SwanSoft NANJING SWANSOFT

SWAN NC SIMULATION SOFTWARE

MITSUBISHI SYSTEM

INSTRACTION OF OPERATION AND PROGRAMMING

Nanjing Swan Software Technology Co.,Ltd.
Version 05/2007

PREFACE

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

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

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

Nanjing Swan Software Technology Company

05/2007

CONTENTS

CHAPTER 1 SUMMARY OF SWAN NC SIMULATION SOFTWARE

1.1 BRIEF INTOUCTION OF THE SOFTWARE

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

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

1.2 FUNCTION OF THE SOFTWARE

1.2.1 CONTROLER

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

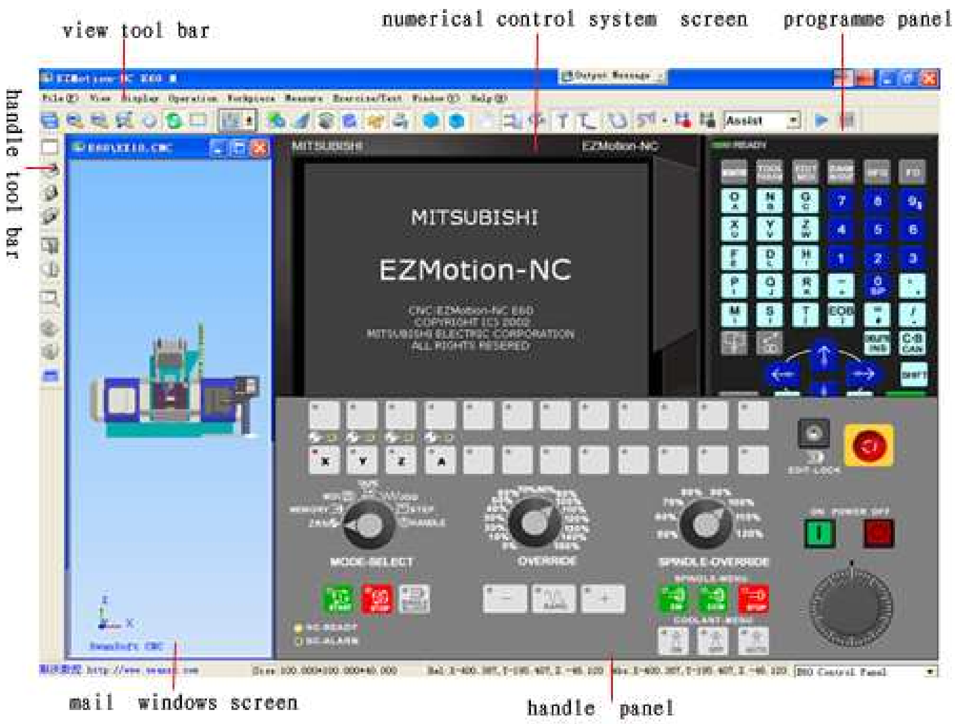


Fig.1.2-1 E60 M(milling machine)

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

（2）Choose reference mandril.

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

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

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

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

Fig.1.2-2 E68 T(lathe)

1.2.2 FUNCTON INTRODUCTION

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

★ Vivid 3DM NC machine and operation panels.

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

★ Support system self-defining code and canned cycle.

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

★ Windows macro record and playback.

★ AVI files record and playback.

★ Placement and mounting of workpiece.

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

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

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

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

★ Adopt data base management tools and performance parameter library.

★ There are many kinds of tools.

★ Support custom-defined tool function.

★ 3DM measurement function of processed model.

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

CHAPTER 2 OPERATIONS OF SWANSC NC SIMULATION SOFTWARE

2.1 STARTUP INTERFACE OF THE SOFTWARE

2.1.1 STARTUP INTERFACE OF PROBATIONAL VERSION

Fig. 2.1-1

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

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

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

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

2.1.2 STARTUP INTERFACE OF NETWORK VERSION

Fig. 2.1-2

（1）Choose NETWORK in the left document frame.

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

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

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

（5）Input the IP address of server.

（6）Click Sign in to login system interface.

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

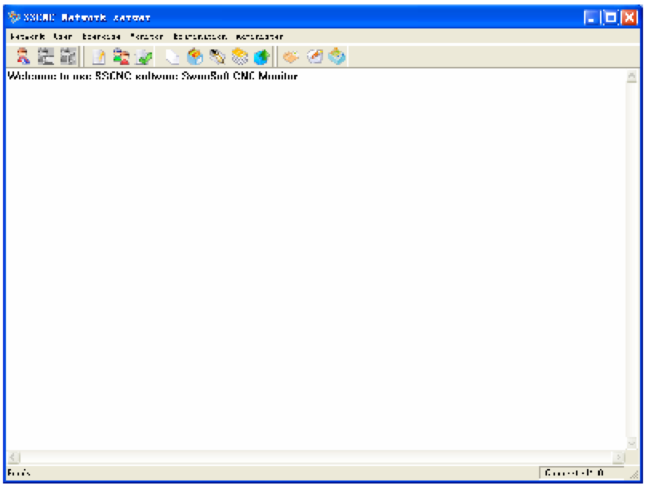


Fig. 2.1-3

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

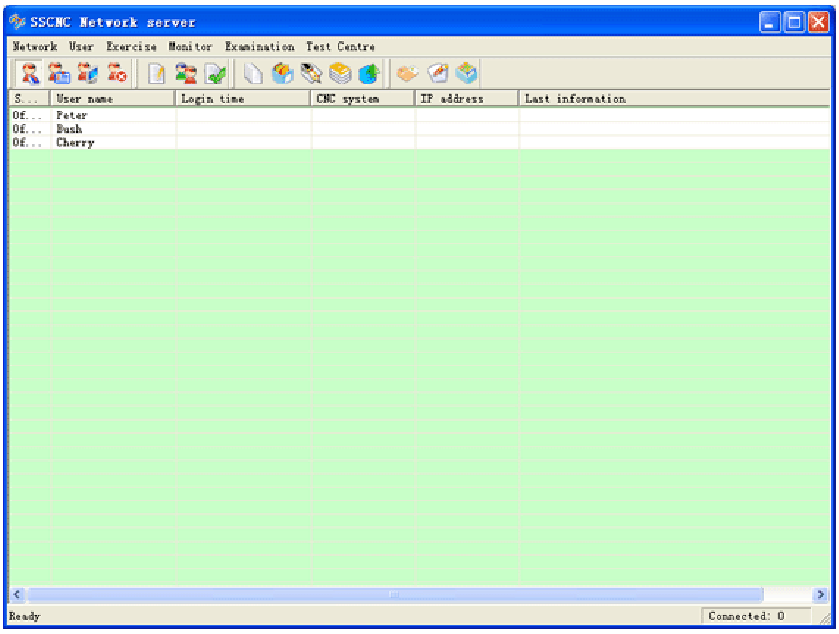


Fig. 2.1-4

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

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

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

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

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

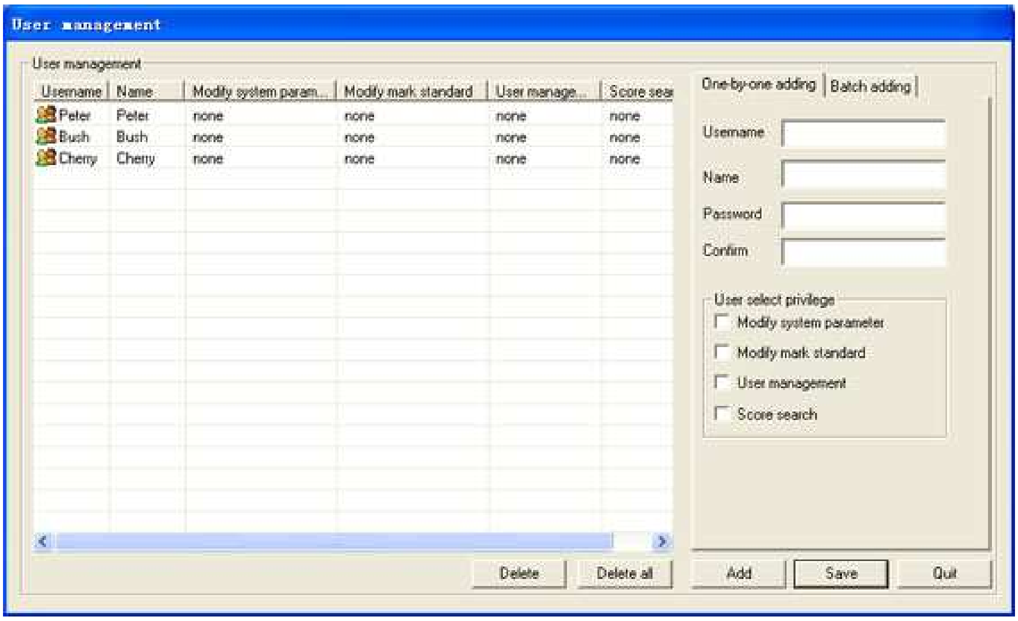


Fig. 2.1-5

2.1.3 SINGLE MACHINE VERSION STARTUP INTERFACE

Fig. 2.1-6

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

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

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

（4）Click Run to login system interface.

2.2 SETUP OF TOOLBAR AND MENU

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

Brief introduction of toolbar：

Setup new NC file

Open saved file(such NC file)

Save file(such as NC file)

Save as

Machine parametar

Cutter library management

Pattern of workpiece display

Choose size of workblank and coordinate of workpiece

Open/close machine door

Scrap iron display

Screen arrange： change screen arrange function by fixed sequence

Whole screen zoom up

Whole screen zoom down

Screen zoom up,zoom down

Screen translation

Screen revolve

X-Z plane selection

Y-Z plane selection

Y-X Plane selection

Machine encloser swich

Workpiece measurement

voice controler

Coordinate display

Jacket water display

Workblank display

Component display

Clarity display

ACT display

Display tools spacing number

Cutter display

Cutter path

Online help

REC parameter setup

REC start

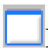
REC stop

teaching start/stop

2.3 FILE MANAGEMENT MENU

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

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

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

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

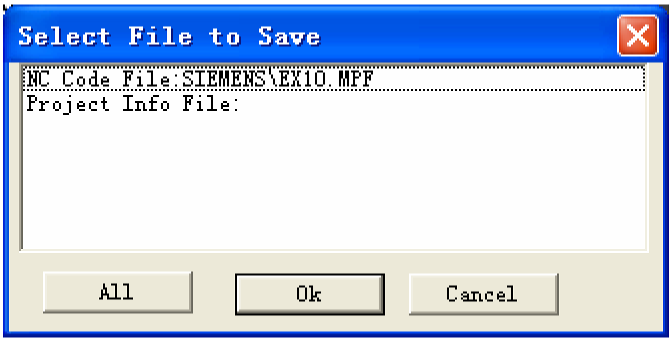


Fig.2.3-1

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

Load project file

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

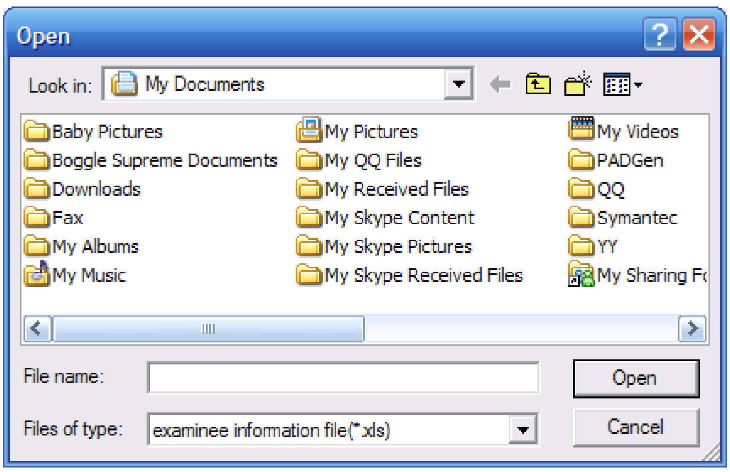


Fig.2.3-2

Project file save

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

2.3.1 MACHINE PARAMETER

a. Machine parameter setup：

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

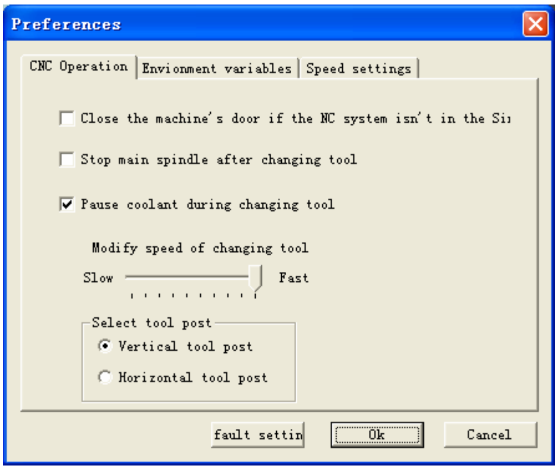


Fig.2.3-3

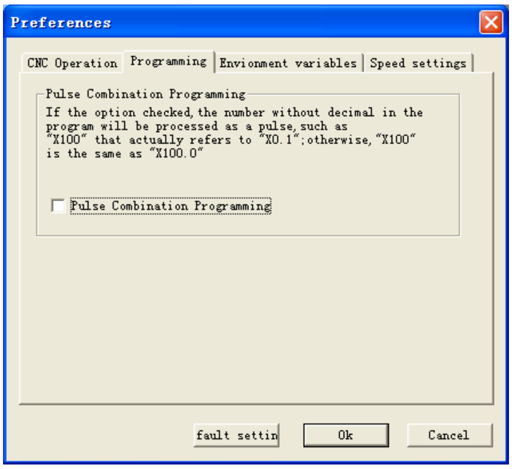


Fig.2.3-4

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

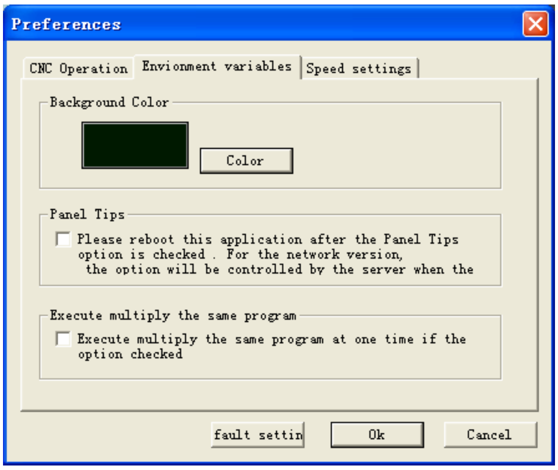


Fig.2.3-5

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

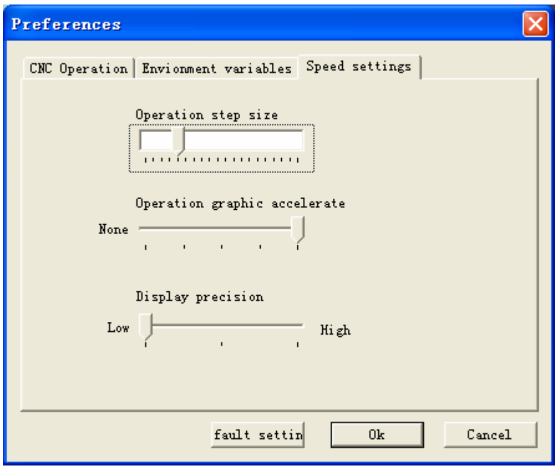


Fig.2.3-6

b.Display color：

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

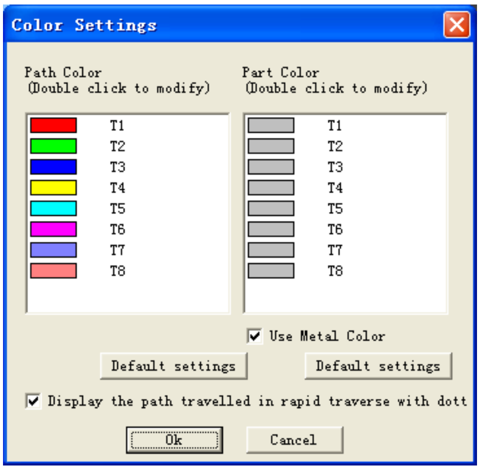


Fig.2.3-7

2.3.2 CUTTER MANAGEMENT

a. Milling machine

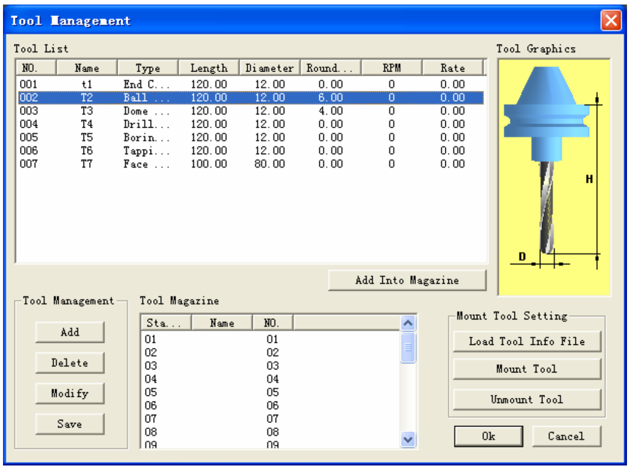


Fig.2.3-8

Add

(1).Input the number of tool

(2).Input the name of tool

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

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

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

Add tool to chief axes

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

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

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

b.lathe

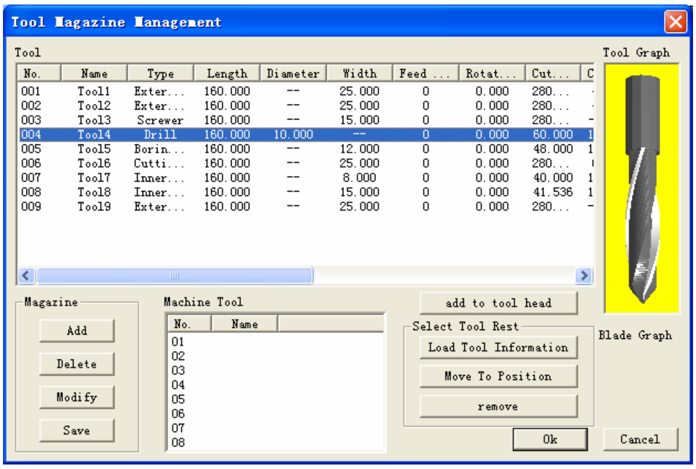


Fig. 2. 3-9

add

(1). Input the number of tool.

(2). Input the name of tool.

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

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

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

Internal circle tool adding：

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

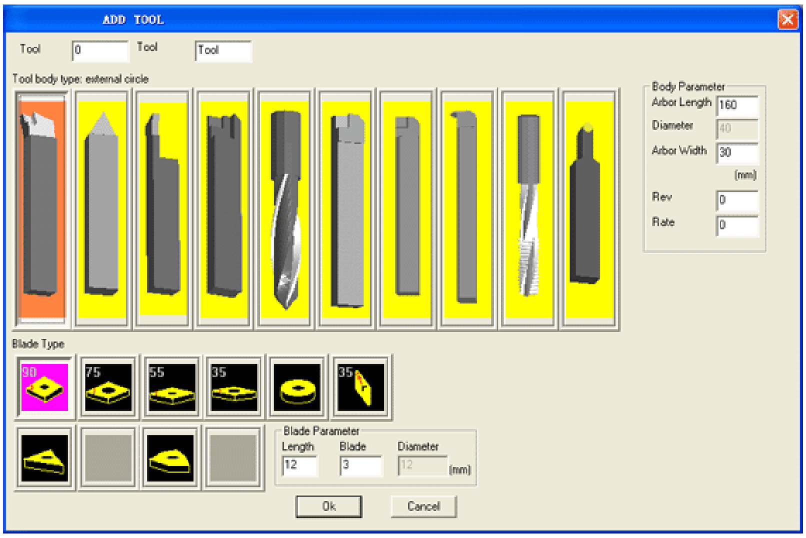


Fig. 2.3-10

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

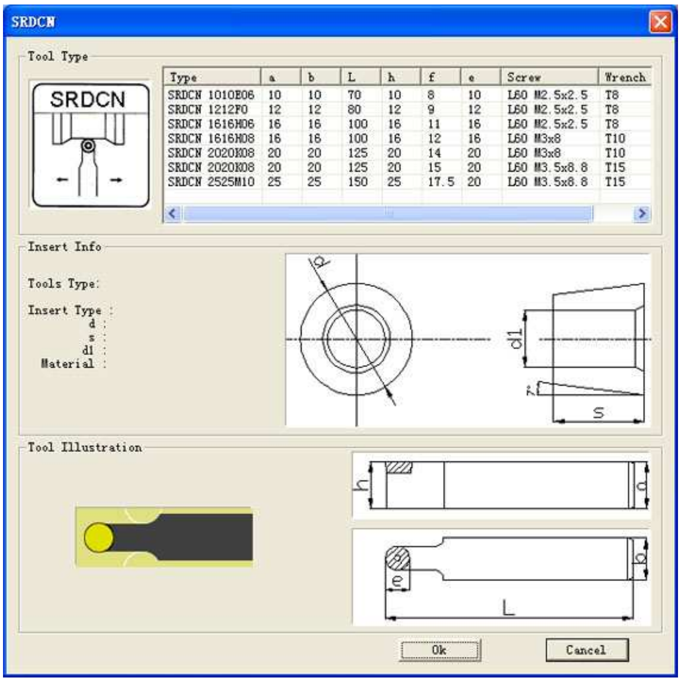


Fig. 2.3-11

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

Add tool to chief axes

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

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

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

2.3.3 WORKPIECE PARAMETER AND ACCESSORY

a. milling machine

Size of workblank、coordinate of workpiece

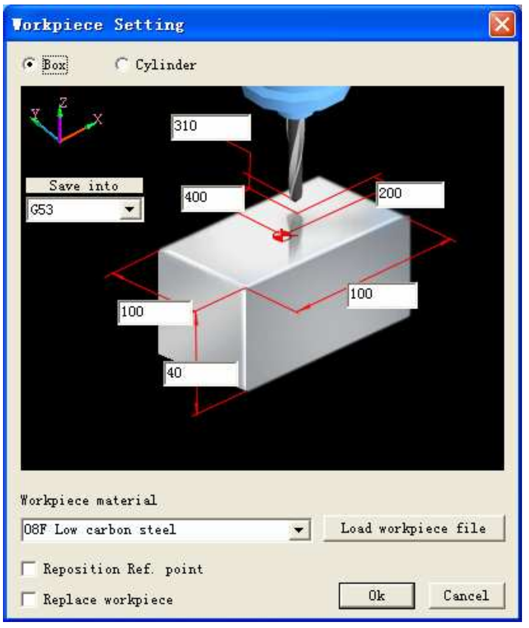


Fig. 2.3-12

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

(2)Define orgin of workpiece X、Y、Z.

(3)select changing machining orgin、changing workpiece.

b.Lathe

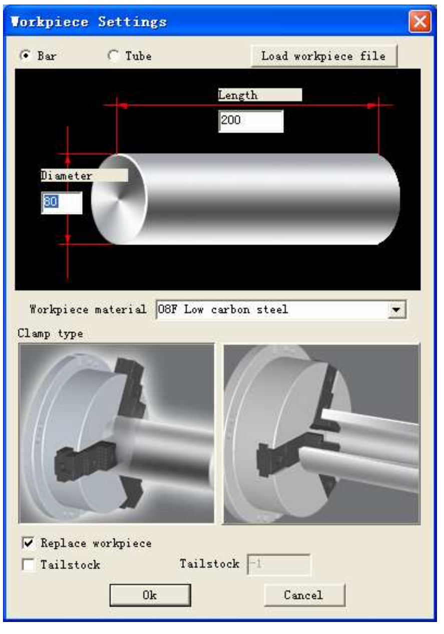


Fig. 2.3-13

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

(2)Define fixture.

(3)Choose tailstock.

Choose workholding fixture

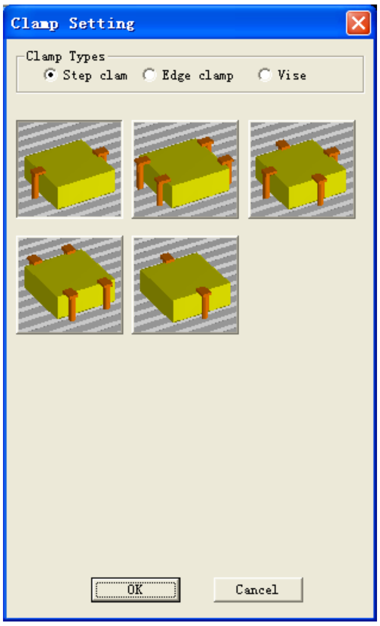


Fig. 2.3-14

Workpiece placement

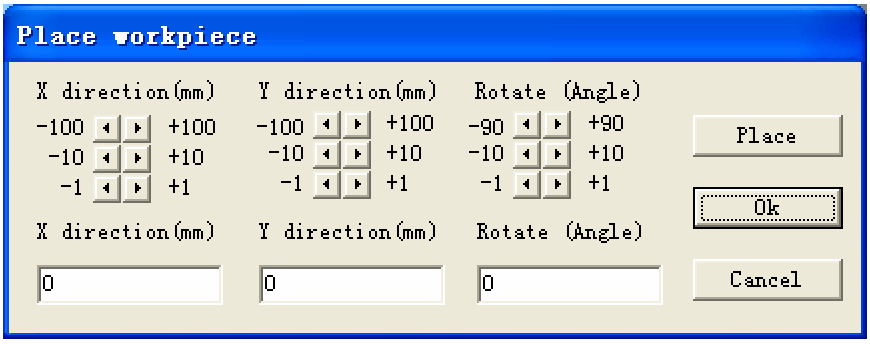


Fig. 2.3-15

(1)Choose the placement of direction X.

(2)Choose the placement of direction Y.

(3)Choose the placement of angle.

(4)Press“Place”and“Confirm”.

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

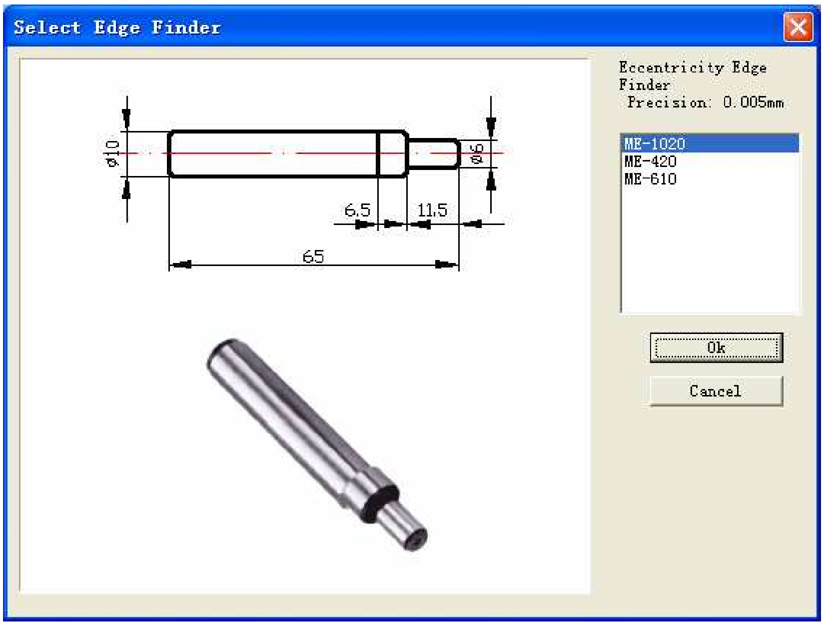


Fig. 2.3-16

Coolant pipe adjusting

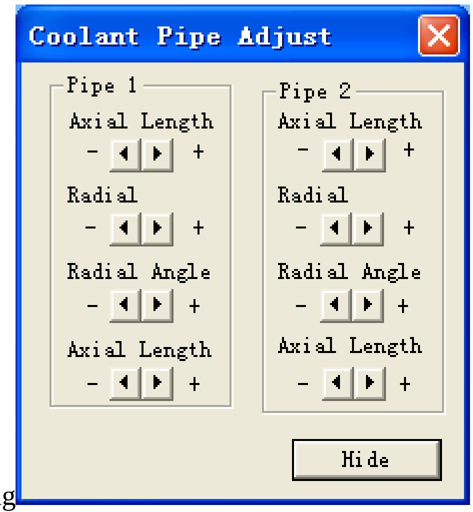


Fig. 2.3-17

2.3.4 RAPID SIMULATIVE MACHINING

(1)Programme by EDIT.

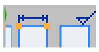
(2)Choose tool.。

(3)Choose workblank and workpiece null point.

(4)Placement mode AUTO.

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

2.3.5 WORKPIECE MEASUREMENT



Three modes of measurement

(1)Feature point.

(2)Feature line.

(3)Distribution of roughness.

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

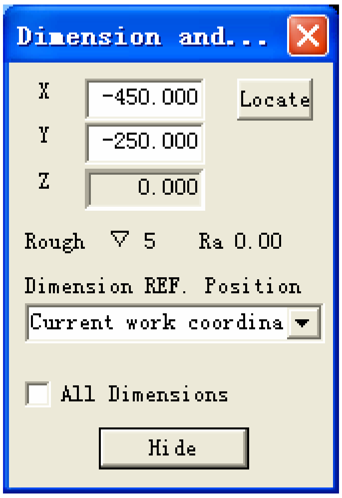


Fig. 2.3-18

2.3.6 REC PARAMETER SETUP

Three modes of REC area selection,setup as

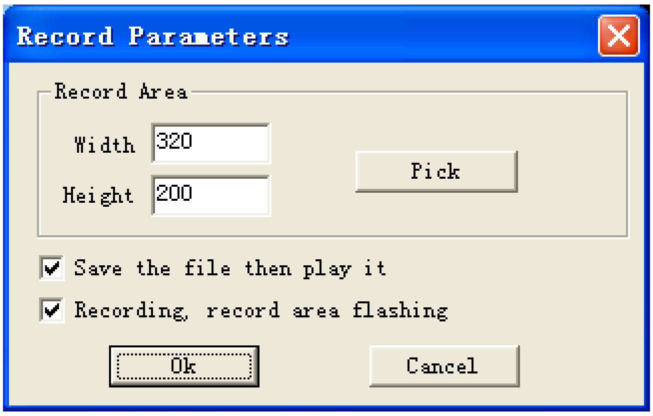


Fig. 2.3-19

2.3.7 WARING MESSAGE

Output current message files

Output all message files

Last day message

Next day message

Delete current message files

Parameter setup

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

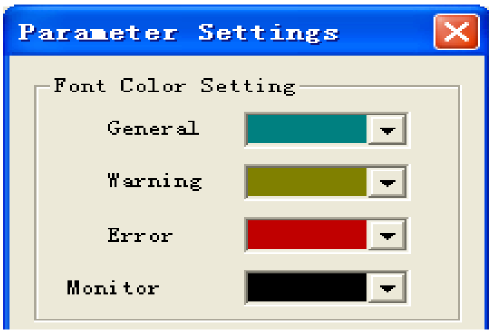


Fig. 2.3-20 Font color setup

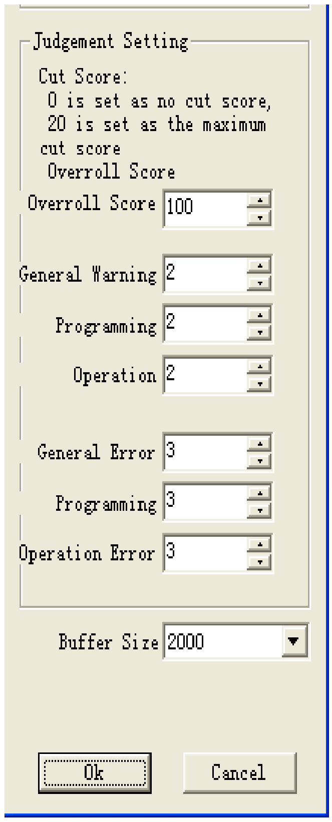


Fig. 2.3-21 Gradeing standard

1. VULGAR WARINGS

Return to reference point!

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

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

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

Modality is not booked！ Please book first!

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

Cutter parameter is incorrect!

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

No tool hasing this tool number in top rest!

Please backoff measuring piercing point bar before auto-toochange!

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

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

2. PROGRAMMING WARING

Search program， no O****!

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

3. MACHINE PPERATION WARING

Electric source is not opened or intense electricity is unavailable!

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

Please close machine door!

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

4. VULGAR ERRORS

Please backoff spindle measurement piercing point bar before startup NCSTART

X direction overshoot

Y direction overshoot

Z direction overshoot

5. PROGRAMMING ERRORS

General G code and cyclic program are something the matter!

No O*** in program direction!

Cutter number is on-unit!

Radius compensation register number D is on-unit!

Length compensation register number H is on-unit!

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

Vice program number is inexistence in subprogram call!

Vice program number is error in subprogram call!

It is lack of value F in G code!

There is no straightaway leadingin in tool compensation!

There is no straightaway eduction in tool compensation!

6. MACHINE OPERATION ERRORS

Cutter comes up against workbench!

Measuring piercing point bar comes up against workbench!

End face comes up against workpiece!

Cutter comes up against holding fixture!

Spindle is not stared,tool collision!

Measuring piercing point bar comes up against tool!

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

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

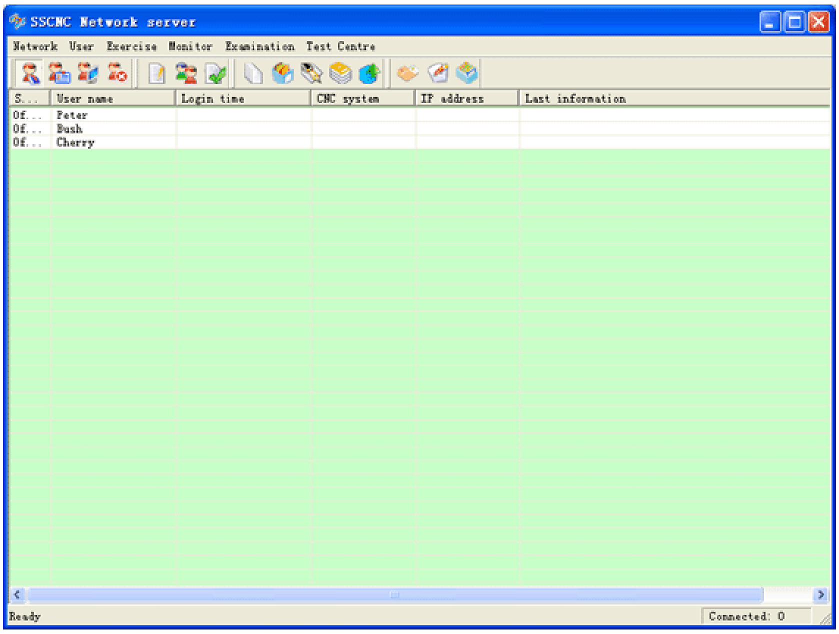


Fig. 2.3-22 Network management

CHAPTER 3 EZMotion-NC OPERATION

3.1 EZMotion-NC MACHINE PANEL OPERATION

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

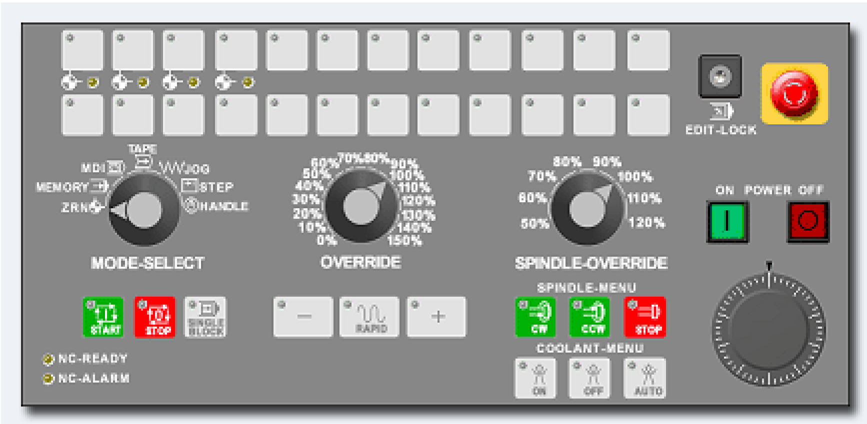


Fig. 3.1－1 E60(milling machine)panel

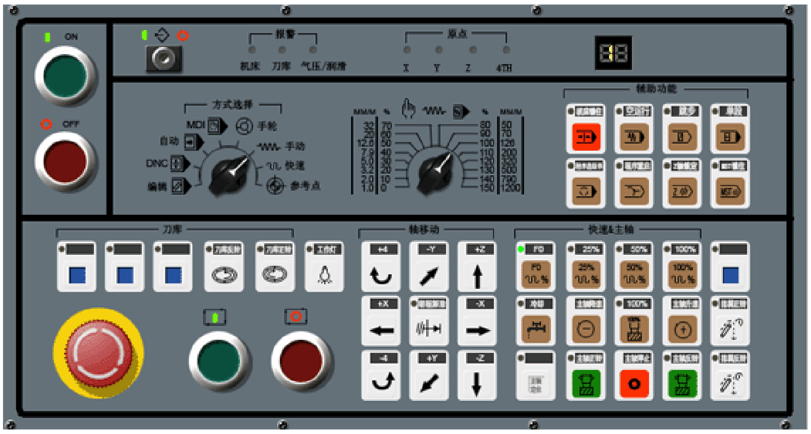
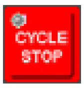


Fig. 3.1－2 E68T (lathe)panel

Program running startup.



Program running stop

Program running M00 stop.

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.

3.2 EZMotion-NC SYSTEM OPERATION

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

Fig. 3.2－1 E60T(lathe)

Fig. 3.2－1 E68M (milling machine)

3.2.1 KEYSTOKE INTRODUCTION

Modifier. Erase data which is in input region.

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

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

Withdraw and linefeed key. End input of a row of program and then feed line.

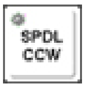
3.2.2 MANUAL OPERATION OF VIRTUAL NC MACHINE

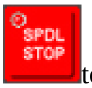
1.START、STOP SPINDLE

(1) Put mode knob at“JOG” 

.

(2) Press 



to start the spindle, while press 

to stop the spindle.

2.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 

3.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” 

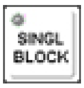
(3) Press 

(4)Press 

4.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.

5.CHOOSE A PROGRAM

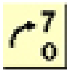
There are two methods to choose：

SEARCH ACCORDING TO PROGRAM NUMBERING

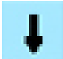
(1) Choose“EDIT”mode

(2) Press 

to input letter“O”

(3) Press 

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

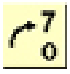
(4) Press cursor  to search; After found, “O7”is showed at the top right corner of screen, and“O7”NC program is on the screen.

CHOOSE AUTO 

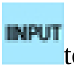
MODE

(1) Press 

to input letter “O”

(2) Press 

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

(3) Press 

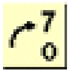
to start searching .“O7” is showed at the top ight corner of screen.

6.DELETE A PROGRAM

(1) Set mode at “EDIT”

(2) Press 

to input letter “O”

(3) Press 

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

(4) Press 

to delete “O7”NC program.

7.DELETE ALL PROGRAMS

(1) Set mode at “EDIT”

(2) Press 

to input letter “O”

(3) Input“-9999”

(4) Press 

to delete all programs

8.SEARCH A SPECIFYED CODE

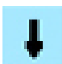
A specifyed code can be: a letteror a complete code. Such as:“N0010”,“M”,“F”,“G03”and so on. Searching is processed in current program. The operation step ：

（1）Set mode at“AUTO”or“EDIT”

（2）Press 

（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.

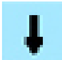
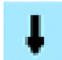
9. EDIT NC PROGRAM（DELETE、INSERT、REPLACE）

（1）Set mode at“EDIT”

（2）Select 

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

（4）Move cursor：

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

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

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

Delete、insert、replace：

Press  to delete code pointted by cursor

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

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

CHAPTER 4 EZMotion programme

4.1 Position

Position command methods ; G90, G91

By using the G90 and G91 commands, it is possible to execute the next coordinate commands using absolute values or incremental values.

The R-designated circle radius and the center of the circle determined by I, J, K are always incremental value commands.

G9D Xx1 Yy1 Zz1 αα1

G90 :Absolute value command

G91 :Incremental command

α :Additional axis

(1) Regardless of the current position, in the absolute value mode, it is possible to move to the position of the workpiece coordinate system that was designated in the program.

N 1 G90 G00 X0 Y0 ;

In the incremental value mode, the current position is the start point (0), and the movement is made only the value determined by the program, and is expressed as an incremental value.

N 2 G90 G01 X200. Y50. F100;

N 2 G91 G01 X200. Y50. F100;

Using the command from the 0 point in the workpiece coordinate system, it becomes the same coordinate command value in either the absolute value mode or the incremental value mode.

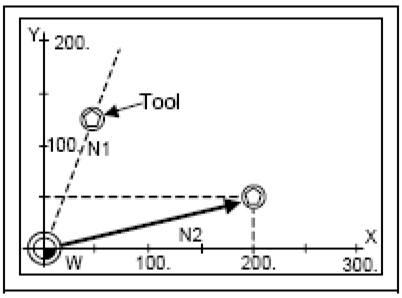


Fig.4.1-1

(2) For the next block, the last G90/G91 command that was given becomes the modal.

(G90) N 3 X100. Y100.;

The axis moves to the workpiece coordinate system X = 100mm and Y = 100mm position.

(G91) N 3 X–100. Y50.;

The X axis moves to -100.mm and the Y axis to +50.0mm as an incremental value, and as a result X moves to 100.mm and Y to 100.mm.

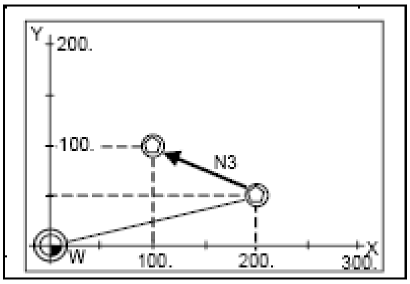


Fig.4.1-2

(3) Since multiple commands can be issued in the same block, it is possible to command specific addresses as either absolute values or incremental values.

N 4 G90 X300. G91 Y100.;

The X axis is treated in the absolute value mode, and with G90 is moved to the workpiece coordinate system 300.mm position. The Y axis is moved +100.mm with G91. As a result, Y moves to the 200.mm position. In terms of the next block, G91 remains as the modal and becomes the incremental value mode.

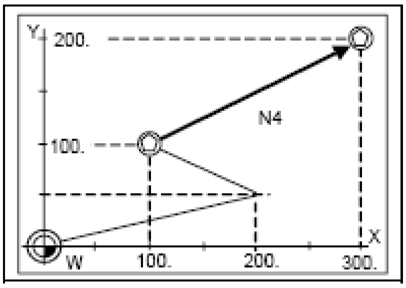


Fig.4.1-3

(4) When the power is turned ON, it is possible to select whether you want absolute value commands or incremental value commands with the #1073 I_Absm parameter.

(5) Even when commanding with the manual data input (MDI), it will be treated as a modal from that block.

Inch/metric command change; G20, G21

G20/G21;

G20 : Inch command

G21 : Metric command

G20 and G21 selection is meaningful only for linear axes and it is meaningless for rotary axes.

The input unit for G20 and G21 will not change just by changing the command unit.

In other words, if the machining program command unit changes to an inch unit at G20 when the initial inch is OFF, the setting unit of the tool offset amount will remain metric. Thus, take note to the setting value.

4.2 G Commands

Positioning (Rapid traverse); G00

This command is accompanied by coordinate words. It positions the tool along a linear or non-linear path from the present point as the start point to the end point which is specified by the coordinate words.

Command format

G00 Xx Yy Zz αα ; (α represents additional axis)

x, y, z, α : Represent coordinates, and could be either absolute values or incremental values, depending on the setting of G90/G91.

(1) Once this command has been issued, the G00 mode is retained until it is changed by another G function or until the G01, G02, G03 or G33 command in the 01 group is issued. If the next command is G00, all that is required is simply that the coordinate words be specified.

(2) In the G00 mode, the tool is always accelerated at the start point of the block and decelerated at the end point. Execution proceeds to the next block after the in-position state has been checked. The in-position width is set by parameter.

(3) Any G command (G72 to G89) in the 09 group is cancelled (G80) by the G00 command.

(4) Whether the tool moves along a linear or non-linear path is determined by parameter, but the positioning time does not change.

(a) Linear path..........: This is the same as linear interpolation (G01), and the speed is limited by the rapid traverse rate of each axis.

(b) Non-linear path...: The tool is positioned at the rapid traverse rate independently for each axis.

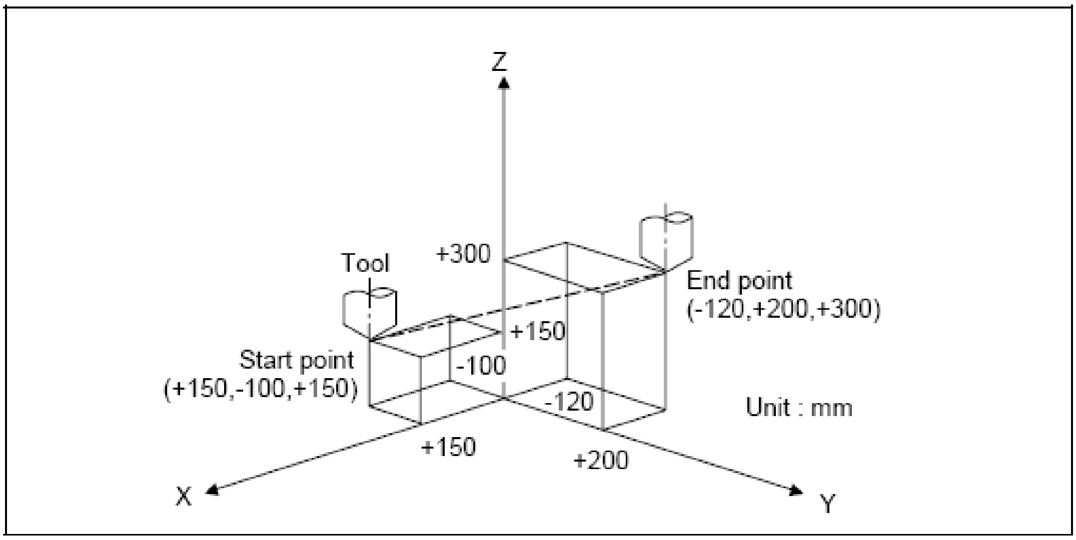


Fig.4.2-1

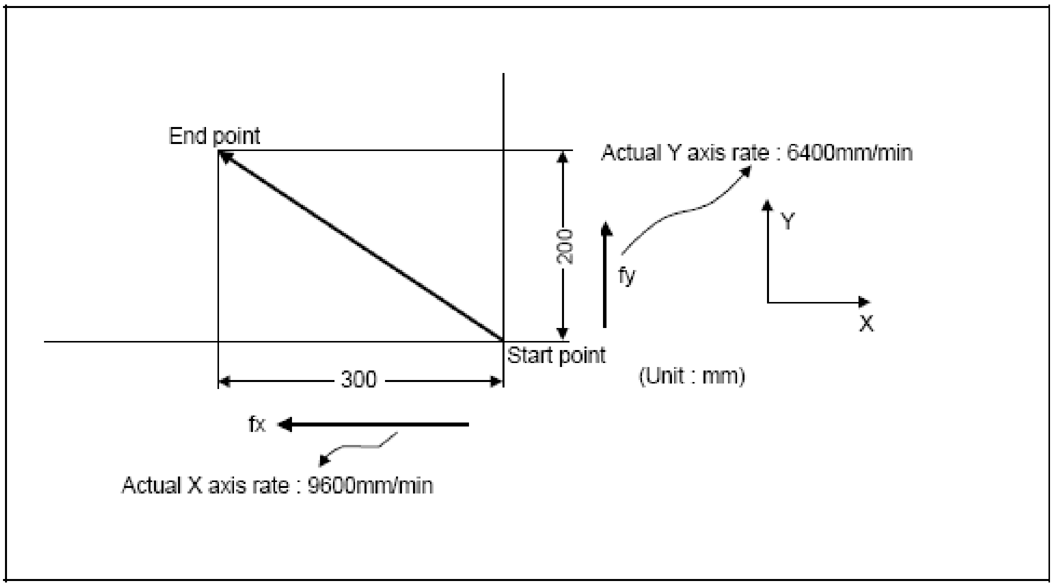


Fig.4.2-2

Linear interpolation; G01

This command is accompanied by coordinate words and a feedrate command.

It makes the tool move (interpolate) linearly from its present position to the end point specified by the coordinate words at the speed specified by address F. In this case, the feedrate specified by address F always acts as a linear speed in the tool nose center advance direction.

Command format

G01 Xx Yy Zz αα Ff ; (α represents additional axis)

x, y, z, α :Coordinate values and may be an absolute position or incremental position depending on the G90/G91 state.

f :Value that indicates the speed data.

Once this command is issued, the mode is maintained until another G function (G00, G02, G03, G33) in the 01 group which changes the G01 mode is issued. Therefore, if the next command is also G01 and if the feedrate is the same, all that is required to be done is to specify the coordinate words. If no F command is given in the first G01 command block, program error (P62) results.

The feedrate for a rotary axis is commanded by °/min (decimal point position unit). (F300 = 300°/min)

The G functions (G70 - G89) in the 09 group are cancelled (G80) by the G01 command.

Example of program

(Example 1) Cutting in the sequence of $P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow P_4 \rightarrow P_1$ at 300 mm/min feedrate

P_0

$\rightarrow P_1$ is for tool positioning

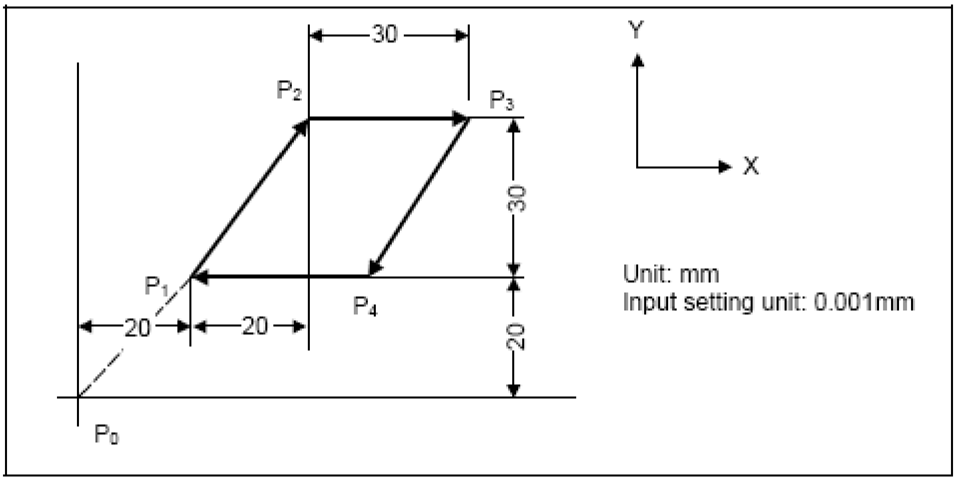


Fig.4.2-3

G90	G00	X20000	Y20000 ;	$P_0 \rightarrow P_1$
	G01	X20000	Y30000 F300	$P_1 \rightarrow P_2$
		X30000	;	$P_2 \rightarrow P_3$
		X-20000	Y-30000 ;	$P_3 \rightarrow P_4$
		X-30000 ;		$P_4 \rightarrow P_1$

Fig.4.2-4

Plane selection; G17, G18, G19

Function and purpose

The plane to which the movement of the tool during the circle interpolation (including helical cutting) and tool diameter compensation command belongs is selected.

By registering the basic three axes and the corresponding parallel axis as parameters, a plane can be selected by two axes that are not the parallel axis. If the rotary axis is registered as a parallel axis, a plane that contains the rotary axis can be selected.

For the standard plane selection, the relation of the basic three axes X, Y and Z and the corresponding parallel axes U, V and W is fixed, so a plane containing the rotary axis (axis name A, B, C) cannot be selected.

The plane selection is as follows:

• Plane that executes circular interpolation (including helical cutting)

• Plane that executes tool diameter compensation

• Plane that executes fixed cycle positioning.

Command format

G17 ;

G18 ;

G19 ;

(ZX plane selection)

(YZ plane selection)

(XY plane selection)

X, Y and Z indicate each coordinate axis or the parallel axis.

Parameter entry

#1026 to 1028

base_I,J,K

#1029 to 1039

aux_I,J,K

I X U

J Y

K Z V

Example of plane selection parameter entry

As shown in the above example, the basic axis and its parallel axis can be registered.

The basic axis can be an axis other than X, Y and Z.

Axes that are not registered are irrelevant to the plane selection.

Plane selection system

In Fig. 1,

I is the horizontal axis for the G17 plane or the vertical axis for the G18 plane

J is the vertical axis for the G17 plane or the horizontal axis for the G19 plane

K is the horizontal axis for the G18 plane or the vertical axis for the G19 plane

In other words,

G17 ..... IJ plane

G18 ..... KI plane

G19 ..... JK plane

The axis address commanded in the same block as the plane selection (G17, G18, G19) determines which basic axis or parallel axis is used for the plane selection.

For the parameter registration example in Fig. 1.

G17X__Y__ ; XY plane

G18X__V__ ; VX plane

G18U__V__ ; VU plane

G19Y__Z__ ; YZ plane

G19Y__V__ ; YV plane

The plane will not changeover at a block where a plane selection G code (G17, G18, G19) is not commanded.

G17X__Y__ ; XY plane

Y__Z__ ; XY plane (plane does not change)

If the axis address is omitted in the block where the plane selection G code (G17, G18, G19) is commanded, it will be viewed as though the basic three axes address has been omitted.

For the parameter registration example in Fig. 1.

G17 ; XY plane

G17U__ ; UY plane

G18U__ ; ZU plane

G18V__ ; VX plane

G19Y__ ; YZ plane

G19V__ ; YV plane

The axis command that does not exist in the plane determined by the plane selection G code (G17, G18, G19) is irrelevant to the plane selection.

For the parameter registration example in Fig. 1.

G17U__Z__ ;

If the above is commanded, the UY plane will be selected, and Z will move regardless of the plane.
If the basic axis and parallel axis are commanded in duplicate in the same block as the plane selection G code (G17, G18, G19), the plane will be determined in the priority order of basic axis and parallel axis.
For the parameter registration example in Fig. 1.
G17U__Y__W__-;
If the above is commanded, the UY plane will be selected, and W will move regardless of the plane.
(Note 1) The plane set with parameter "#1025 I_plane" will be selected when the power is turned ON or reset.

Circular interpolation; G02, G03

G02 (G03) Xx Yy Ii Jj Kk Ff;

G02 : Clockwise (CW)
G03 : Counterclockwise (CCW)
Xx, Yy : End point
Ii, Jj : Arc center
Ff : Feedrate

For the arc command, the arc end point coordinates are assigned with addresses X, Y (or Z, or parallel axis X, Y, Z), and the arc center coordinate value is assigned with addresses I, J (or K). Either an absolute value or incremental value can be used for the arc end point coordinate value command, but the arc center coordinate value must always be commanded with an incremental value from the start point.
The arc center coordinate value is commanded with an input setting unit. Caution is required for the arc command of an axis for which the input command value differs. Command with a decimal point to avoid confusion.

Plane selection

The planes in which the arc exists are the following three planes (refer to the detailed drawings), and are selected with the following method.
XY plane
G17; Command with a (plane selection G code)
ZX plane
G18; Command with a (plane selection G code)
YZ plane
G19; Command with a (plane selection G code)

Helical interpolation ; G17 to G19, G02, G03

Function and purpose

While circular interpolating with G02/G03 within the plane selected with the plane selection G code (G17, G18, G19), the 3rd axis can be linearly interpolated.
Normally, the helical interpolation speed is designated with the tangent speed F' including the 3rd axis interpolation element as shown in the lower drawing of Fig. 1. However, when designating the arc plane element speed, the tangent speed F on the arc plane is commanded as shown in the upper drawing of Fig. 1.
The NC automatically calculates the helical interpolation tangent speed F' so that the tangent speed on the arc plane is F.

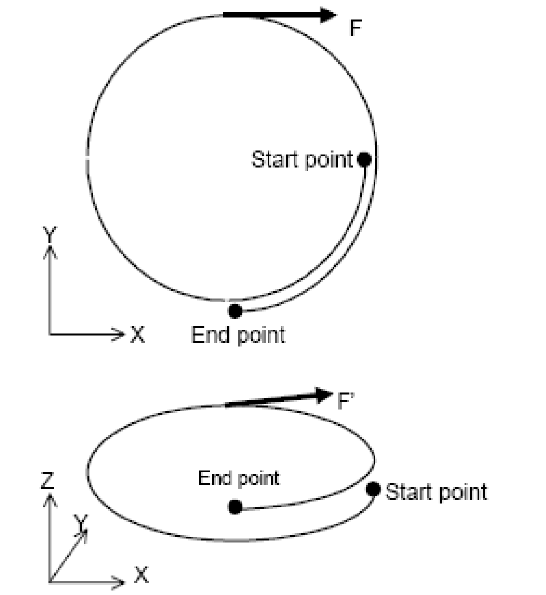


Fig. 1 Designation of helical interpolation speed

Fig.4.2-5

Constant lead thread cutting ; G33

Function and purpose

The G33 command exercises feed control over the tool which is synchronized with the spindle rotation and so this makes it possible to conduct constant-lead straight thread-cutting and tapered thread-cutting.

Command format

G32 Zz Ff1 Qq ; (Normal lead thread cutting commands)

Zz : Thread cutting direction axis address (X, Y, Z, a) and thread length

Ff : Lead of long axis (axis which moves most) direction.

Qq : Thread cutting start shift angle, (0 to 360°)

G33 Zz Ee1 Qq ; (Precision lead thread cutting commands)

Zz : Thread cutting direction axis address (X, Y, Z, α) and thread length

Ee : Lead of long axis (axis which moves most) direction

Qq : Thread cutting start shift angle, (0 to 360°)

(1) The E command is also used for the number of ridges in inch thread cutting, and whether the ridge number or precision lead is to be designated can be selected by parameter setting.

(Precision lead is designated by setting the parameter "#1229 set 01/bit 1" set to 1.)

(2) The lead in the long axis direction is commanded for the taper thread lead.

Uni-directional positioning; G60

Function and purpose

The G60 command can position the tool at a high degree of precision without backlash error by locating the final tool position from a single determined direction.

G60 Xx Yy Zz αα ;

α : Optional axis

(1) The creep distance for the final positioning as well as the final positioning direction is set by parameter.

(2) After the tool has moved at the rapid traverse rate to the position separated from the final

position by an amount equivalent to the creep distance, it move to the final position in accordance with the rapid traverse setting where its positioning is completed.

(3) The above positioning operation is performed even when Z-axis commands have been assigned for Z-axis cancel and machine lock. (Display only)

(4) When the mirror image function is ON, the tool will move in the opposite direction as far as the intermediate position due to the mirror image function but the operation within the creep distance during its final advance will not be affected by mirror image.

(5) The tool moves to the end point at the dry run speed during dry run when the G0 dry run function is valid.

(6) Feed during creep distance movement with final positioning can be stopped by resetting, emergency stop, interlock, feed hold and rapid traverse override zero.

The tool moves over the creep distance at the rapid traverse setting. Rapid traverse override is valid.

(7) Uni-directional positioning is not performed for the drilling axis during drilling fixed cycles.

(8) Uni-directional positioning is not performed for shift amount movements during the fine boring or back boring fixed cycle.

(9) Normal positioning is performed for axes whose creep distance has not been set by parameter.

(10) Uni-directional positioning is always a non-interpolation type of positioning.

(11) When the same position (movement amount of zero) has been commanded, the tool moves back and forth over the creep distance and is positioned at its original position from the final advance direction.

(12) Program error (P61) results when the G60 command is assigned with an NC system which has not been provided with this particular specification.

Synchronous feed; G94, G95

Function and purpose

Using the G95 command, it is possible to assign the feed amount per rotation with an F code.

When this command is used, the rotary encoder must be attached to the spindle.

When the G94 command is issued the per-minute feed rate will return to the designated per-minute feed (asynchronous feed) mode.

Command format

G94;

G95;

G94 : Per-minute feed (mm/min) (asynchronous feed) (F1 = 1mm/min)

G95 : Per-revolution feed (mm/rev) (synchronous feed) (F1 = 0.01mm/rev)

The G95 command is a modal command and so it is valid until the G94 command (per-minute feed) is next assigned.

(1) The F code command range is as follows.

The movement amount per spindle revolution with synchronous feed (per-revolution feed) is assigned by the F code and the command range is as shown in the table below.

	Metric input		Inch input	
Input unit system	0.001mm		0.0001inch	
Command mode	Feed per minute	Feed per rotation	Feed per minute	Feed per rotation
Command address	F (mm/min)	E (mm/rev)	F (inch/min)	E (inch/rev)
Minimum command unit	1 (= 1.00), (1. = 1.00)	1 (= 0.01), (1. = 1.00)	1 (= 1.000), (1. = 1.000)	1 (= 0.001), (1. = 1.000)
Command range	0.01 to 1000000.00	0.001 to 999.999	0.001 to 100000.0000	0.0001 to 999.9999

Fig.4.2-6

(2) The effective speed (actual movement speed of machine) under per-revolution feed conditions is given in the following formula (Formula 1).

FC = F × N × OVR ..... (Formula 1)

Where FC = Effective rate (mm/min, inch/min)

F = Commanded feedrate (mm/rev, inch/rev)

N = Spindle speed (r/min)

OVR = Cutting feed override

When a multiple number of axes have been commanded at the same time, the effective rate FC in formula 1 applies in the vector direction of the command.

Exact stop check; G09

Function and purpose

In order to prevent roundness during corner cutting and machine shock when the tool feedrate changes suddenly, there are times when it is desirable to start the commands in the following block once the in-position state after the machine has decelerated and stopped or the elapsing of the deceleration check time has been checked. The exact stop check function is designed to accomplish this purpose.

Either the deceleration check time or in-position state is selected with parameter "#1193 inpos".

In-position check is valid when "#1193 inpos" is set to 1.

The in-position width is set with parameter "#2224 sv024" on the servo parameter screen by the machine manufacturer.

Command format

G09 ;

The exact stop check command G09 has an effect only with the cutting command (G01 - G03) in its particular block.

Example of program

N001 G09 G01 X100.000 F150 ; The following block is started once the deceleration
check time or in-position state has been checked after
the machine has decelerated and stopped.

N002 Y100.000

Exact stop check mode ; G61

Function and purpose

Whereas the G09 exact stop check command checks the in-position status only for the block in which the command has been assigned, the G61 command functions as a modal. This means that deceleration will apply at the end points of each block to all the cutting commands (G01 to G03) subsequent to G61 and that the in-position status will be checked. G61 is released by automatic corner override (G62), tapping mode (G63), or cutting mode (G64).

Command format

G61 ;

In-position check is executed in the G61 block, and thereafter, the in-position check is executed at the end of the cutting command block is executed until the check mode is canceled.

Function and purpose

The deceleration check is a function that determines the method of the check at the completion of the axis movement block's movement.

The deceleration check includes the in-position check and commanded speed check method.

The G0 and G1 deceleration check method combination can be selected.

(Refer to section "Deceleration check combination".)

With this function, the deceleration check in the reverse direction of G1 → G0 or G1 → G1 can be changed by changing the parameter setting.

G1 → G0 deceleration check

Detailed description

(1) In G1 → G0 continuous blocks, the parameter "#1502 G0Ipfg" can be changed to change the deceleration check in the reverse direction.

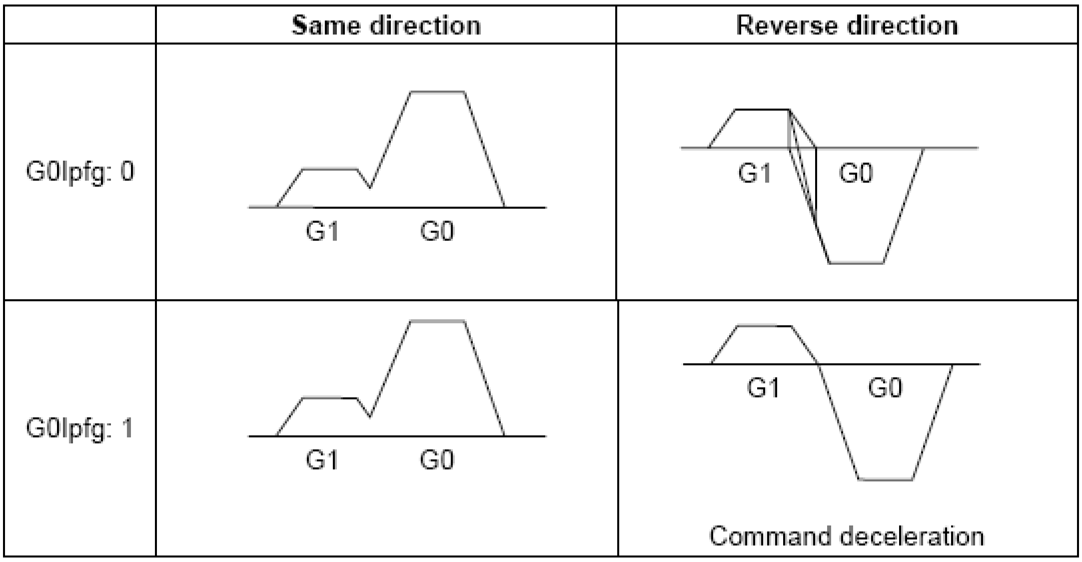


Fig.4.2-7

Automatic corner override ; G62

Function and purpose

With tool radius compensation, this function reduces the load during inside cutting of automatic corner R, or during inside corner cutting, by automatically applying override to the feed rate.

Automatic corner override is valid until the tool radius compensation cancel (G40), exact stop mode (G61), tapping mode (G63) and cutting mode (G64) command is issued.

Command format

G62 ;

Machining inside corners

When cutting an inside corner as in Fig. 1, the machining allowance amount increases and a greater load is applied to the tool. To remedy this, override is applied automatically within the corner set range, the feedrate is reduced, the increase in the load is reduced and cutting is performed effectively.

However, this function is valid only when finished shapes are programmed.

Tapping mode ; G63

Function and purpose

The G63 command allows the control mode best suited for tapping to be entered, as indicated below :

1) Cutting override is fixed at 100%.

2) Deceleration commands at joints between blocks are invalid.

3) Feed hold is invalid.

4) Single block is invalid.

5) In-tapping mode signal is output.

G63 is released by the exact stop mode (G61), automatic corner override (G62), cutting mode (G64) command.

Command format

G63 ;

7.10 Cutting mode ; G64

Function and purpose

The G64 command allows the cutting mode in which smooth cutting surfaces are obtained to be established. Unlike the exact stop check mode (G61), the next block is executed continuously with the machine not decelerating and stopping between cutting feed blocks in this mode.

G64 is released by the exact stop mode (G61), automatic corner override (G62), tapping mode (G63) command.

This cutting mode is established in the initialized status.

Command format

G64 ;

Per-second dwell ; G04

Function and purpose

The execution of the next block will be waited for the designated time.

Command format

G04 X__; or G04 P__;

Command unit : 0.001 s

Decimal point commands are invalid for address P and if such commands are assigned, everything following the decimal point will be ignored.

Detailed description

(1) The table below lists the dwell time.

Input setting unit	Command range based on address X	Command range based on address P
0.01/0.001mm	0.001 to 99999.999 (s)	1 to 99999999 (s) (×0.001 s)
0.0001inch	0.001 to 99999.999 (s)	1 to 99999999 (s) (×0.001 s)

Fig.4.2-8

(2) When a cutting command is in the previous block, the dwell command starts calculating the dwell time after the machine has decelerated and stopped. When it is commanded in the same

block as an M, S, T or B command, the calculation starts simultaneously.
(3) The dwell function is valid during interlock.
(4) Dwell is valid even for machine lock.

Constant surface speed control; G96, G97

10.4.1 Constant surface speed control

Function and purpose

These commands automatically control the spindle speed in line with the changes in the radius coordinate values as cutting proceeds in the diametrical direction, and they serve to keep the cutting point speed constant during the cutting.

Command format

G96 Ss Pp; Constant surface speed ON

Ss : Peripheral speed
Pp : Assignment of constant surface speed control axis

G97 ; Constant surface speed cancel

Detailed description

(1) The constant surface speed control axis is set by parameter (#1181 G96_ax).
0 : Fixed at 1st axis (P command invalid)
1 : 1st axis
2 : 2nd axis
3 : 3rd axis
(2) When the above-mentioned parameter is not zero, the constant surface speed control axis can be assigned by address P.
(Example) With G96_ax (1)

Program Constant surface speed control axis

G96 S100 ; 1st axis
G96 S100 P3 ; 3rd axis
(3) Example of selection program and operation
G90 G96 G01 X50. Z100. S200 ;

∼

G97 G01 X50. Z100. F300 S500 ;

∼

M02 ;
The spindle speed is controlled so that the peripheral
speed is 200m/min.
The spindle speed is controlled to 500r/min.
The modal returns to the initial setting.

Spindle clamp speed setting; G92

Function and purpose

The maximum clamp speed of the spindle can be assigned by address S following G92 and the minimum clamp speed by address Q.

Command format

G92 Ss Qq;

Ss : Maximum clamp speed

Qq : Minimum clamp speed

Besides this command, parameters can be used to set the rotational speed range up to 4 stages in 1 r/min units to accommodate gear selection between the spindle and spindle motor.

The lowest upper limit and highest lower limit are valid among the rotational speed ranges based on the parameters and based on G92 Ss Qq ;

Set in the parameter whether to carry out rotation speed clamp only in the constant surface speed mode or even when the constant surface speed is canceled.

Tool length offset/cancel; G43, G44/G49

Function and purpose

The end position of the movement command can be offset by the preset amount when this command is used. A continuity can be applied to the program by setting the actual deviation from the tool length value decided during programming as the offset amount using this function.

Command format

When tool length offset is + When tool length offset is −

G43 Zz Hh ; Tool length offset + start

:

G49 Zz ; Tool length offset cancel

G44 Zz Hh ; Tool length offset − start

:

G49 Zz ;

Detailed description

(1) Tool length offset movement amount

The movement amount is calculated with the following expressions when the G43 or G44 tool length offset command or G49 tool length offset cancel command is issued.

Z axis movement amount

G43 Zz Hn$_1$; z + (λh_1) Offset in + direction by tool offset amount

G44 Zz Hh$_1$; z − (λh_1) Offset in − direction by tool offset amount

G49 Zz ; z −(+) (λh_1) Offset amount cancel.

λh_1 : Offset amount for offset No. h_1

Regardless of the absolute value command or incremental value command, the actual end point will be the point offset by the offset amount designated for the programmed movement command end point coordinate value.

The G49 (tool length offset cancel) mode is entered when the power is turned ON or when M02 has been executed.

(Example 1) For absolute value command

H01 = −100000

N1 G28 Z0 T01 M06 ;

N2 G90 G92 Z0 ;

N3 G43 Z5000 H01 ;

N4 G01 Z-50000 F500 ;

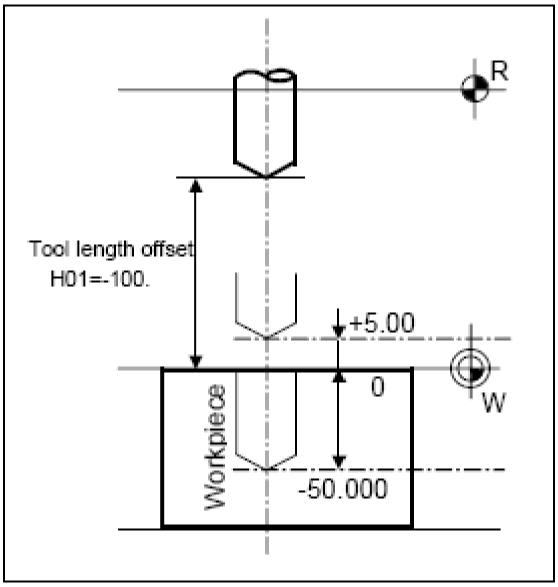


Fig.4.2-9

Program Support Functions

(a) G81 (Drilling, spot drilling)

Program

G81 Xx1 Yy1 Zz1 Rr1 Ff1 ;

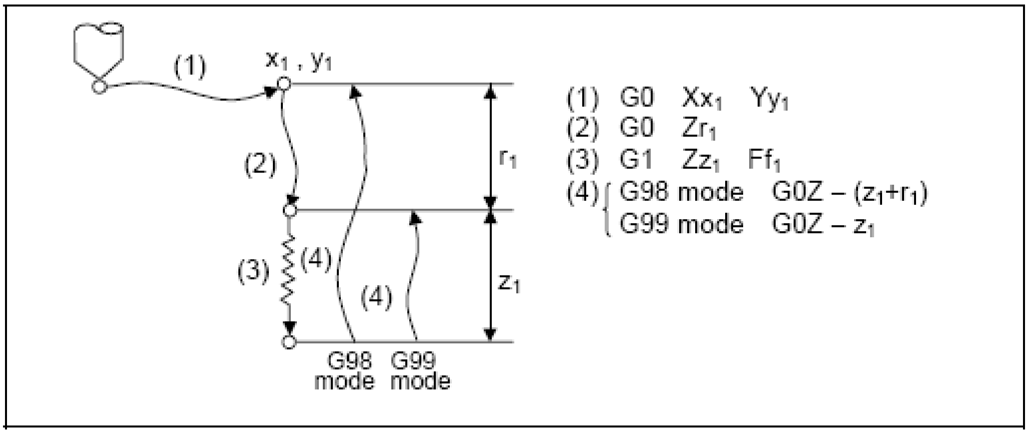


Fig.4.2-10

(b) G82 (Drilling, counter boring)

Program

G82 Xx_1 Yy_1 Zz_1 Rf_1 Ff_1 Pp_1 ;

P : Dwell designation

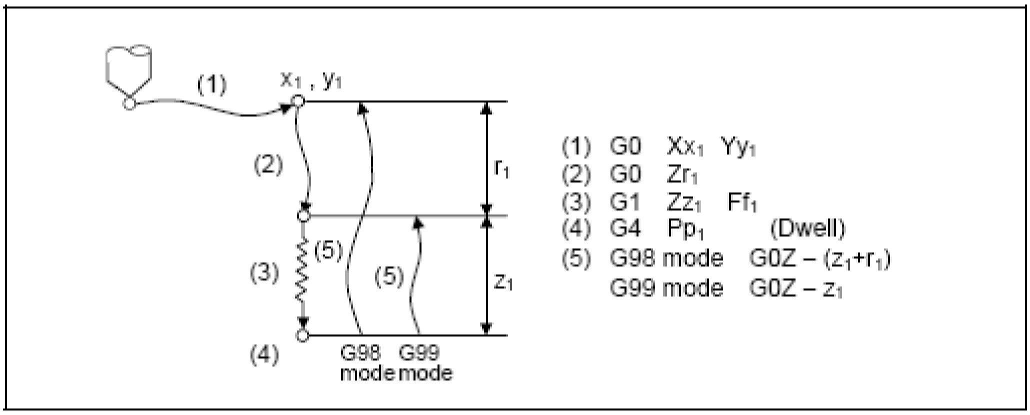


Fig.4.2-11

(c) G83 (Deep hole drilling cycle)

Program

G83 Xx_1 Yy_1 Zz_1 Rr_1 Qq_1 Ff_1 ;

Q : This designates the cutting amount per pass, and is always designated with an incremental value

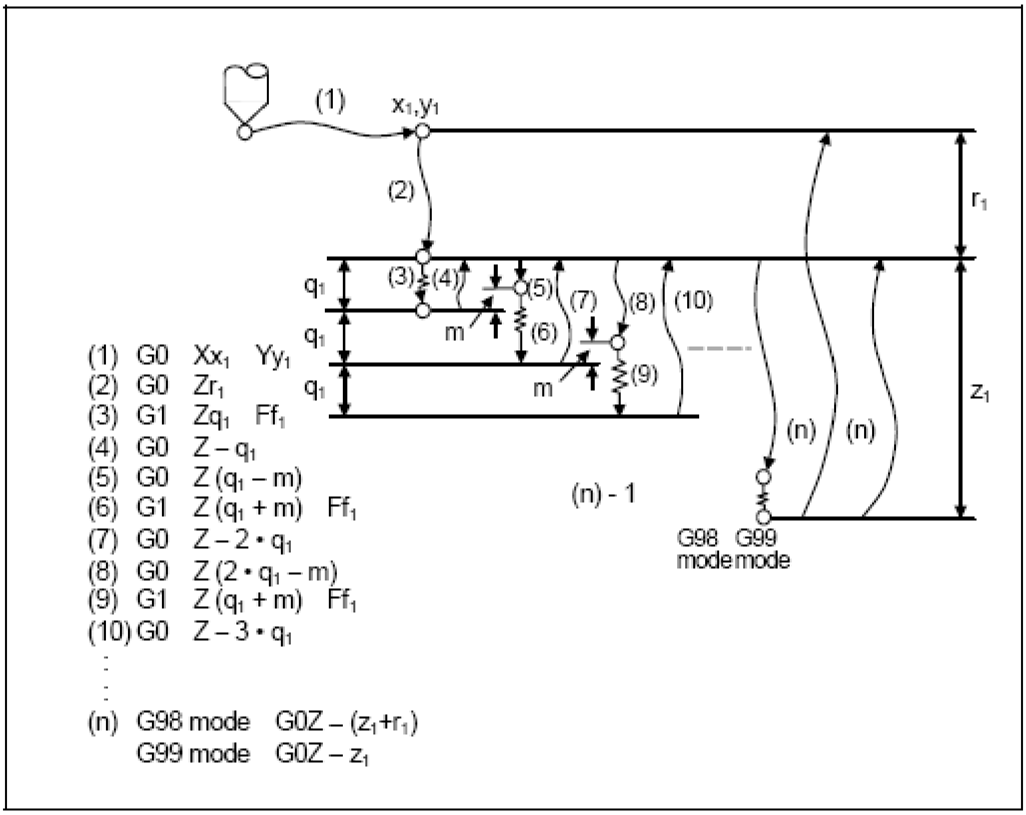


Fig.4.2-12

When executing a second and following cutting in the G83 as shown above, the movement will change from rapid traverse to cutting feed several mm before the position machined last. When the hole bottom is reached, the axis will return according to the G98 or G99 mode.

m will differ according to the parameter #8013 G83 return. Program so that q_1>m.

The operation stops at after the (1), (2) and (n) commands during single block operation.

(d) G84 (Tapping cycle)

Program

G84 Xx_1 Yy_1 Zz_1 Rr_1 Ff_1 Pp_1 Rr_2 (or S_1, S_2) ;

P : Dwell designation

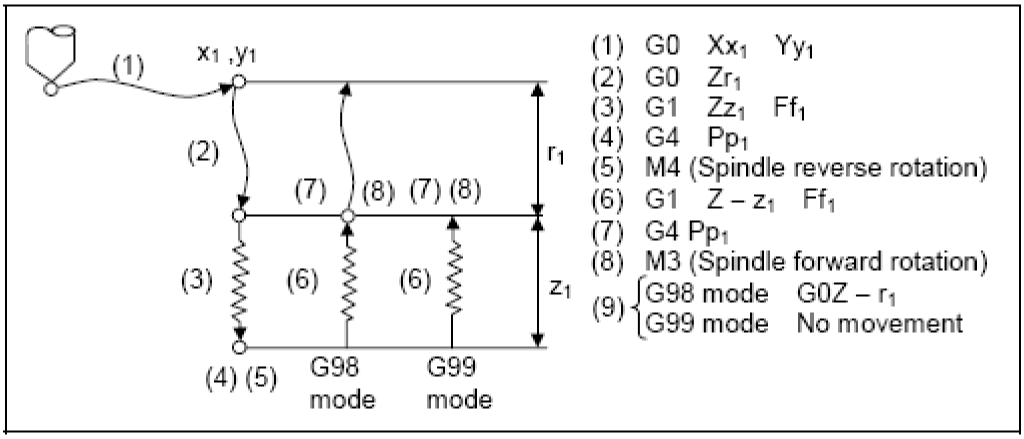


Fig.4.2-13

When $r_2 = 1$, the synchronous tapping mode will be entered, and when $r_2 = 0$, the asynchronous tapping mode will be entered.

When G84 is executed, the override will be canceled and the override will automatically be set to 100%. Dry run is valid when the control parameter "G00 DRY RUN" is on and is valid for the positioning command. If the feed hold button is pressed during G84 execution, and the sequence is at (3) to (6), the movement will not stop immediately, and instead will stop after (6). During the rapid traverse in sequence (1), (2) and (9), the movement will stop immediately.

The operation stops at after the (1), (2) and (9) commands during single block operation.

During the G84 modal, the "Tapping" NC output signal will be output.

During the G84 synchronous tapping modal, the M3, M4, M5 and S code will not be output.

G85 (Boring)

Program

G85 Xx_1 Yy_1 Zz_1 Rr_1 Ff_1 ;

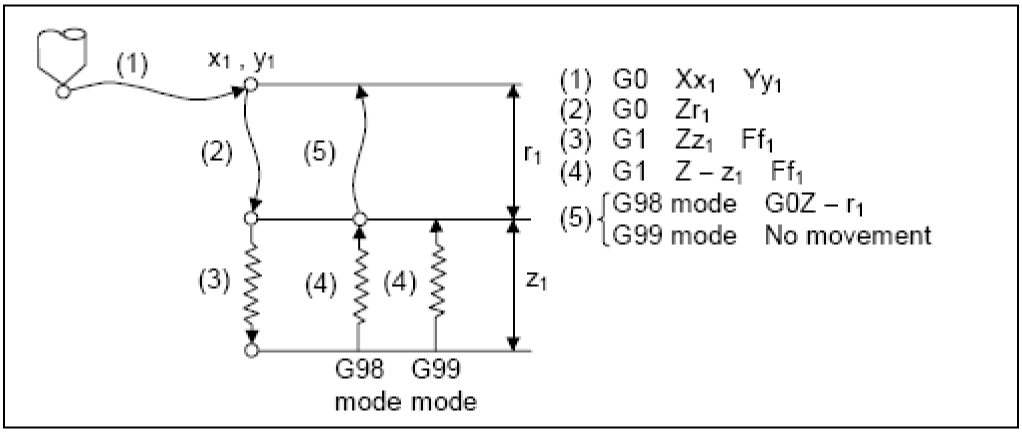


Fig.4.2-14

The operation stops at after the (1), (2), and (4) or (5) commands during single block operation.

(f) G86 (Boring)

Program

G86 Xx_1 Yy_1 Zz_1 Rr_1 Ff_1 Pp_1 ;

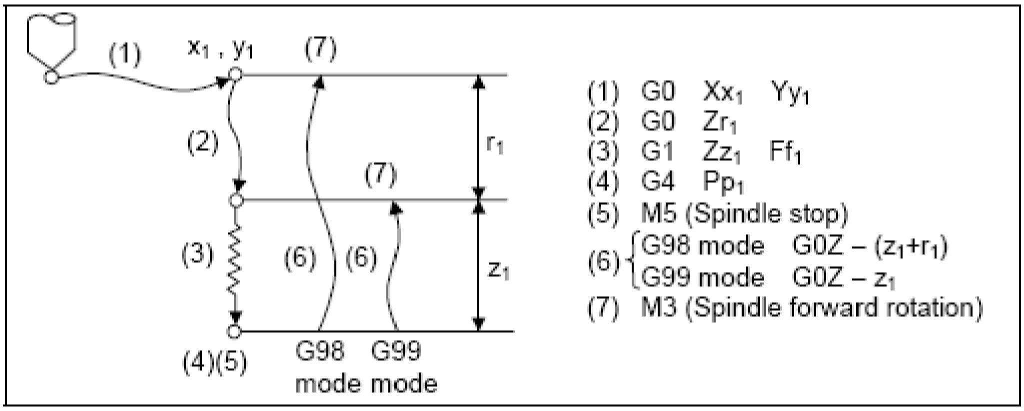


Fig.4.2-15

(g) G87 (Back boring)

Program

G87 Xx_1 Yy_1 Zz_1 Rr_1 Iq_1 Jq_2 Ff_1 ;

(Note) Take care to the z_1 and r_1 designations.

(The z_1 and r_1 symbols are reversed).

There is no R point return.

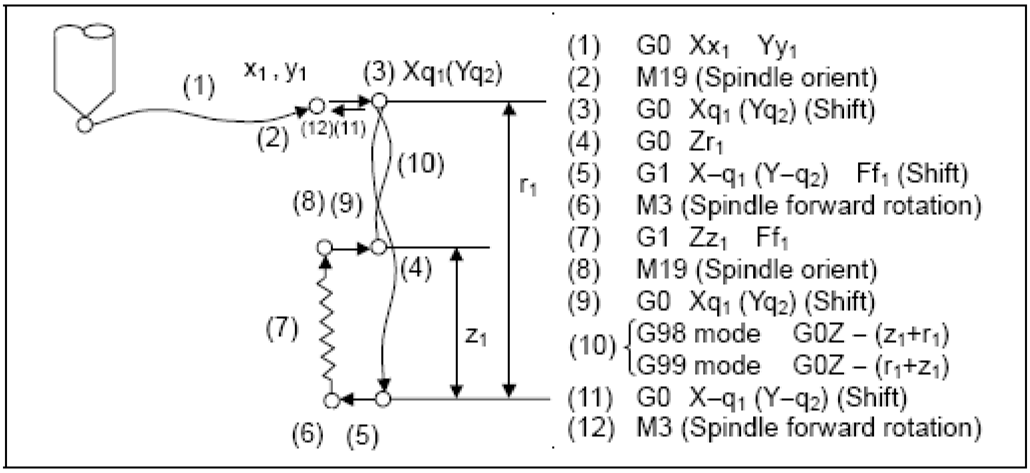


Fig.4.2-16

The operation stops at after the (1), (4), (6) and (11) commands during single block operation.

When this command is used, high precision drilling machining that does not scratch the machining surface can be done.

(Positioning to the hole bottom and the escape (return) after cutting is executed in the state shifted to the direction opposite of the cutter.)

The shift amount is designated as shown below with addresses I, J and K.

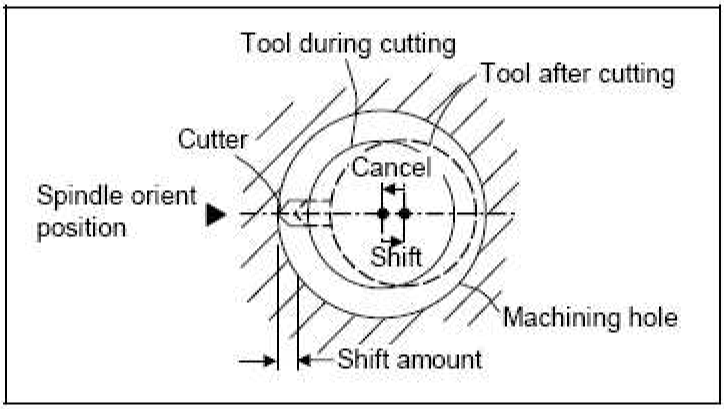


Fig.4.2-17

The shift amount is executed with linear interpolation, and the feed rate follows the F command.

Command I, J, and K with incremental values in the same block as the hole position data.

I, J and K will be handled as modals during the canned cycle.

(Note) If the parameter (#1080 Dril_Z) which fixes the hole drilling axis to the Z axis is set, the shift amount can be designated with address Q instead of I and j. In this case, whether to shift or not and the shift direction are set with parameter #8207 G76/87 No shift and #8208 G76/87 Shift (−). The symbol for the Q value is ignored and the value is handled as a positive value.

The Q value is a modal during the canned cycle, and will also be used as the G83, G73 and G76 cutting amount.

(h) G88 (Boring)

Program

G88 Xx_1 Yy_1 Zz_1 Rr_1 Ff_1 Pp_1 ;

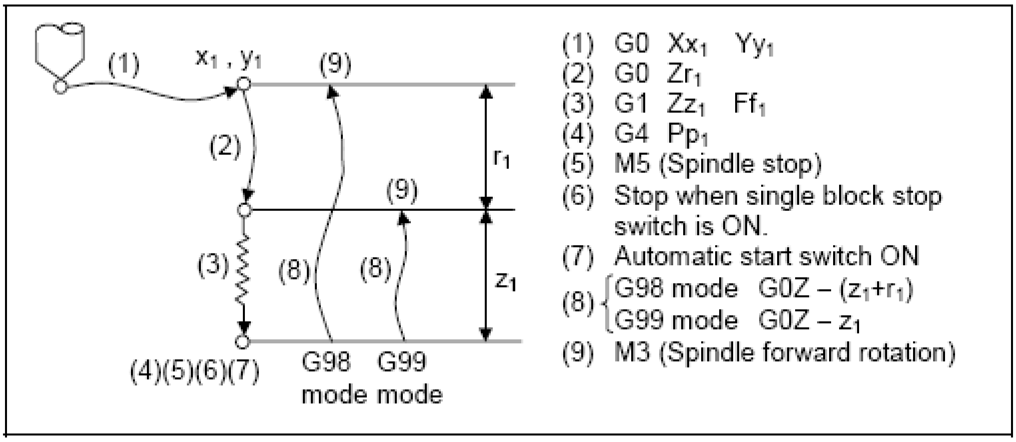


Fig.4.2-18

The operation stops at after the (1), (2), (6) and (9) commands during single block operation.

(i) G89 (Boring)

Program

G89 Xx_1 Yy_1 Zz_1 Rr_1 Ff_1 Pp_1 ;

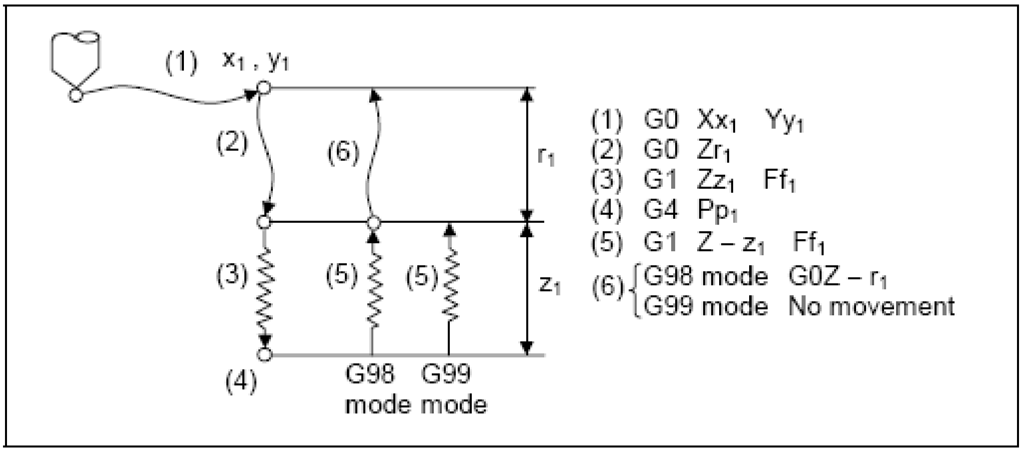


Fig.4.2-19

G73 (Step cycle)

Program

G73 Xx_1 Yy_1 Zz_1 Qq_1 Rr_1 Ff_1 Pp_1 ;

P : Dwell designation

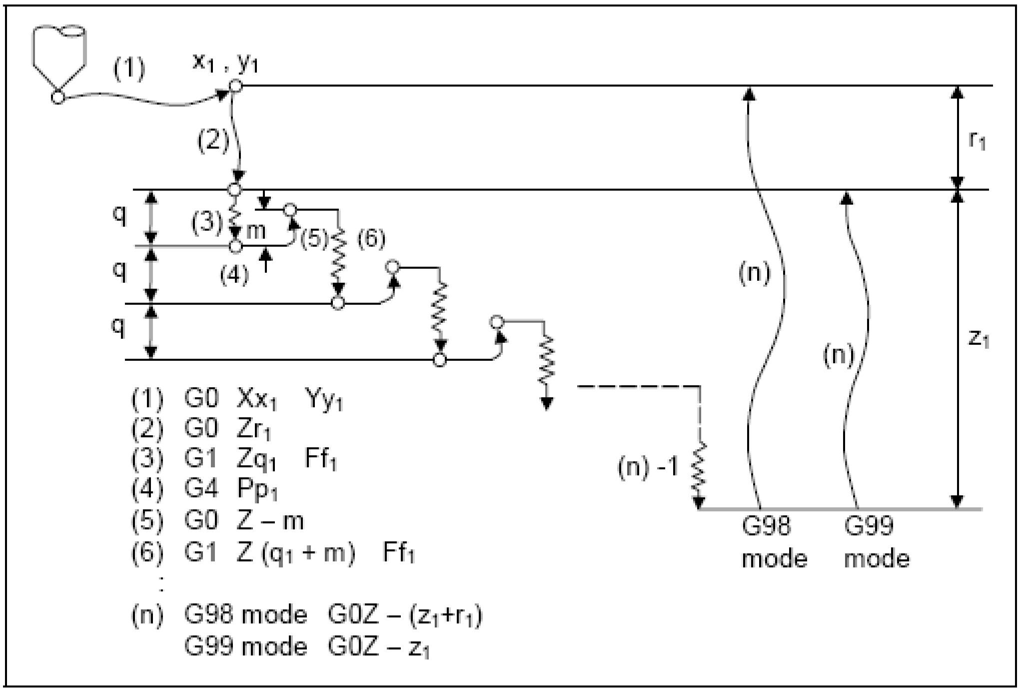


Fig.4.2-20

When executing a second and following cutting in the G73 as shown above, the movement will return several m mm with rapid traverse and then will change to cutting feed.

The return amount m will differ according to the parameter #8012 G73 return.

The operation stops at after the (1), (2) and (n) commands during single block operation.

(k) G74 (Reverse tapping cycle)

Program

G74 Xx_1 Yy_1 Zz_1 Rr_1 Pp_1 Rr_2(or S_1,S_2) ;

P : Dwell designation

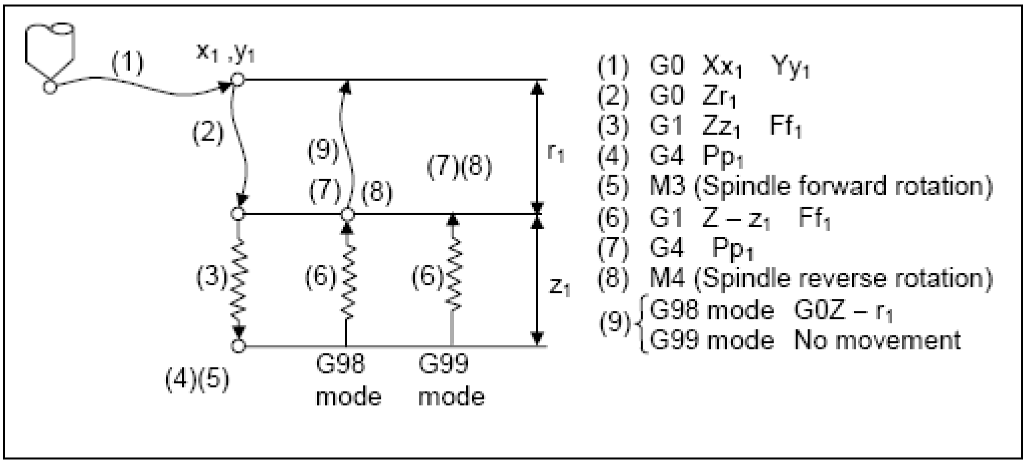


Fig.4.2-21

When $r_2 = 1$, the synchronous tapping mode will be entered, and when $r_2 = 0$, the asynchronous tapping mode will be entered.

When G74 is executed, the override will be canceled and the override will automatically be set to 100%. Dry run is valid when the control parameter #1085 G00Drn is set to "1" and is valid for the positioning command. If the feed hold button is pressed during G74 execution, and the sequence is at (3) to (6), the movement will not stop immediately, and instead will stop after (6). During the rapid traverse in sequence (1), (2) and (9), the movement will stop immediately.

The operation stops at after the (1), (2) and (9) commands during single block operation.

During the G74 and G84 modal, the "Tapping" NC output signal will be output.

During the G74 synchronous tapping modal, the M3, M4, M5 and S code will not be output.

This function allows spindle acceleration/deceleration pattern to be approached to the speed loop acceleration/deceleration pattern by dividing the spindle and drilling axis acceleration/deceleration pattern into up to three stages during synchronous tap.

Refer to the item "d) G84 (Tapping cycle)" for details of multi-stages of the spindle acceleration/deceleration pattern.

(l) G76 (Fine boring)

Program

G76 Xx_1 Yy_1 Zz_1 Rr_1 Iq_1 Jq_2 Ff_1 ;

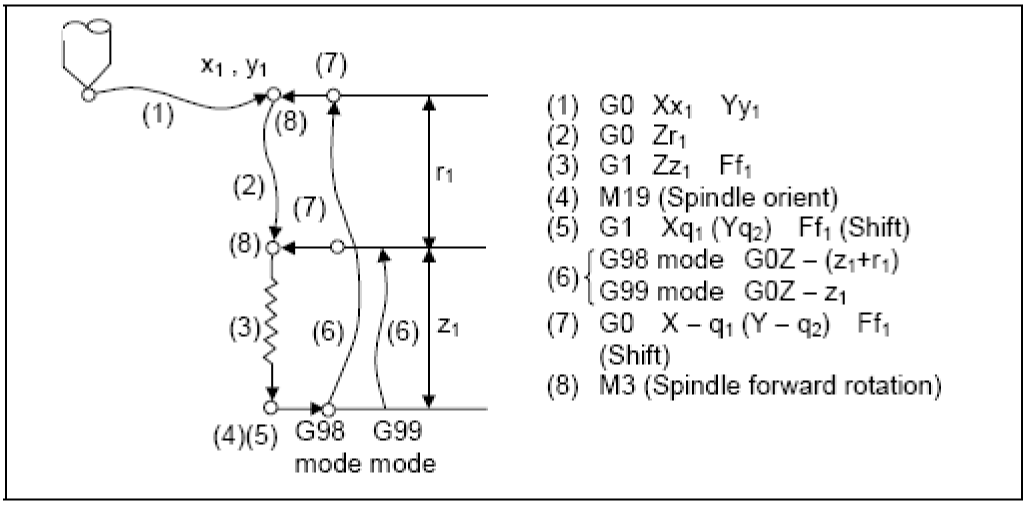


Fig.4.2-22

The operation stops at after the (1), (2) and (7) commands during single block operation. When this command is used, high precision drilling machining that does not scratch the machining surface can be done.

(Positioning to the hole bottom and the escape (return) after cutting is executed in the state shifted to the direction opposite of the cutter.)

4.3 Miscellaneous Functions

Miscellaneous functions (M8-digits BCD)

Function and purpose

The miscellaneous (M) functions are also known as auxiliary functions, and they include such numerically controlled machine functions as spindle forward and reverse rotation, operation stop and coolant ON/OFF.

These functions are designated by an 8-digit number (0 to 99999999) following the address M with this controller, and up to 4 groups can be commanded in a single block.

(Example) G00 Xx Mm1 Mm2 Mm3 Mm4 ;

When five or more commands are issued, only the last four will be valid.

The output signal is an 8-digit BCD code and start signal.

The six commands of M00, M01, M02, M30 M98 and M99 are used as auxiliary commands for specific objectives and so they cannot be used as general auxiliary commands. This therefore leaves 94 miscellaneous functions which are usable as such commands. Reference should be made to the instructions issued by the machine manufacturer for the actual correspondence between the functions and numerical values.

When the M00, M01, M02, and M30 functions are used, the next block is not read into the pre-read buffer due to pre-read inhibiting.

An M function can be specified together with other commands in the same block, and when such a function is specified together with a movement command in the same block, there are two possible sequences in which the commands are executed. Which of these sequences actually applies depends on the machine specifications.

(1) The M function is executed after the movement command.

(2) The M function is executed at the same time as the movement command.
Processing and completion sequences are required in each case for all M commands except M98 and M99.
The 6M functions used for specific purposes will now be described.

Program stop : M00

When the tape reader has read this function, it stops reading the next block. As far as the NC system's functions are concerned, only the tape reading is stopped. Whether such machine functions as the spindle rotation and coolant supply are stopped or not differs according to the machine in question.
Re-start is enabled by pressing the automatic start button on the machine operation board.
Whether resetting can be initiated by M00 depends on the machine specifications.

Optional stop : M01

If the tape reader reads the M01 command when the optional stop switch on the machine operation board is ON, it will stop and the same effect as with the M00 function will apply.
If the optional stop switch is OFF, the M01 command is ignored.

(Example)

```
:
N10 G00 X1000 ;
N11 M01 ;
N12 G01 X2000 Z3000 F600 ;
:
```

Optional stop switch status and operation
Stops at N11 when switch is ON
Next command (N12) is executed without
stopping at N11 when switch is OFF

Program end : M02 or M30

This command is normally used in the final block for completing the machining, and so it is primarily used for tape rewinding. Whether the tape is actually rewound or not depends on the machine specifications.
Depending on the machine specifications, the system is reset by the M02 or M30 command upon completion of tape rewinding and any other commands issued in the same block.
(Although the contents of the command position display counter are not cleared by this reset action, the modal commands and compensation amounts are canceled.)
The next operation stops when the rewinding operation is completed (the in-automatic operation lamp goes off). To restart the unit, the automatic start button must be pressed or similar steps must be taken.
When the program is restarted after M02 and M30 are completed, if the first movement command is designated only with a coordinate word, the interpolation mode will function when the program ends. It is recommended that a G function always be designated for the movement command designated first.

(Note 1) Independent signals are also output respectively for the M00, M01, M02 and M30 commands and these outputs are each reset by pressing the reset key.

(Note 2) M02 or M30 can be assigned by manual data input (MDI). At this time, commands can be issued simultaneously with other commands just as with the tape.

Subprogram call/completion : M98, M99

These commands are used as the return instructions from branch destination subprograms and branches to subprograms.

M98 and M99 are processed internally and so M code signals and strobe signals are not output.

Internal processing with M00/M01/M02/M30 commands

Internal processing suspends pre-reading when the M00, M01, M02 or M30 command has been read. Other tape rewinding operations and the initialization of modals by resetting differ according the machine specifications.

4.4 Program Support Functions

Precautions for using canned cycle

(1) Before the canned cycle is commanded, the spindle must be rotating in a specific direction with an M command (M3 ; or M4 ;).

Note that for the G87 (back boring) command, the spindle rotation command is included in the canned cycle so only the rotation speed command needs to be commanded beforehand.

(2) If there is a basic axis, additional axis or R data in the block during the canned cycle mode, the hole drilling operation will be executed. If there is not data, the hole will not be drilled.

Note that in the X axis data, if the data is a dwell (G04) time command, the hole will not be drilled.

(3) Command the hole machining data (Q, P, I, J, K) in a block where hole drilling is executed. (Block containing a basic axis, additional axis or R data.)

(4) The canned cycle can be canceled by the G00 to G03 or G33 command besides the G80 command. If these are designated in the same block as the canned cycle, the following will occur.

(Where, 00 to 03 and 33 are m, and the canned cycle code is n)

Gm Gn X___Y___Z___R___Q___P___L___F___;

Execute Ignore Execute Ignore Record

Gm Gn X___Y___Z___R___Q___P___L___F___;

Ignore Execute Ignore Record

Note that for the G02 and G03 commands, R will be handled as the arc radius.

(5) If an M function is commanded in the same block as the canned cycle command, the M code and MF will be output during the initial positioning. The next operation will be moved to with FIN (finish signal).

If there is a No. of times designated, the above control will be executed only on the first time.

(6) If another control axis (ex., rotary axis, additional axis) is commanded in the same block as the canned cycle control axis, the canned cycle will be executed after the other control axis is moved first.

(7) If the No. of repetitions L is not designated, L1 will be set. If L0 is designated in the same block as the canned cycle G code command, the hole machining data will be recorded, but the hole machining will not be executed.

(Example) G73X___Y___Z___R___Q___P___F___L0___;

Execute Record only code having an address

(8) When the canned cycle is executed, only the modal command commanded in the canned cycle program will be valid in the canned cycle subprogram. The modal of the program that

called out the canned cycle will not be affected.

(9) Other subprograms cannot be called from the canned cycle subprogram.

(10) Decimal points in the movement command will be ignored during the canned cycle subprogram.

(11) If the No. of repetitions L is 2 or more during the incremental value mode, the positioning will also be incremented each time.

(Example) G91G81X10. Z−50.R−20.F100.L3 ;

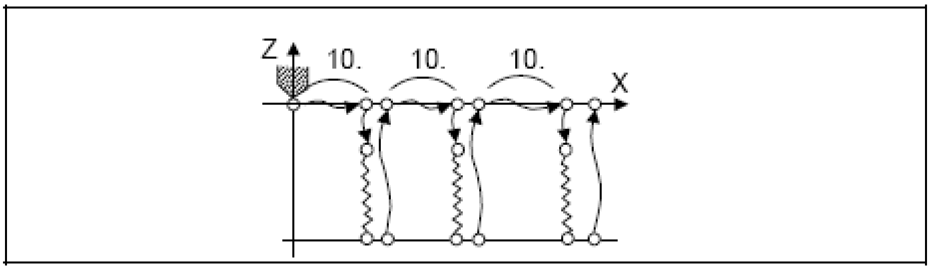


Fig.4.3-1

(12) If the spindle rotation speed value during return is smaller than the spindle speed, the spindle rotation speed value is valid even during return.

(13) If the 2nd and 3rd acceleration/deceleration stage inclinations following the spindle rotation speed and time constants set in the parameters are each steeper than the previous stage's inclination, the previous stage's inclination will be valid.

(14) If the values set in the spindle base specification parameter "stap1-4" (tap rotation speed) and "taps21-24" (synchronous tap changeover spindle rotation speed 2) exceed the maximum rotation speed, the spindle rotation speed will be clamped at the maximum rotation speed.

(15) If the spindle rotation speed during return is not 0, the tap return override value will be invalid.

(16) In a block where the movement direction of any axis reverses as shown below, the servo system load will greatly increase so do not command the in-position width in the machining program.

G0 X100., I10.0 ;

X–200. ;

(17) The in-position width and the position error amount are compared at a set time, so the position error amount at the point to be judged as in-position will be smaller than the commanded in-position width.

(18) Synchronous and asynchronous tap can be selected with the M function.

Special canned cycle; G34, G35, G36, G37.1

Function and purpose

The special canned cycle is used with the standard canned cycle. Before using the special canned cycle, program the canned cycle sequence selection G code and hole machining data to record the hole machining data. (If there is no positioning data, the canned cycle will not be executed, and only the data will be recorded.)

Even after the special canned cycle is executed, the recorded standard canned cycled will be held until canceled.

If the special canned cycle is designated when not in the canned cycle mode, only positioning will be executed, and the hole drilling operation will not be done.

Bolt hole circle (G34)

G34 X x_1 Y y_1 I r J θ K n ;

X, Y :Positioning of bolt hole cycle center. This will be affected by G90/G91.

I :Radius r of the circle. The unit follows the input setting unit, and is given with a positive number.

J :Angle θ of the point to be drilled first. The CCW direction is positive.
(The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001 ° .)

K :No. of holes n to be drilled. 1 to 9999 can be designated, but 0 cannot be designated. When the value is positive, positioning will take place in the CCW direction, and when negative, will take place in the CW direction. If 0 is designated, the alarm P221 Special Canned Holes Zero will occur.

Drilling of n obtained by dividing the circumference by n will start at point created by the Z axis and angle θ. The circumference is that of the radius R centering on the coordinates designated with XX and Y. The hole drilling operation at each hole will hold the drilling data for the standard canned cycle such as G81.

The movement between hole positions will all be done in the G00 mode. G34 will not hold the data even when the command is completed.

Line at angle (G35)

G35 X x1 Y y1 I d J θ K n ;

X, Y :Designation of start point coordinates. This will be affected by G90/G91.

I :Interval d. The unit follows the input setting unit. If d is negative, the drilling will take place in the direction symmetrical to the point that is the center of the start point.

J :Angle θ. The CCW direction is positive.
(The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001 ° .)

K :No. of holes n to be drilled. 1 to 9999 can be designated, and the start point is included.

Using the position designated by X and Y as the start point, the Zn holes will be drilled with interval d in the direction created by X axis and angle θ. The hole drilling operation at each hole position will be determined by the standard canned cycle, so the hole drilling data (hole machining mode and hole machining data) must be held beforehand. The movement between hole positions will all be done in the G00 mode. G35 will not hold the data even when the command is completed.

Arc (G36)

G36 X x1 Y y1 I r J θ P Δθ K n ;

X, Y :Center coordinates of arc. This will be affected by G90/G91.

I :Radius r of arc. The unit follows the input setting unit, and is given with a positive No.

J :Angle θ of the point to be drilled first. The CCW direction is positive. (The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001 ° .)

P :Angle interval Δθ. When the value is positive, the drilling will take place in the CCW direction, and in the CW direction when negative. (The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001 ˚.)

K :No. of holes n to be drilled. 1 to 9999 can be designated.

The n holes aligned at the angle interval Δθ will be drilled starting at point created by the X axis and angle θ. The circumference is that of the radius R centering on the coordinates designated with XX and Y. As with the bolt hole circle, the hole drilling operation at each hole will depend on the standard canned cycle.

The movement between hole positions will all be done in the G00 mode. G36 will not hold the dataeven when the command is completed.

Grid (G37.1)

G37.1 X x1 Y y1 I Dx P nx J Dy K ny ;

X, Y :Designation of start point coordinates. This will be affected by G90/G91.

I :Interval Dx of the X axis. The unit will follow the input setting unit. If Dx is positive, the interval will be in the forward direction looking from the start point, and when negative, will be in the reverse direction looking from the start point.

P :No. of holes nx in the X axis direction. The setting range is 1 to 9999.

J :Interval Dy of the Y axis. The unit will follow the input setting unit. If Dy is positive, the interval will be in the forward direction looking from the start point, and when negative, will be in the reverse direction looking from the start point.

K :No. of holes ny in the Y axis direction. The setting range is 1 to 9999.

The nx points on a grid are drilled with an interval Δx parallel to the X axis, starting at the position designated with X, Y. The drilling operation at each hole position will depend on the standard canned cycle, so the hole drilling data (hole machining mode and hole machining data) must be held beforehand.

The movement between hole positions will all be done in the G00 mode. G37.1 will not hold the data even when the command is completed.

Subprogram control; M98, M99, M198

Calling subprogram with M98 and M99 commands

Function and purpose

Fixed sequences or repeatedly used patterns can be stored in the memory as subprograms which can then be called from the main program when required. M98 serves to call subprograms and M99 serves to return operation from the subprogram to the main program. Furthermore, it is possible to call other subprograms from particular subprograms and the nesting depth can include as many as 8 levels.

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