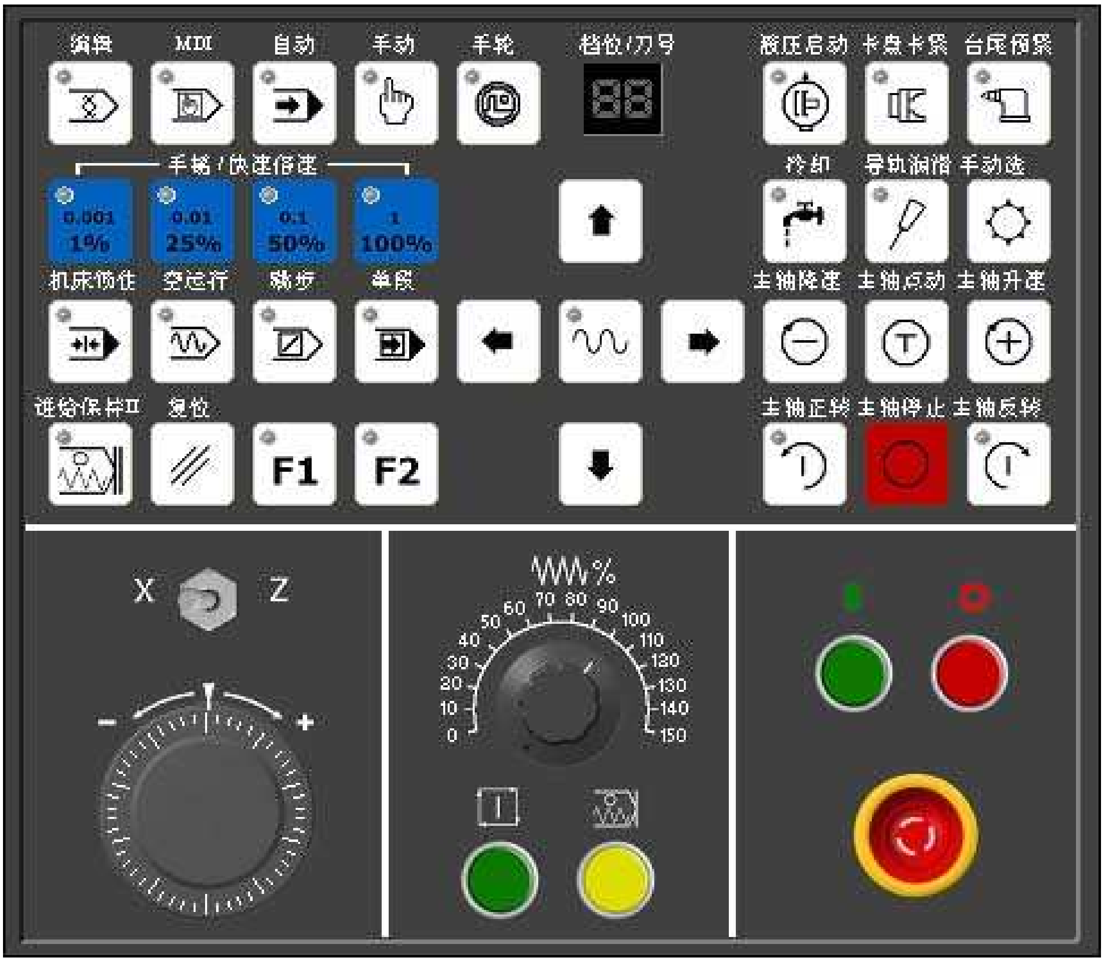


Fig 2

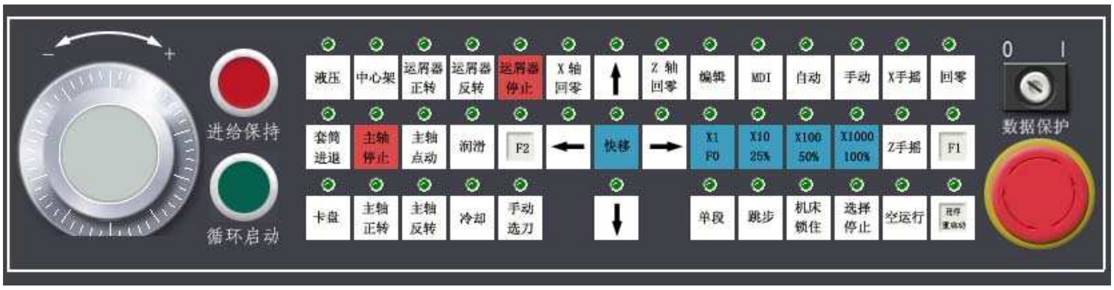
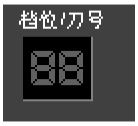


Fig 3

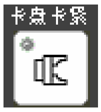
Feeding keeping/Recycle

Data protection



Mode/Tool number

hydraumatic start



Chuck lock

Guideways lubrication

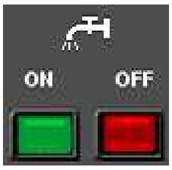
9、PANEL OF YUNNAN MACHINE

Start

Stop

Key of protecting programe inputing

Rotate cutter holder

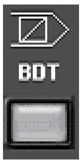


Cooling switch

Vision of ready

Error

Vision of finish



Skip

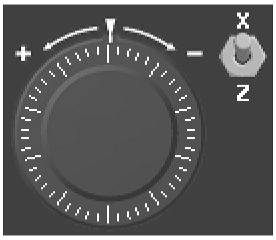
Dry run

Step working

Spindle rotates clockwise

Stop spindle rotating

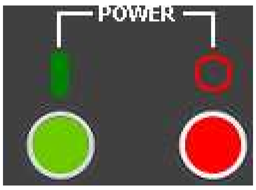
Spindle rotates anticlockwise



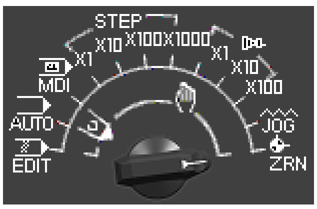
Handwheel

OVR（Speed of feeding adjusting）

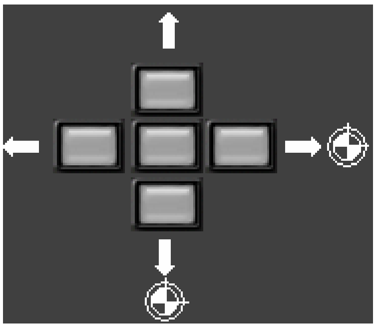

Emergent stop



Power on/off

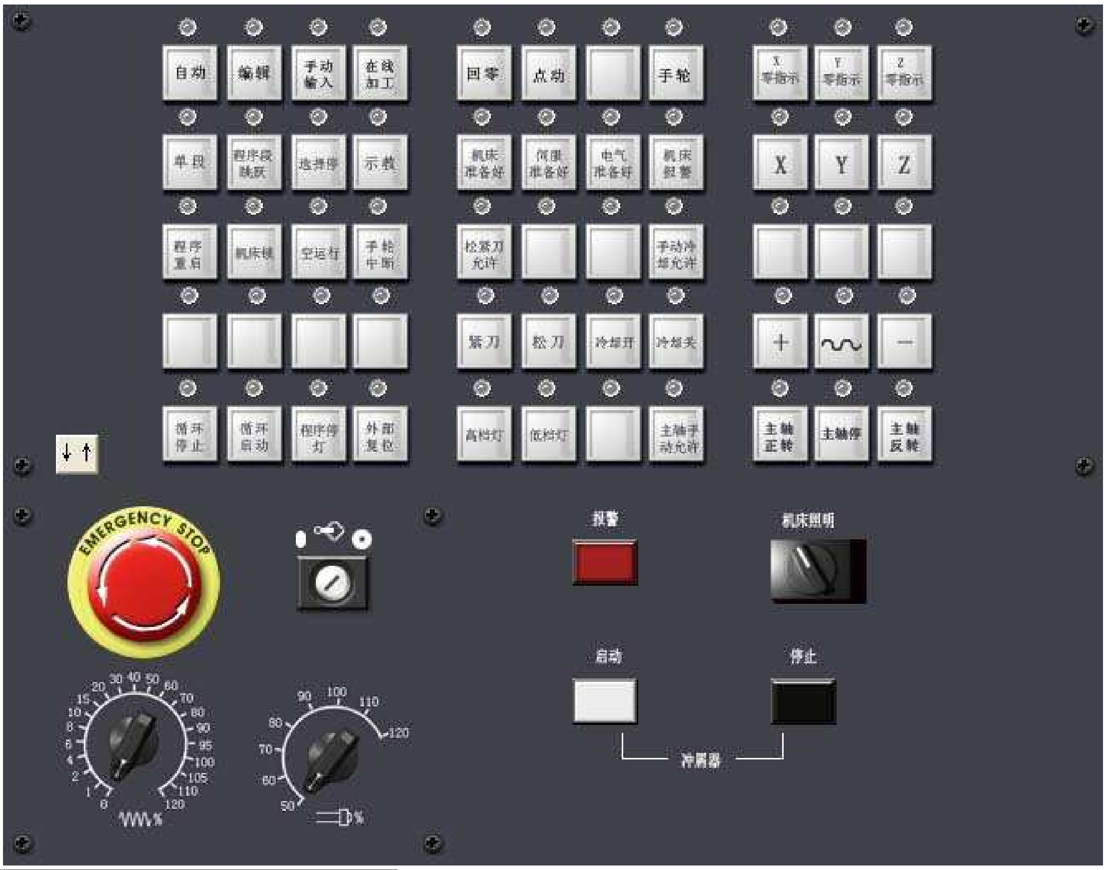


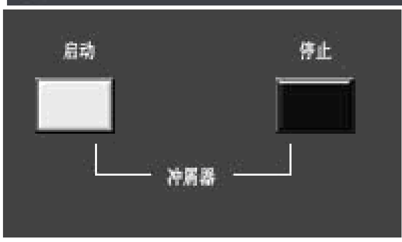
EDIT/AUTO（Auto run）/MDI/STEP（Feeding）/ Multiple of handwheel adjusting/JOG/ZRN（Back reference pion



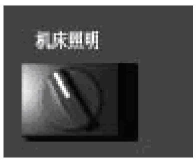
Axis selection, Switch for back reference point

10、PANEL OF BEIJING MACHINE

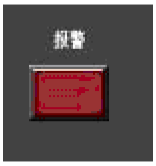




Start/Stop chips washing



Light switch



Alarm

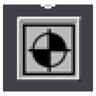
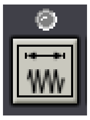
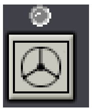
Button for moving table manually

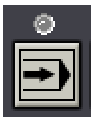
Knop for adjusting feeding speed

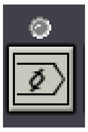
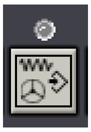
Knop for adjusting speed of spindle

Switch and Edit programe

11、PANEL OF TOP MACHINE

 Back to reference point  Feeding  Move table or tool manually

 Auto mode  Connect pc with cable 232

 Input programe or edit programe on panel  Teaching

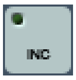
12、PANEL OF NANJING SHUANMAI MACHINEn

Auto mode


Input programe or edit programe on panel

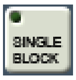
Manual Date Input


JOG mode，move table and tool manually.



Feeding


Connect pc and numerical control machine with cable 232 and then opt input programe for process.



Single block run


Opt stop

Home


Dry run

Cycle start


Stop cycle start

Stop programe

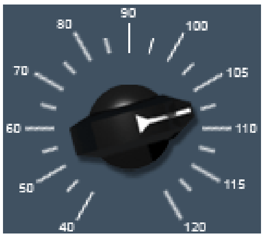

Spindle rotates clockwise

Stop spindle


Spindle rotates anticlockwise

Slect step length: ×1 is 0.001mm, ×10 is 0.01mm, ×100 is 0.1mm, ×100 is 1mm. Put cursor on the goal then press left key of mouse.

Adjust driving speed in programe, rang from 0 to 120%. Put cursor on the knob then press left key of mouse and rotate.

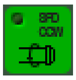


Adjust spindle rotating speed, rang from 0 to 120%.

13、PANEL OF DALIAN MACHINE

 Step run  Press this button, every axis will rotate constantly.

 Opt stop  Dry run

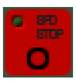
 Spindle rotates clockwise  Spindle rotates anticlockwise

 Cooling  Cooling

 Dry run  Home start

 Work light  M30

 Machine lock  Programe restart

 Rapid  Stop spindle

 Teach  Z axis cancel

EDIT/AUTO（Cycle run）/MDI/ JOG/INC(Increment feeding)

Adjust driving speed in programe, rang from 0 to 120%. Put cursor on the knob then press left key of mouse and rotate.

Adjust spindle rotating speed, rang from 0 to 120%.

Address: Junlin Guoji Building, 5 Guangzhou Road,Suite A 1306, Nanjing, Jiangsu
210008 CHINA
Phone : 086-025-51860015
Fax : 086-025-51860015
Http : www.swansc.com
E-mail: sales@swansc.com
Najing Swan Software Technology Company