NANJING SWANSOFT

SWAN NC SIMULATION SOFTWARE

FANUC SYSTEM
INSTRUCTION OF OPERATION AND PROGRAMMING

Nanjing Swan Software Technology Co., Ltd.
Version 07/2006
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 standard design procedure 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 SW AN NC SIMULATION SOFTWARE ...................................................... 1
  1.1 BRIEF INTRODUCTION OF THE SOFTWARE ........................................................................ 1
  1.2 FUNCTION OF THE SOFTWARE .......................................................................................... 1
    1.2.1 CONTROLLER .............................................................................................................. 1
    1.2.2 FUNCTION INTRODUCTION ....................................................................................... 3

CHAPTER 2  OPERATIONS OF SW AN SC NC SIMULATION SOFTWARE ....................................... 5
  2.1 STARTUP INTERFACE OF THE SOFTWARE ........................................................................ 5
    2.1.1 STARTUP INTERFACE OF PROBATIONAL VERSION .................................................. 5
    2.1.2 STARTUP INTERFACE OF NETWORK VERSION ......................................................... 5
    2.1.3 SINGLE MACHINE VERSION STARTUP INTERFACE ................................................ 7
  2.2 SETUP OF TOOLBAR AND MENU ..................................................................................... 7
  2.3 FILE MANAGEMENT MENU ................................................................................................ 9
    2.3.1 MACHINE PARAMETER .............................................................................................. 10
    2.3.2 CUTTER MANAGEMENT .............................................................................................. 12
    2.3.3 WORKPIECE PARAMETER AND ACCESSORY ............................................................ 15
    2.3.4 RAPID SIMULATIVE MACHINING .............................................................................. 17
    2.3.5 WORKPIECE MEASUREMENT ..................................................................................... 17
    2.3.6 REC PARAMETER SETUP ............................................................................................ 18
    2.3.7 WARING MESSAGE ...................................................................................................... 18

CHAPTER 3  FANUC 0D OPERATION ............................................................................................. 22
  3.1 FANUC 0D MACHINE PANEL OPERATION ......................................................................... 22
  3.2 FANUC 0D NC SYSTEM OPERATION ............................................................................... 24
    3.2.1 KEYSTROKE INTRODUCTION .................................................................................. 25
    3.2.2 MANUAL OPERATION OF VIRTUAL NC MACHINE .................................................. 33

CHAPTER 4  FANUC 0i OPERATION .............................................................................................. 40
  4.1 FANUC 0i PANEL OPERATION .......................................................................................... 40
  4.2 FANUC 0i NC SYSTEM OPERATION .................................................................................. 43
    4.2.1 BUTTON INTRODUCTION ......................................................................................... 44
    4.2.2 MANUAL OPERATION OF MACHINE ...................................................................... 46

CHAPTER 5  FANUC 18i OPERATION ............................................................................................ 54
  5.1 FANUC 18i PANEL OPERATION ......................................................................................... 54
  5.2 FANUC 18i NC SYSTEM OPERATION ................................................................................. 57
    5.2.1 BUTTON INTRODUCTION ....................................................................................... 57
    5.2.2 MANUAL OPERATION OF MACHINE ...................................................................... 59
  5.3 AUXILIARY FUNCTION (M FUNCTION) ............................................................................. 66
  5.4 EXAMPLES ....................................................................................................................... 67
5. PANEL OF YOUJIA MACHINE ................................................................. 153
6. PANEL OF BAOJI MACHINE .............................................................. 155
7. PANEL OF GREAT WALL MACHINE .................................................. 157
8. PANEL OF SHENYANG MACHINE ..................................................... 158
9. PANEL OF YUNNAN MACHINE .......................................................... 159
10. PANEL OF BEIJING MACHINE ......................................................... 161
11. PANEL OF TOP MACHINE .............................................................. 162
12. PANEL OF NANJING SHUANMAI MACHINE ................................ 163
13. PANEL OF DALIAN MACHINE .......................................................... 164
CHAPTER 1  SUMMARY OF SWAN NC SIMULATION SOFTWARE

1.1 BRIEF INTRODUCTION 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 further divided into 8 major types, 28 systems 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. Meanwhile 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 CONTROLLER

1. The screen configurations 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 workpiece directly and confirm that according to special hint on the bottom-left of window.
1.2.2 FUNCTION 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 SSCNCSRVE.exe to login the main interface of SERVER, as the following Fig. show:

![Fig. 2.1-3](image)

(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](image)

(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 click SAVE.

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

2.1.3 SINGLE MACHINE VERSION STARTUP INTERFACE

Fig. 2.1-5

(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 status bar.

Brief introduction of toolbar:
Setup new NC file  
Open saved file(such as NC file)  
Save file(such as NC file)  
Save as  
Machine parametr  
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 switch  
Workpiece measurement  
Voice controller  
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) call in 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 display 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.

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.
**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 tool change rate.

---

Fig.2.3-2

![Project file save](image)

Fig.2.3-3

![Machine parameter setup](image)
Click “Color Choose” to change background color of machine.

Adjust “Processing Drawing Display Acceleration” and “Display Precision” to gain appropriate speed of service of simulation software.
b. Display color:
Click “Confirm” after choose feeding route and color of machineing.

2.3.2 CUTTER MANAGEMENT

a. Milling machine
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 chosen.
(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

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, thickness can be defined.
(5). Click “Confirm” to add them to tool management library.

Internal circle tool adding:

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

![Fig. 2.3-10](image)

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

![Fig. 2.3-11](image)

(3). Choose the tool needed in diao “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](image1)

(1) Define the length, width and highness of workblank and its material.
(2) Define origin of workpiece X, Y, Z.
(3) Select changing machining origin, changing workpiece.

b. Lathe

![Fig. 2.3-13](image2)
(1) Define workblank type, length, diameter and its material.
(2) Define fixture.
(3) Choose tailstock.

Choose workholding fixture

![Clamp Setting](image)

Fig. 2.3-14

Workpiece placement

![Place workpiece](image)

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.
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..

![Diagram of Dimension and...]

**2.3.6 REC PARAMETER SETUP**

Three modes of REC area selection, setup as

![Diagram of Record Parameters]

**2.3.7 WARING MESSAGE**

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

When click “Parameter setup”, window “Info window parameter” will be appearance.
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 leading-in 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 following graph show. The panel composed with Choosing button, Program Running Control Switch and so on is used to control the running status of machine.

![Plane view of the FANUC 0-D machine panel](image1)

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

![Plane view of the FANUC 0-TD lathe machine panel](image2)

Fig. 3.1—2 FANUC 0-TD(lathe)panel

Put cursor on half, 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 SPINDLE**

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: \( \times 1-0.001\text{mm}, \times 10-0.01\text{mm}, \times 100-0.1\text{mm}, \times 1K-1\text{mm} \). Put cursor on the knob and click mouse left key to choose.

**FEED-RATE (F) ADJUSTING KNOB**

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

**SPINDLE 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 doesn’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:
3.2.1 KEYSTROKE INTRODUCTION

Number/Letter key

Number/letter key is used to input data into input region (as the following graph shows), and system will distinguish which to adopt, letter or number by itself.
Input sequence of key: K→J→I→K•••for loop.

**EDIT KEY**

- **ALTER**
  - Replace key. The data inputed replace the data cursor pointing.

- **DELETE**
  - Delete key. Delete the data cursor pointing. Or delete a NC program or all the programs.

- **INSERT**
  - Insert key. Insert the area behind cursor with data which is in the input region.

- **CANCEL**
  - 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.

- **MNU/OFF**
  - 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) Adjust travelling speed.

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


(1) Put mode knob at “MPG”;
(2) Put cursor on “Hand Wheel”, then hold pressing mouse for rotating. Loosen for stopping 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”.

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”
**DELETE ALL NC PROGRAMS**

Put choosing mode at EDIT

Press **DELETE** to delete all NC programs

Press **7** to key in letter “O”

Key in “9999”

Press **DELETE** to delete all NC programs

**SEARCH A SPECIFIED CODE**

A specified 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 **PRGM**

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 **PRGM**

Input NC program name edited, such as “07”, and press **INSERT** 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 specified code.

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

**DELETE, INSERT, REPLACE**

Press **DELETE** to delete the data cursor specified.
Press \textbf{INSERT} to insert the area behind the code specified by cursor with the data in input region.

Press \textbf{ALTER} to replace the code specified by cursor with the data in input region.

**INPUT NC CODE BY HAND WITH CONTROL BOX OPERATION PANEL**

Put mode switch at \textbf{EDIT}

Press \textbf{PRGRM} to login program page.

Press \textbf{7} to input “O7”-program number

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

Press \textbf{INSERT} to start inputing.

Just one code can be inputted 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

**INPUT A NC PROGRAM FROM COMPUTER**

Set mode at \textbf{DNC}

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

Press \textbf{PRGRM} to switch to \textbf{PROGRAM} page.

Input program numbering “Oxxxx”

Press \textbf{INPUT} to read in NC code.

**INPUT ORIGIN PARAMETER OF COMPONENT**

Put switch at \textbf{EDIT} or \textbf{AUTO}

Press \textbf{MENU} to login parameter setup page, and then press “Workpiece”

Switch between No1~No3 and No4~No6 coordinate system page with \textbf{PAGE}

and \textbf{PAGE} and No1~No6 and G54~G59 are one to one correspondence.
Choose coordinate system with CURSOR and .

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

Press to input the data in input region into the specified place.

**INPUT CUTTER COMPENSATION PARAMETER**

Put mode switch at EDIT or AUTO

Press to login parameter setup page, and then press “Redress”

Select length compensation and radius compensation with and .
Choose compensating parameter numbering with CURSOR and.
Input compensation value to length compensation H or radius compensation D.

**POSITION DISPLAY**

Press 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 to return to reference point.

**2. MOVE**

**THERE ARE FOUR METHODS FOR MANUAL MOVING:**
Method 1: Continuously moving (rävss). 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 (rävss). It is for microadjustment, such as preseting reference point.
Set mode at “JOG”.

Method 3: Increment feeding (rävss). Set mode at “JOGINC”.
(1) Choose multiplying power: ×1-0.001mm, ×10 -0.01mm, ×100-0.1mm, ×1K-1mm.
(2) Choose axes. One step per pressing.

Method 4: “Hand Pulse” using (rävss). It is for microadjustment. It’s easy for operator 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 number “7”. Search program numbered “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 right 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”.

35
(4) Press DELETE 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 DELETE to delete all programs

10. SEARCH A SPECIFIED CODE

A specified code can be: a letter or 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

(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

(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 searching a specified 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 DELETE to delete code pointed by cursor

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

Press ALTER to replace code specified 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 EIB to login program page.

(3) Press #7, 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 EOB to open NC file under the list of computer, and the program displays on current screen.

14. INPUT COMPONENT ORIGIN PARAMETER

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

Press MENU 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.
(2) Select coordinate by CURSOR: \[\ downarrow \] and \[\ uparrow \].

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

(3) Press \[\ text{INPUT} \] to input the data in input region to specified place.

15. INPUT CUTTER COMPENSATION PARAMETER
INPUT RADIUS COMPENSATION PARAMETER:

(1) Put mode switch at “JOG”

(2) Press \[\ text{MENU} \] to login parameter setup page, and press “Redress”.

(3) Select length compensation, radius compensation by PAGE: \[\ downarrow \] and \[\ uparrow \].

(4) Select compensation parameter numbering by CURSOR: \[\ downarrow \] and \[\ uparrow \].
(5) Input compensation value to length compensation H or radius compensation D.

(6) Press 🗣️ to input the inputted compensation value to specified 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.

![Coordinate Display](image)

**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 "MDI".

(2) Press 🎥, and press 📠. Input 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:

- **AUTO**: Auto-processing mode.
- **EDIT**: It is used to input NC program and edit code through operation panel directly.
- **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.

Manual start of spindles for corotation.

Manual start of spindles for reversal.

Manual stop of spindles

Manual moving of machine mesa

Milling machine button

Lathe button

Button for single step feeding magnification choosing
When choose mobile axes of machine, the distance of one step is: \( \times 1.001 \text{mm}, \times 10 \text{mm}, \times 100 - 0.1 \text{mm}, \times 1000 - 1 \text{mm} \). Put cursor on button, and then click mouse left key to choose.

Feed rate (F) adjusting knob

![Feed rate adjusting knob](image)

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

Spindle speed adjusting knob

![Spindle speed adjusting knob](image)

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

![Dry running](image)

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

Manual teaching

![Manual teaching](image)

Choose tool in tool library

![Tool](image)

Press it to choose tool.

Locking key of program editing

![Locking key](image)
Put knob at "" to edit and modify program.

Restart program

Program can be started from specified 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

![FANUC 0i (milling machine) panel](image)

Fig. 4.2—1 FANUC 0i (milling machine) panel
4.2.1 BUTTON INTRODUCTION

Number/letter key

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

Press \[ \text{SHIFT} \] to shift input mode, for example: \( O—P, \ 7—A. \)
Edit key

- **ALTERN** Replace key. The data inputed replace the data cursor pointing.
- **DELETE** Delete key. Delete the data cursor pointing; Or delete a NC program or all the programs.
- **INSERT** Insert key. Insert the area behind cursor with data which is in the input region.
- **CANCEL** Cancel key. Remove the data in input region.
- **EOL** 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.
- **OFFSET SET** Parameter input page. First press to login coordinate setup page; second press to login tool compensation page. Press PAGE to shift different page.
- **SYSTEM** System parameter page
- **MESSAGE** Info page. Such “Alarm”.
- **CUSTOM GRAPH** Fig. parameter setup page.
4.2.2 MANUAL OPERATION OF MACHINE
RETURN TO REFERENCE POINT

(1) Put mode knob at JOG.

(2) Choose axeses X Y Z. 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 \[ \text{Rapid moving} \] to make axeses move rapidly.

Method 2: Increment moving . It is for microadjustment, such as preseting reference.
(1) Set mode at \[ \text{Increment moving} \] : 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 for operator 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 spindel get positive and negative rotation. Press to stop spindel.

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:

search 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] to key in number “7”. The searching number keyed in is: “07”

(3) Press [O SRH] and “O7” displays on the screen.

(4) You can input block number “N30”, and “O7” displays on the screen. Press [N SRH] to search block.

DELETE A PROGRAM

(1) Set mode at “EDIT”

(2) Press [PROG] to input letter “O”

(3) Press [7] to input number “7”. Key in the numbering of program to be deleted “O7”.

(4) Press [DELETE] 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 [DELETE] to delete all programs

SEARCH A SPECIFIED CODE

A specified code can be: a letter or 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 [Srte], [BG-EDT], [O SRH], [SRDL], [SRH], [REWIND] to start searching from current programs.

EDIT NC PROGRAM (DELETE, INSERT, REPLACE)

(1) Choose “EDIT” mode
(2) Choose 

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

(4) Move cursor:

Method 1: Press \(\text{PAGE}\) or \(\text{PAGE}\) to turn page, and press \(\text{CURSOR}\): \(\downarrow\) or \(\uparrow\) to move cursor.

Method 2: Use the method searching a specified code to move cursor.

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

(6) Input number of automatically generating block: Press \(\text{SET}\) 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](Image)

Delete, insert, replace:

Press \(\text{DELETE}\) to delete code pointed by cursor

Press \(\text{INSERT}\) to insert the place behind code specified by cursor with data in input region.

Press \(\text{ALTER}\) to replace code specified by cursor with data in input region.

MANUAL INPUT OF NC PROGRAM WITH OPERATION PANEL

(1) Put mode switch at “EDIT” 

49
(2) Press `PROG`, and then press `DIR` to login program page.

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

(4) Press `INSERT` to start input.

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

**INPUT COMPONENT ORGIN PARAMETER**

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

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

Fig. 4.2–7 FANUC Oi-T(lathe)

(2) Select coordinate by PAGE:

CURSOR: ┌───┐
  └───┘

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

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

INPUT CUTTER COMPENSATION PARAMETER

(1) Press OFFSET to login parameter setup page, and press "Redress".

(2) Select length compensation, radius compensation by PAGE: ┌───┐ and └───┘.

Grpag 4.2–8 FANUC Oi-M(milling machine) 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 specified place.

**POSITION DISPLAY**

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

**MDI (MANUAL DATA INPUT)**

(1) Press → to shift to “MDI” mode

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

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

(4) Press → to start program.

**MIRROR IMAGE FUNCTION**

Press → → → → as the graph 4.2-10 shows.

MIRROR IMAGE X, MIRROR IMAGE Y, MIRROR IMAGE Z and mirrorimage functions of X axes, Y axes and Z axes 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.

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 following graph show. The panel composed with Choosing button, 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 SPINDLE**

- Manual starting corotation of spindle.
- Manual starting reversion of spindle.
- Manual stalling of spindle.

**MANUAL MOVING MACHINE PANEL BUTTONS**

**INCREMENT 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 running. Adjustment range: 0～120%. Put cursor on the knob and click mouse left key to choose.

**SPINDLE 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 skipped 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.

**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 executed 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 following graph shows).

Shift lowercase and capital in combination key by \textbf{SHIFT}, such as: \(X - u\), \(Y - v\).

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig5.2_2}
\caption{FANUC 18i-M (milling machine) input of number and symbol}
\end{figure}

\textbf{Edit key}

\textbf{Replace key.} The data inputed replace the data cursor pointing.

\textbf{Delete key.} Delete the data cursor pointing; Or delete a NC program or all the programs.

\textbf{Insert key.} Insert the area behind cursor with data which is in the input region.

\textbf{Cancel key.} Remove the data in input region.

\textbf{Carriage return & line feed key.}

\textbf{Upper case key.}

\textbf{Program display and editing page.}

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

\textbf{Parameter input page.}
5.2.2 MANUAL OPERATION OF MACHINE

THERE ARE THREE METHODS 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 for operator 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 spinddle get positive and negative rotation. Press to stop spinddle.

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 [Drive Icon]

(2) Press [Down Arrow] to call out program after choose a program such as O0001.

(3) Press program starting button [SINGLE STEP RUN]

**SINGLE STEP RUN**

(1) Put single step switch [SBK] at “ON”.

(2) Only one command is executed, every time you press the button [SINGLE STEP RUN] when program is running.

**CHOOSE A PROGRAM**

There are two methods for choosing:

i. Search according to program numbering

(1) Choose “EDIT” mode

(2) Press [PROG] and then press [DIR] to input letter “O”.

(3) Press [3] to input number “3”. The searching number keyed in is: “03”

(4) Press CURSOR: [ ] to start searching. 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 [PROG] to input letter “O”

(2) Press [3] to key in number “3”. The searching number keyed in is: “03”

(3) Press [DIR]
and “O3” displays on the screen.

(4) You can input block number “N30”, and then press \[SRH\] or \[SRM\] to search program block.

**DELETE A PROGRAM**

Set mode at “EDIT”

Press \[PROG\] to input letter “O”

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

Press \[DELETE\] to delete “O3” NC program.

**DELETE ALL PROGRAMS**

Set mode at “EDIT”

Press \[PROG\] to input letter “O”

Input “-9999”

Press \[DELETE\] to delete all programs.

**SEARCH A SPECIFIED CODE**

A specified code can be: a letter or 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 \[PROG\]

Choose a NC program

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

Press \[SRH\]

in \[ABS\] \[REL\] \[ALL\] \[HNDL\] \[G-EDT\] \[O\] \[SRH\] \[SRM\] \[REWND\] \[>\]

start searching from current programs.

**EDIT NC PROGRAM (DELETE、INSERT、REPLACE)**
Choose “EDIT” mode.

Choose input edited NC program name, such as “02”. Press to edit.

Move cursor:

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

Method 2: Use the method searching a specified code to move cursor.

Input data: Click number/letter key by mouse.

is used to delete data in input region.

Input number of automatically generating program block: Press .

“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 to delete code pointed by cursor.

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

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

MANUAL INPUT OF NC PROGRAM WITH OPERATION PANEL

Put mode switch at “EDIT”.

Press , and then press to login program page.

Press , and input “03” program numbering (the numbering keyed in can’t be the same with existing numbering).

Press to start input.

Press 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 to shift to program page.

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

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

INPUT COMPONENT ORIGIN PARAMETER

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

(2) Select coordinate by PAGE: or CURSOR:

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

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

INPUT CUTTER COMPENSATION PARAMETER

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

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

POSITION DISPLAY

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

MDI(MANUAL DATA INPUT)

(1) Press to shift to “MDI” mode.

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

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

(4) Press to start program.

MIRRORIMAGE FUNCTION

Press OFFSET → SETING → [OFF][E] MIRROR IMAGE X, MIRROR IMAGE Y, MIRROR IMAGE Z and mirrorimage functions of X axes, Y axes and Z axes 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.

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 spindle, program stop, open and close of coolant and so on.

<table>
<thead>
<tr>
<th>code</th>
<th>instruction</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Program stop</td>
</tr>
<tr>
<td>M01</td>
<td>Choosing stop</td>
</tr>
<tr>
<td>M02</td>
<td>End of program (Reset)</td>
</tr>
<tr>
<td>M03</td>
<td>Spindle corotation (CW)</td>
</tr>
<tr>
<td>M04</td>
<td>Spindle reversal (CCW)</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle stop</td>
</tr>
<tr>
<td>M06</td>
<td>Too change</td>
</tr>
<tr>
<td>M08</td>
<td>Open coolant</td>
</tr>
<tr>
<td>M09</td>
<td>Close coolant</td>
</tr>
<tr>
<td>M19</td>
<td>Spindle directive stop</td>
</tr>
<tr>
<td>M28</td>
<td>Return to origin</td>
</tr>
<tr>
<td>M30</td>
<td>End of program (Reset) and recur</td>
</tr>
</tbody>
</table>
Table 5.3—1  auxiliary function (M function) list

5.4 EXAMPLES

Example 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

Example 2:

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](image)

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](image)

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

**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](image)

- Arc cutting(G02/G03)

**Format**
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

![Diagram](image)

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. Format

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

![Diagram](image-url)
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_D_Y_
G42 X_D_Y_
```

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.

![Diagram](image)

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**
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>
<thead>
<tr>
<th>Codes</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G40</td>
<td>Cancel the offset of</td>
</tr>
<tr>
<td>G41</td>
<td>Offset at the left of moving direction of the cutter</td>
</tr>
<tr>
<td>G42</td>
<td>Offset at the right of moving direction of the cutter</td>
</tr>
</tbody>
</table>

Table 6.2-1

G43/G44/G49

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

1. Format

\[
\begin{align*}
\text{G43} & \; \text{Z}_- \; \text{H}_- ; \\
\text{G44} & \; \text{Z}_- \; \text{H}_- ; \\
\text{G49} & \; \text{Z}_- ;
\end{align*}
\]

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.

<table>
<thead>
<tr>
<th>Codes</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G43</td>
<td>Make the value of the cutter’s offset add to the value of Z coordinates of the program</td>
</tr>
<tr>
<td>G44</td>
<td>Make the value of the cutter’s offset subtract the value of Z coordinates of the program</td>
</tr>
<tr>
<td>G49</td>
<td>Cancel the offset of the length of the cutter</td>
</tr>
</tbody>
</table>

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 Z0;
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

![Diagram](image)

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.

![Diagram](image)

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

![Diagram](Fig.6.2-8)

3. Example

![Diagram](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 surface of workpiece
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**

   \[
   \text{G74 X__Y__Z__R__ P__F__K__}
   \]

   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. When 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.
**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

\[
\text{G81 X_Y_Z_R_F_K_;}
\]

- \text{X_ Y_: Data of hole site}
- \text{Z_: Depth of the bottom of hole (absolute coordinate)}
- \text{R_: Starting point or raising point per time (absolute coordinate)}
- \text{F_: Feeding rate of cutting}
- \text{K_: Number of replication (if necessary)}

2. Function

![Diagram of G81 (G98) and G81 (G99)](image)

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

\[
\text{G82 X_Y_Z_R_P_F_K_;}
\]
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

G82 (G98) | G82 (G99)
---|---
Initial level | Initial level
Point R | Point R level
Point Z | Point Z

![Fig.6.2-12](image)

G82 drilling cycle, anti-boring cycle

3. Example

![Fig.6.2-13](image)

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

\[
\text{G83 } X_\ Y_\ Z_\ R_\ Q_\ F_\ K_\ :
\]

- \(X_\ Y_\) : 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](image)

G83 Middle feeding and withdraw quickly

3. Example

As Fig.5.2-9 shows

\[
\begin{align*}
\text{N005 G80 G90 G0 X0 Y0 M06 T1} & : \text{change the aiguille of } \varnothing 20 \\
\text{N010 G55} & : \text{call workpiece coordinate of G55} \\
\text{N020 M03 S1000} \\
\text{N030 G43 H1 Z50} \\
\text{N040 G98 G83 Z-30 R1 Q2 F200} & : \text{depth drilling circle, 2MM per time} \\
\text{N050 G80 G0 Z50} & : \text{cancel fixed circle} \\
\text{N060 M05} \\
\text{N070 M30}
\end{align*}
\]

G84

Tapping cycle (G84)

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

1. Format

\[
\text{G84 } X_\ Y_\ Z_\ P_\ F_\ K_\ :
\]
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.

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

3. Example
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

\[
\text{G85 X}_\_\_Y\_\_Z\_R\_F\_K\_;
\]

- \(X\_\_\_\_\_) \text{: Depth of the bottom of hole (absolute coordinate)}
- \(R\_\_\_) \text{: Starting point or raising point per time (absolute coordinate)}
- \(F\_\_\_) \text{: Feeding rate of cutting}
- \(K\_\_\_) \text{: Number of replication (if necessary)}

2. Function

<table>
<thead>
<tr>
<th>G85 (G98)</th>
<th>G85 (G99)</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image.png" alt="Diagram" /></td>
<td><img src="image.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

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_](image)

   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.6.2-18](image)

   G86 The spindle feeding to the bottom of the hole rotates reversely 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 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

\[ \text{G87 X}_Y\text{Z}_R\text{Q}_P\text{F}_K: \]

- **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

![Diagram of G87 and G88]

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

\[ \text{G88 X}_Y\text{Z}_R\text{P}_F\text{K}: \]
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 (G08)   G88 (G99)

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
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

7.1 COORDINATE SYSTEM

Program origin

The origins of program and coordinate system must be set. Generally, program origin is set as the point convenient for programming.

![Coordinate System Diagram](image)

Fig. 7.1-1 Setting origin of coordinate system

Remaining ship distance

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

![Remaining Ship Distance Diagram](image)

Fig. 7.1-2 Example for setting of program origin
Setting workpiece coordinate system

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

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 into one command. The relative coordinate commands required by axis X and axis Z are U and V.
### 7.2 G CODE COMMAND

#### 7.2.1 G CODE SET AND ITS MEANING

<table>
<thead>
<tr>
<th>G Code</th>
<th>Group</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>00</td>
<td>Positioning</td>
</tr>
<tr>
<td>G01</td>
<td>01</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td>01</td>
<td>Circular interpolation/Helical interpolation CW</td>
</tr>
<tr>
<td>G03</td>
<td>02</td>
<td>Circular interpolation/Helical interpolation CCW</td>
</tr>
<tr>
<td>G04</td>
<td>00</td>
<td>Dwell</td>
</tr>
<tr>
<td>G09</td>
<td>00</td>
<td>Exact stop</td>
</tr>
<tr>
<td>G20</td>
<td>06</td>
<td>Input in inch</td>
</tr>
<tr>
<td>G21</td>
<td>06</td>
<td>Input in mm</td>
</tr>
<tr>
<td>G22</td>
<td>04</td>
<td>Stored stroke check function on</td>
</tr>
<tr>
<td>G23</td>
<td>04</td>
<td>Stored stroke check function off</td>
</tr>
<tr>
<td>G27</td>
<td>00</td>
<td>Reference position return check</td>
</tr>
<tr>
<td>G28</td>
<td>00</td>
<td>Return to reference position</td>
</tr>
<tr>
<td>G29</td>
<td>00</td>
<td>Return from reference position</td>
</tr>
<tr>
<td>G30</td>
<td>02</td>
<td>2nd reference position return</td>
</tr>
<tr>
<td>G32</td>
<td>01</td>
<td>Thread cutting</td>
</tr>
<tr>
<td>G40</td>
<td>07</td>
<td>Cutter compensation cancel/Three dimensional compensation cancel</td>
</tr>
<tr>
<td>G41</td>
<td>07</td>
<td>Cutter compensation left/Three dimensional compensation</td>
</tr>
<tr>
<td>G42</td>
<td>07</td>
<td>Cutter compensation right</td>
</tr>
<tr>
<td>G50</td>
<td>00</td>
<td>Scaling cancel</td>
</tr>
<tr>
<td>G52</td>
<td>00</td>
<td>Local coordinate system setting</td>
</tr>
</tbody>
</table>

① absolute coordinate program---X40.Z5.;
② relative coordinate program ---U20.W-40.;
③ mixed coordinate program---X40.W-40.;
### Table 7.2-1 G code set and its meaning

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G53</td>
<td>Machine coordinate system selection</td>
</tr>
<tr>
<td>G70</td>
<td>Finish machining cycle</td>
</tr>
<tr>
<td>G71</td>
<td>Inside and outside diameter rough cutting cycle</td>
</tr>
<tr>
<td>G72</td>
<td>Step rough cutting cycle</td>
</tr>
<tr>
<td>G73</td>
<td>Pattern repeating</td>
</tr>
<tr>
<td>G74</td>
<td>Peck drilling cycle-Z axis</td>
</tr>
<tr>
<td>G75</td>
<td>Grooving in X axis</td>
</tr>
<tr>
<td>G76</td>
<td>Thread cutting cycle</td>
</tr>
<tr>
<td>G80</td>
<td>Canned cycle cancel</td>
</tr>
<tr>
<td>G83</td>
<td>Peck drilling cycle</td>
</tr>
<tr>
<td>G84</td>
<td>Tapping cycle</td>
</tr>
<tr>
<td>G85</td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G87</td>
<td>Back boring cycle</td>
</tr>
<tr>
<td>G88</td>
<td>Back tapping cycle</td>
</tr>
<tr>
<td>G89</td>
<td>Back boring cycle</td>
</tr>
<tr>
<td>G90</td>
<td>Cutting cycle ‘A’</td>
</tr>
<tr>
<td>G92</td>
<td>Thread cutting cycle</td>
</tr>
<tr>
<td>G94</td>
<td>Cutting cycle ‘B’</td>
</tr>
<tr>
<td>G96</td>
<td>Constant surface speed control</td>
</tr>
<tr>
<td>G97</td>
<td>Constant surface speed control cancel</td>
</tr>
<tr>
<td>G98</td>
<td>Feed per minute</td>
</tr>
<tr>
<td>G99</td>
<td>Feed per rotation</td>
</tr>
</tbody>
</table>

(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).
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 command position by linear mode and command.

   \[ G01 \ X(U)_ \ Z(W)_ \ F_ : \]

   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
1. Format

\[
\begin{align*}
&G02(G03) \ X(U) \ Z(W) \ I(K) \ F \ : \\
&G02(G03) \ X(U) \ Z(W) \ R \ F : \\
\end{align*}
\]

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).

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.

G32

➢ Thread cutting(G32)

1. Format

\[
\text{G32 X(U) Z(W) F__ ;}
\]

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

\[
\text{G41 X__ Z__;}
\text{G42 X__ Z__;}
\]
When the edge is a imaginary tool nose, cutting process 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

<table>
<thead>
<tr>
<th>Command</th>
<th>Cutting position</th>
<th>Tool path</th>
</tr>
</thead>
<tbody>
<tr>
<td>G40</td>
<td>Cancel</td>
<td>Tool moves according to program path</td>
</tr>
<tr>
<td>G41</td>
<td>Right</td>
<td>Tool offsets at the left side of path</td>
</tr>
<tr>
<td>G42</td>
<td>Left</td>
<td>Tool offsets at the right side of path</td>
</tr>
</tbody>
</table>

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 surface. 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).

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

   ![Diagram](image)

   **G54 X_ Z_;**

2. Function

At most six workpiece coordinate systems can be setted by using G54～G59. When the power is on and origin is returned, system will select coordinate system automatically (G54). They keep the validity before “modality” change these coordinate.

G70

- Finish turning cycle(G70)

1. Format

   ![Diagram](image)

   **G70 P(ns) Q(nf)**

   ns: First segment number of finish machining shape program
nf: Last segment number of finish machining shape program

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

G71
➢ Excircle rough turning canned cycle (G71)

1. Format

\[
\begin{align*}
\text{N(ns)} & \quad \ldots \ldots \\
\text{F} & \quad \text{Program segments from ns to nf specify moving commands from A to B} \\
\text{S} & \\
\text{T} & \\
\text{N(nf)} & \quad \ldots \ldots \\
\end{align*}
\]

\(\triangle d\): Cutting depth (radius specifying)
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) specifying.

e: Travel of back off
This command is state command. It will not change before a value is specified. FANUC system parameter (NO.0718) specifying.

ns: First segment number of finish machining shape program
nf: Last segment number of finish machining shape program

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

\(\triangle W\): 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 machining shape from A to A' to B is determined by program, and finish machining obligated amount $\Delta u/2$ and $\Delta w$ is obligated in specified area turned by $\Delta d$ (cutting depth).

**G72**

- Face cutting canned cycle (G72)

  ![Face cutting canned cycle](image)

  **Fig. 7.2-11**

1. **Format**

   
   
   
   

   \[
   \text{G72} W (\Delta d) R(e) \\
   \text{G72} P(ns) Q(nf) U(\Delta u) W(\Delta w) F(f) S(s) T(t)
   \]

   The meanings of $\Delta d, e, ns, nf, \Delta u, \Delta w, f, s$ 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)

  ![Contour machining multiple system cycle](image)

  **Fig. 7.2-12**

1. **Format**

   
   
   

   \[
   \text{G73} U(\Delta i) W(\Delta k) R(d) \\
   \text{G73} P(ns) Q(nf) U(\Delta u) W(\Delta w) F(f) S(s) T(t)
   \]
N(ns)........
...........
F__ Program segments from ns to nf specify moving commands from A to B
S__
T__
N(nf)........
\( \Delta i \): Distance of back off in X direction (radius specifying), FANUC system parameter (NO.0719) specifying.
\( \Delta k \): Distance of back off in Z direction (radius specifying), FANUC system parameter (NO.0720) specifying.
d: Times of division
This value is the same with repeat times of machining, FANUC system parameter (NO.0719) specifying.
ns: First segment number of finish machining shape program
nf: Last segment number of finish machining shape program
\( \Delta U \): Distance and direction of finish machining obligated amount in X direction.
\((\text{diameter/radius})\)
\( \Delta W \): 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 machining methods effectively.

G74
➢ Endface pecking drilling cycle(G74)

\[
\begin{align*}
G74 \ R(e); \\
G74 \ X(u) \ Z(w) \ P(\Delta i) \ Q(\Delta k) \ R(\Delta d) \ F(f)
\end{align*}
\]
e: Quantity of back off
This command is state command. It will not change before a value is specified. FANUC system parameter (NO.0722) specifying.

x: X coordinate of B
u: Increment from A to B.
z: Z coordinate of C
w: Increment from A to C
\( \Delta i \): distance of moving in X direction (no symbol)
\( \Delta k \): distance of moving in Z direction (no symbol)
\( \Delta d \): Amount of back off when cutting bottom. Symbol of \( \Delta d \) must be \((+).\) But, if X (U) and \( \Delta I \) 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)

1. Format

\[
\begin{align*}
\text{G75 R(e);} \\
\text{G75 X(u) Z(w) P(\Delta i) Q(\Delta k) R(\Delta d) F(f)}
\end{align*}
\]

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

\[
\begin{align*}
\text{G76 P(m)(r)(a) Q(\Delta d_{\text{min}}) R(d)} \\
\text{G76 X(u) Z(w) R(i) P(k) Q(\Delta d) F(L)}
\end{align*}
\]
m: Number of replication for finish machining (1 to 99)
This command is state command. It will not change before a value is specified. FANUC system parameter (NO.0723) specifying.

r: Quantity of chamfer
This command is state command. It will not change before a value is specified. FANUC system parameter (NO.0109) specifying.

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) specifying. For example: P (02/m, 12/r, 60/a)

$\triangle d_{\text{min}}$: 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) specifying.

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.
$\triangle d$: 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 specify 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 switch to login single block. As Fig shows, 1→2→3→4 cycle. The symbol (+/-) 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 similar with linear cutting cycle.

2. Function

Lathe turning cycle

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

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

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

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

```
G92
```
Thread cutting cycle (G92)

1. Format

Straight thread cutting cycle:

\[
\text{G92 X(U)} \_ \_ \text{Z(W)} \_ \_ \_ \_ \text{F} \_ \_ \_ \_ : \\
\]

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

Taper thread cutting cycle:

\[
\text{G92 X(U)} \_ \_ \text{Z(W)} \_ \_ \_ \_ \text{R} \_ \_ \_ \_ \text{F} \_ \_ \_ \_ : \\
\]

2. Function

Thread cutting cycle

![Diagram of thread cutting cycle](image)
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)RF;

2. Function

Step cutting

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 machining.

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

7.3 AUXILIARY FUNCTION (M FUNCTION)

<table>
<thead>
<tr>
<th>Code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Program stop</td>
</tr>
<tr>
<td>M01</td>
<td>Optional stop</td>
</tr>
<tr>
<td>M02</td>
<td>End of program</td>
</tr>
<tr>
<td>M30</td>
<td>End of program reset</td>
</tr>
<tr>
<td>M03</td>
<td>Spindle corotating</td>
</tr>
<tr>
<td>M04</td>
<td>Spindle reversing</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle stop</td>
</tr>
<tr>
<td>M08</td>
<td>Cutting solution open</td>
</tr>
<tr>
<td>M09</td>
<td>Cutting solution close</td>
</tr>
<tr>
<td>M40</td>
<td>Spindle gear at middle</td>
</tr>
<tr>
<td>M41</td>
<td>Spindle low gear override</td>
</tr>
<tr>
<td>M42</td>
<td>Spindle high gear override</td>
</tr>
<tr>
<td>M68</td>
<td>Hydraulic chuck clamping</td>
</tr>
</tbody>
</table>
7.4 PRESETTING CUTTER OF LATHE

Operation steps:
Firstly, Methods for setting workpiece orgin of FANUC 0-TD \| NC lathe:
1. Test cutting for presetting directly

(1) first use billmpse tool to cut one excircle, measure excircle diameter, press

→ [GECM] 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 → [GECM] 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] key, press [CAN] key to reset “zero”, after measure out round diameter, back off along Z positive direction, select mode, input G01 U.F0.3 cut 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

<table>
<thead>
<tr>
<th>M69</th>
<th>Hydraulic chuck clamping loosening</th>
</tr>
</thead>
<tbody>
<tr>
<td>M78</td>
<td>Tailstock advancing</td>
</tr>
<tr>
<td>M79</td>
<td>Tailstock reversing</td>
</tr>
<tr>
<td>M94</td>
<td>Mirrorimage cancel</td>
</tr>
<tr>
<td>M95</td>
<td>Mirrorimage of X axis</td>
</tr>
<tr>
<td>M98</td>
<td>Calling of subprogram</td>
</tr>
<tr>
<td>M99</td>
<td>End of subprogram</td>
</tr>
</tbody>
</table>

Table 7.3-1  Auxiliary function (M function)
(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 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 , then press , 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 uses such as: G54 X50 Z50……

(3) Attention: use G53 delect G54～G59 workpiece coordinate.

Second. Methods of FANUC 0iT set zero point:

Operation step:

1. Test cutting for presetting directly

(1) Use billmpse tool to test cut a excircle, then measure the diameter, press , input "out round diameter value", press key, cutting tool "X" repair value is inputted into geometry shape automatically.

(2) Use billmpse tool to test cut excircle endface, then press , input "Z 0", press , workpiece “Z” repair value is inputted into geometry shape automatically.

2. Use G50 to set workpiece zero point

(1) Use billmpse tool to test cut a section of excircle, press , then press , now “U” is shining. Press to set "zero", test out round, select “MDI” mode, input G01 U-xx(×xis 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 , 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 if selectG55, input X0, Z0, then press program such as:G55 X60 Z50……
(2) Attention: use G53 direction to clear G54~G59 direction.

7.5 EXAMPLE
G90 Inside and outside diameter cutting cycle

T0101 : cutter compensation
M03 S1000
G0 X105 Z5
G90 X90 Z-80 F0.3 : calling Inside and outside diameter cutting cycle for rough turning
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 machining 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 machining cycle  
N190 G0 X200 Z200  
N200 M05  
N210 T0100 ; cutter compensation cancel  
N220 M30  
G74 Endface pecking drilling cycle

![Diagram](image)

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
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

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

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

Fig.7.5-12
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  ; constant 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 specified 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

<table>
<thead>
<tr>
<th>variable number</th>
<th>variable type</th>
<th>function</th>
</tr>
</thead>
<tbody>
<tr>
<td>#0</td>
<td>dummy variable</td>
<td>This variable is always vacant, and no value can be assigned to it</td>
</tr>
<tr>
<td>#1-#33</td>
<td>local variable</td>
<td>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.</td>
</tr>
<tr>
<td>#100-#199</td>
<td>Common variable</td>
<td>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.</td>
</tr>
<tr>
<td>#500-#999</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#1000---</td>
<td>system variable</td>
<td>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.</td>
</tr>
</tbody>
</table>
● Range of variable value
Local variable common variable can have the value 0 or the value:
$-10^{47}$ to $10^{29}$ or $10^{-3}$ to $10^{47}$
P/S warning NO.111 will accr 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 to variable #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 macrovariable 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 always vacant variable, and it is for reading only.
(a) Quotation
When a undefined variable is quoted, the address itself is ignored too.

<table>
<thead>
<tr>
<th>when #1=&lt;vacant&gt;</th>
<th>when #1=0</th>
</tr>
</thead>
<tbody>
<tr>
<td>G90 X100 Y#1</td>
<td>G90 X100 Y#1</td>
</tr>
<tr>
<td>↓</td>
<td>↓</td>
</tr>
<tr>
<td>G90 X100</td>
<td>G90 X100 Y0</td>
</tr>
</tbody>
</table>

(b) Operation
<Vacant> is the same with 0 except the condition that you assign <vacant>.

<table>
<thead>
<tr>
<th>when #1=&lt;empt&gt;</th>
<th>when #1=0</th>
</tr>
</thead>
<tbody>
<tr>
<td>#2 = #1</td>
<td>#2 = #1</td>
</tr>
<tr>
<td>↓</td>
<td>↓</td>
</tr>
<tr>
<td>#2 = &lt; empt &gt;</td>
<td>#2 = 0</td>
</tr>
</tbody>
</table>
operation manual

(c) Conditional expression

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

<table>
<thead>
<tr>
<th>when #1=&lt; vacant &gt;</th>
<th>when #1=0</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1 EQ #0 ↓ true</td>
<td>#1 EQ #0 ↓ error</td>
</tr>
<tr>
<td>#1 NE #0 ↓ true</td>
<td>#1 NE #0 ↓ error</td>
</tr>
<tr>
<td>#1 GE #0 ↓ true</td>
<td>#1 GE #0 ↓ error</td>
</tr>
<tr>
<td>#1 GT #0 ↓ error</td>
<td>#1 GT #0 ↓ error</td>
</tr>
</tbody>
</table>

- 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;
/ #2 G00 X100.0;
N#3 Y200.0;

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.

<table>
<thead>
<tr>
<th>function</th>
<th>format</th>
<th>remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>definition</td>
<td>#i=#j</td>
<td></td>
</tr>
<tr>
<td>Function</td>
<td>Example</td>
<td>Notes</td>
</tr>
<tr>
<td>---------------</td>
<td>------------------</td>
<td>--------------------------------------------</td>
</tr>
<tr>
<td>Sine</td>
<td>#i=SIN[#j];</td>
<td>Angle is expressed with degree. 90°30’ for 90.5 degree</td>
</tr>
<tr>
<td>Antisine</td>
<td>#i=ASIN[#j];</td>
<td></td>
</tr>
<tr>
<td>Cosine</td>
<td>#i=COS[#j];</td>
<td></td>
</tr>
<tr>
<td>Arccosine</td>
<td>#i=ACOS[#j];</td>
<td></td>
</tr>
<tr>
<td>Tangent</td>
<td>#i=TAN[#j];</td>
<td></td>
</tr>
<tr>
<td>Arctangent</td>
<td>#i=ATAN[#j];</td>
<td></td>
</tr>
<tr>
<td>Square root</td>
<td>#i=SQRT[#j];</td>
<td></td>
</tr>
<tr>
<td>Absolute value</td>
<td>#i=ABS[#j];</td>
<td></td>
</tr>
<tr>
<td>Rounding</td>
<td>#i=ROUNND[#j];</td>
<td></td>
</tr>
<tr>
<td>Superior rounding</td>
<td>#i=FIX[#j];</td>
<td></td>
</tr>
<tr>
<td>Inferior rounding</td>
<td>#i=FUP[#j];</td>
<td></td>
</tr>
<tr>
<td>Natural logarithm</td>
<td>#i=LN[#j];</td>
<td></td>
</tr>
<tr>
<td>Exponential function</td>
<td>#i=EXP[#j];</td>
<td></td>
</tr>
<tr>
<td>Or</td>
<td>#i-#jOR#k;</td>
<td>Logical operation is executed by scale of two one bit by one bit</td>
</tr>
<tr>
<td>Exclusive-or</td>
<td>#i-#jXOR#k;</td>
<td></td>
</tr>
<tr>
<td>And</td>
<td>#i-#jAND#k;</td>
<td></td>
</tr>
<tr>
<td>Turn BCD to BIN</td>
<td>#i=BIN[#j];</td>
<td>It is used for handshaking with.</td>
</tr>
<tr>
<td>Turn BIN to BCD</td>
<td>#i=BCD[#j];</td>
<td></td>
</tr>
</tbody>
</table>

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 occur 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 occur 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°~360°
   When the bit of parameter (NO.6004#0) NAT is setted 1°~180°

(3) Variable #j can be replaced with constant.
   • Natural logarithm  #i=LN[#j];
     (1) Attention: relative error may be more than 10^{-8}.
     (2) When antilogarithm (#j) is 0 or less than 0, P/S warning NO.111 will occur
     (3) Variable #j can be replaced with constant.
   • EXP function    #i=EXP[#j]
     (1) Attention: relative error may be more than 10^{-8}.
     (2) Overflow and P/S warning NO.111 will occur 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) command 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 the function.
     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 occur 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 $10 \times 10^{-8}$ 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 specified by standout mode.
8.4.2 CONDITIONAL TRANSFER(IF) STATEMENT

Specify conditional expression behind IF.

IF[<conditional expression >]GOTOn   If specified conditional expression is satisfied, move to the program segment marked sequence number n. If not, execute next program segment.

IF[<conditional expression >]THEN    If conditional expression is satisfied, execute 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 values. Note: symbol NO can’t be used.

<table>
<thead>
<tr>
<th>operator</th>
<th>meanings</th>
</tr>
</thead>
<tbody>
<tr>
<td>EQ</td>
<td>equal</td>
</tr>
<tr>
<td>NE</td>
<td>unequal</td>
</tr>
<tr>
<td>GT</td>
<td>greater than</td>
</tr>
<tr>
<td>GE</td>
<td>Less than or Equal to</td>
</tr>
<tr>
<td>LT</td>
<td>less than</td>
</tr>
<tr>
<td>LE</td>
<td>Less than or Equal to</td>
</tr>
</tbody>
</table>

Classical program:

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

```plaintext
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 is specified after WHILE. When the specified condition is satisfied, execute program from D0 to END. Otherwise goto the block after END.

Illustration:
When the specified condition is satisfied, execute program from D0 to END. Otherwise goto the block after END. This instruction format applies IF location. 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 specified 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 effect 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 compute 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 specified 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 don’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 specified by address P is called, when G65 is specified. And data can be transferred into custom macro body.

Illustration:

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

(2) When repeat is asked, specify the number of replication from 1 to 9999 after 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 except 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, K (I is 1-10). Type specified by independent variable is changed automatically according to the used letters.

Independent variable specifying I

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

(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 specified by alphabetic sequence. And it must accord the format of word address. But I, J and K need to be specified by alphabetic sequence.

Independent variable specifying II

Independent variable specifying II use each A, B, and C just once, while I, J, K 10 times.
Independent variable specifying II is used to transfer variables such as 3D coordinate value.
## Operation Manual

### User Macro Program

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

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

**Limitation:**

- **Format:** G65 must be specified 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 specified is effective.
- **Position of decimal** The unit of argument data without decimal is the least setting unit of each address. Transfer of value of independent variable is based on variation of actual system configuration of machine. Using decimally can improve program compatibility.
- **Nesting call:** Call can be nested for 4 levels, and so does 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 recovered.
- **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 specified by absolute value or increment. When drill clockwise, B is appointed 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 B4 5.0 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 macro after block specified to move along moving axes. G97 is to cancel mode call.

Illustration:
• call

1. Use the mode call program number specified by address P after G66.
2. When repeat is asked, specify the number of replication from 1 to 9999 after address L.
3. It’s same with modeless call(G65) that data specified by independent variable is transferred to macro body.

• Cancel Latter block won’t execute mode call of macro when G97 code is specified.

• Nesting call: Call can be nested for 4 levels, and so does modeless call (G95) and mode call (G66). Subprogram call (M98) is not.

Limitation:

1. Multi-macro can’t be called in G66 block.
2. G66 must be specified before independent variable.
3. Macro can not be called in some segments such as those only having auxiliary function but no moving order.
4. Local variable (independent variable) is only specified 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 specified by absolute value are used for facilitating program.

• Format of call G65 P9110 Xx Yy Zz Rr Ff Ll;

  X: coordinate X of hole (specified by absolute value)(#24)
  Y: coordinate Y of hole (specified by absolute value)(#25)
  Z: coordinate Z (specified by absolute value) (#26)
  R: coordinate R (specified by absolute value) (#18)
  F: rate of feeding (#9)
  L: number of replication

• Program of macro call 00001;

G28 G91 X0 Y0 Z0;
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

138
#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 mode 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.  

<table>
<thead>
<tr>
<th>program number</th>
<th>parameter number</th>
</tr>
</thead>
<tbody>
<tr>
<td>09010</td>
<td>6050</td>
</tr>
<tr>
<td>09011</td>
<td>6051</td>
</tr>
<tr>
<td>09012</td>
<td>6052</td>
</tr>
<tr>
<td>09013</td>
<td>6053</td>
</tr>
<tr>
<td>09014</td>
<td>6054</td>
</tr>
<tr>
<td>09015</td>
<td>6055</td>
</tr>
<tr>
<td>09016</td>
<td>6056</td>
</tr>
<tr>
<td>09017</td>
<td>6057</td>
</tr>
<tr>
<td>09018</td>
<td>6058</td>
</tr>
<tr>
<td>09019</td>
<td>6059</td>
</tr>
</tbody>
</table>

● 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

<table>
<thead>
<tr>
<th>program number</th>
<th>parameter number</th>
</tr>
</thead>
<tbody>
<tr>
<td>09020</td>
<td>6080</td>
</tr>
<tr>
<td>0902</td>
<td>6081</td>
</tr>
<tr>
<td>09022</td>
<td>6082</td>
</tr>
<tr>
<td>0902</td>
<td>6083</td>
</tr>
<tr>
<td>09024</td>
<td>6084</td>
</tr>
<tr>
<td>09025</td>
<td>6085</td>
</tr>
<tr>
<td>09026</td>
<td>6086</td>
</tr>
<tr>
<td>09027</td>
<td>6087</td>
</tr>
<tr>
<td>09028</td>
<td>6088</td>
</tr>
<tr>
<td>09029</td>
<td>6089</td>
</tr>
</tbody>
</table>

● 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 segment.

2. Multi-macros can’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

<table>
<thead>
<tr>
<th>program number</th>
<th>parameter number</th>
</tr>
</thead>
<tbody>
<tr>
<td>09020</td>
<td>6080</td>
</tr>
<tr>
<td>09021</td>
<td>6081</td>
</tr>
<tr>
<td>09022</td>
<td>6082</td>
</tr>
<tr>
<td>09023</td>
<td>6083</td>
</tr>
<tr>
<td>09024</td>
<td>6084</td>
</tr>
<tr>
<td>09025</td>
<td>6085</td>
</tr>
<tr>
<td>09026</td>
<td>6086</td>
</tr>
<tr>
<td>09027</td>
<td>6087</td>
</tr>
<tr>
<td>09028</td>
<td>6088</td>
</tr>
<tr>
<td>09029</td>
<td>6089</td>
</tr>
</tbody>
</table>
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 is specified in processing program.

Call: Set the 5th level of parameter NO.6001 TCS=1. When T code is specified in processing program, macro 09000 can be called. T code specified 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.

<table>
<thead>
<tr>
<th>#501</th>
<th>accumulative hours of use of tool number 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>#502</td>
<td>accumulative hours of use of tool number 2</td>
</tr>
<tr>
<td>#503</td>
<td>accumulative hours of use of tool number 3</td>
</tr>
</tbody>
</table>
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 tool change 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 specified 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 specified 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 machining, 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 set by parameter from NO.3411 to NO.3420.

Illustration:

- When the next statement isn’t buffered (unbuffered M code, G31 and so on)

![Illustration 1](image1)

- Buffer next statement in the modes except tool radius compensation mode (G41, G42).

![Illustration 2](image2)

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 segment 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 segment. 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 specified in MDI mode. But during Auto-running, macro call can’t shift to MDI mode.

- Sequence number retrieving
  In single program segment mode, when program is executed, the segment can stop.

  In the program segments including macro call command (G65, G66 or G67), it can not stop even in single program segment mode. When set 5th bit of parameter of SBM() NO.6001, the segments including arithmetic operation command and control command can stop.

  Single program segment running is used to debug custom macro. Note that, in tool radius compensation mode C, when single program segment stop occur the statement is not regarded as including moving program segment. And in some conditions, right consideration can not be executed.

- Optional program segment jumping
  The symbol appeared in < expression > is regarded as division operator; not regarded as optional program segment to jump over code.

- Running in EDIT mode
  Set parameters NE8 and NEP1, 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 effective, when reset or alarm, machine stops.

- Constant value used in < expression >
  +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 tries to satisfy consumers. It aimed at so many machines integrates many panels of different companies 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.

INSTRUCTION OF OPERATION PANEL:

1、PANEL OF DALIAN MACHINE

<table>
<thead>
<tr>
<th>Operation Panel</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power / X zero / Z zero display</td>
<td>Emergency stop</td>
</tr>
<tr>
<td>System stop</td>
<td>NC system stop</td>
</tr>
<tr>
<td>Program circle start</td>
<td>Limit reset</td>
</tr>
<tr>
<td>Program circle stop</td>
<td>Axes direction quick moving</td>
</tr>
</tbody>
</table>

Operational approach:

- Click to execute Swan software.
- Select NC lathe or mill.
- The linking interface shows on Low Right side.
- Click pulldown list at the corner of Low Right to select panel of required machine.
Machine operation panel

Handwheel ratio/Quickmove speed

Axes direction moving

Feeding ratio adjustment

Handwheel

Mode selection: edit, auto, MDI, handwheel or single step, JOG

 Origin returning to 0 selection

Axis direction moving selection

Dry running selection

Single segment selection

Skip switch

Machine locking switch

Program protection switch
2. PANEL OF JINAN MACHINE

Fig. 1

- **循环启动** (loop start)
- **进给保持** (feed keeping)
- **emergent stop**
- **spindle corotation** (spindle rotation)
- **spindle stop**
- **spindle reversion** (spindle reversal)
- **spindle deceleration**
- **spindle setting**
- **raising speed of spindle**
- **display returning to X orgin**
- **display returning to Z orgin**
- **display alarm**
- **程序保护** (program protection)
- **卡盘加紧** (chuck clamping)
- **单段** (single segment)
- **程序跳** (program skip)
- **门连锁** (machine gate)
- **sleeve enter/back**
Machine operation panel

- Dry run (dry run)
- Machine locking (Machine locking)
- Chang-tool run (chang-tool run)
- Cooling on/off (cooling on/off)
- Position record (position record)
- Outline unlocked (outline unlocked)

Handwheel control (handwheel control)
Feeding adjustment (feeding adjustment)

Edit/auto/input by hand/feeding ratio/feeding by hand/return to origin (edit/auto/input by hand/feeding ratio/feeding by hand/return to origin)

Quick adjustment (quick adjustment)

Fig. 2

149
3. PANEL OF SECOND NANJING MACHINE

Machine operation panel

Operation manual
Machine operation panel

- **SBK** signal run
- **DNC** online operation
- **DRN** free run
- **PROTECT** program input
- **FWD** principal axis positive rotate
- **PROG** program switch
- **REV** principal axis negative rotate
- **principal axis negative** rotate
- **tool** knife rest
- **COOL** cooling fluid switch
- **handle** selectionswitch

- **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
- NC system power switch
- Principle axis run/stop
- Program run/stop
- Cutting fluid switch
- Slant transportation switch
- Chuck act-nip/final act-nip
- Feeding ratio adjustment
- Axis move
- Selection switch
5、PANEL OF YOUJIA MACHINE

- 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
- 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
Machine operation panel

- 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

Fast moving and tool number

Multiple of
8、PANEL OF SHENYANG MACHINE

Fig 1

Fig 2
9. PANEL OF YUNNAN MACHINE
Machine operation panel

- **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 anticlockwise**
- **Handwheel feeding adjusting**
- **OVR (Speed of)**
- **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

Knob for adjusting feeding speed

Knob for adjusting speed of spindle

Switch and Edit program

11. PANEL OF TOP MACHINE
12. PANEL OF NANJING SHUANMAI MACHINE

- **Auto mode**
- **Input program or edit program 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 program for process.**
- **Single block run**
- **Opt stop**
- **Home**
- **Dry run**
- **Cycle start**
- **Stop cycle start**
- **Stop program**
- **Spindle rotates clockwise**
- **Stop spindle**
- **Spindle rotates anticlockwise**

Select step length: \( \times 1 \) is 0.001mm, \( \times 10 \) is 0.01mm, \( \times 100 \) is 0.1mm, \( \times 1000 \) is 1mm. Put cursor on the goal then press left key of mouse.

Adjust driving speed in program, range from 0 to 120%. Put cursor on the knob then press left key of mouse and rotate.
Adjust spindle rotating speed, range 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 program, range from 0 to 120%. Put cursor on the knob then press left key of mouse and rotate.

Adjust spindle rotating speed, range 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