Nanjing Swan Software Technology Co., Ltd.
Version 05/2007
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., overall design of product, technique consultation, quadratic research (second development service). We also develop CAD&CAM software, numerical-controlled system, and the technology of surface simulation. Besides, we provide UG-software-based quadratic research service, which can help companies 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
05/2007
CONTENTS

CHAPTER 1  SUMMARY OF SWAN 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 SWAN NC SIMULATION SOFTWARE ..................................... 4
  2.1 STARTUP INTERFACE OF THE SOFTWARE .................................................................. 4
    2.1.1 STARTUP INTERFACE OF PROBATIONAL VERSION .......................................... 4
    2.1.2 STARTUP INTERFACE OF NETWORK VERSION ............................................... 4
    2.1.3 SINGLE MACHINE VERSION STARTUP INTERFACE ....................................... 6
  2.2 SETUP OF TOOLBAR AND MENU ............................................................................ 6
  2.3 FILE MANAGEMENT MENU ..................................................................................... 8
    2.3.1 MACHINE PARAMETER .................................................................................. 9
    2.3.2 CUTTER MANAGEMENT .................................................................................. 10
    2.3.3 WORKPIECE PARAMETER AND ACCESSORY ............................................... 12
    2.3.4 RAPID SIMULATIVE MACHINING .................................................................. 15
    2.3.5 WORKPIECE MEASUREMENT ....................................................................... 15
    2.3.6 REC PARAMETER SETUP .............................................................................. 16
    2.3.7 WARING MESSAGE ....................................................................................... 16

CHAPTER 3  EZMotion-NC OPERATION ............................................................................. 19
  3.1 EZMotion-NC MACHINE PANEL OPERATION ......................................................... 19
  3.2 EZMotion-NC SYSTEM OPERATION ......................................................................... 20
    3.2.1 KEYSTOKE INTRODUCTION .......................................................................... 20
    3.2.2 MANUAL OPERATION OF VIRTUAL NC MACHINE ......................................... 20

CHAPTER 4  EZMotion programme ................................................................................. 24
  4.1 Position .................................................................................................................. 24
  4.2 G Commands ......................................................................................................... 26
  4.3 Miscellaneous Functions ....................................................................................... 45
  4.4 Program Support Functions .................................................................................. 47
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.
(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.
(6) Put the output: Z, Y, X axes workpiece nullpoint into G54~G59.
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.
★ tool-change 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.
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](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

(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 as NC file)
Save file (such as NC file)
Save as
Machine parameter
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.

![Image of Select File to Save](image1)

![Image of Open](image2)
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](image)

![Fig.2.3-4](image)

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

![Fig.2.3-5](image)
Adjust “Processing Drawing Display Acceleration” and “Display Precision” to gain appropriate speed of service of simulation software.

![Operation Settings](image)

**Fig.2.3-6**

b. Display color:

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

![Color Settings](image)

**Fig.2.3-7**

### 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 holder, 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). bilmpe 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 diago box “add tool”, as the fowing graph show:

![Fig. 2.3-10](image)

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

![Fig. 2.3-11](image)

(3) Choose the tool needed in diago “tool”, 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 origin of workpiece X, Y, Z.
(3) Select changing machining origin, 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
Workpiece placement

(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..

![Dimension and...](image)

Fig. 2.3-18

### 2.3.6 REC PARAMETER SETUP

Three modes of REC area selection, setup as

![Record Parameters](image)

Fig. 2.3-19

### 2.3.7 WARING MESSAGE

- ![Output current message files](image)
- ![Output all message files](image)
- ![Last day message](image)
- ![Next day message](image)
- ![Delete current message files](image)
- ![Parameter setup](image)

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 OPERATION 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 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.

![Machine panel](image)

**Fig. 3.1-1** E60 (milling machine) panel

![Machine panel](image)

**Fig. 3.1-2** E68T (lathe) panel

- **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.

**Program running startup.**

**Program running stop.**

**Program running M00 stop.**
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:

3.2.1 KEYSTICKE INTRODUCTION

- CAN: Modifier. Erase data which is in input region.
- DELET: Delete key. Delete the data curor pointing; Or delete a NC program or all the programs.
- INSRT: Insert key. Insert the area behind curor with data which is in the input region.
- EOB: 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 number “7”. Search program numbered “O7”.

(4) Press cursor to input number “7”. Search program numbered “O7”.

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.

**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 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.

**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 searching a specified 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 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.
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 αα

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.

N1 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.

N2 G90 G01 X200. Y50. F100;
N2 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.

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

(G90) N3 X100. Y100.;

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

(G91) N3 X–100. Y50.;

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

![Diagram](image)

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.

![Diagram](image)

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
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 P1 → P2 → P3 → P4 → P1 at 300 mm/min feedrate P0 → P1 is for tool positioning
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 cannot 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
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.
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 Ff ; (Normal lead thread cutting commands)
Zz : Thread cutting direction axis address (X, Y, Z, α) and thread length
Ff : Lead of long axis (axis which moves most) direction.
Qq : Thread cutting start shift angle, (0 to 360°)

G33 Zz Ee ; 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.
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).

\[ FC = F \times N \times OVR \]  
(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).

<table>
<thead>
<tr>
<th>Command mode</th>
<th>Metric input</th>
<th>Inch input</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.001mm</td>
<td>0.0001inch</td>
</tr>
<tr>
<td>Command address</td>
<td>Feed per minute</td>
<td>Feed per rotation</td>
</tr>
<tr>
<td>Command address</td>
<td>F (mm/min)</td>
<td>E (mm/rev)</td>
</tr>
<tr>
<td>Minimum command unit</td>
<td>1 (= 1.00), ( \frac{1}{1} = 1.00 )</td>
<td>1 (= 0.01), ( \frac{1}{1} = 1.00 )</td>
</tr>
<tr>
<td>Command range</td>
<td>0.01 to 100000.00</td>
<td>0.001 to 999.999</td>
</tr>
</tbody>
</table>
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 

<table>
<thead>
<tr>
<th>Same direction</th>
<th>Reverse direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>G0lpg: 0</td>
<td></td>
</tr>
<tr>
<td><img src="Fig.4.2-7" alt="Diagram" /> G1 G0</td>
<td><img src="Fig.4.2-7" alt="Diagram" /> G1 G0</td>
</tr>
<tr>
<td>G0lpg: 1</td>
<td></td>
</tr>
<tr>
<td><img src="Fig.4.2-7" alt="Diagram" /> G1 G0</td>
<td><img src="Fig.4.2-7" alt="Diagram" /> G1 G0</td>
</tr>
</tbody>
</table>

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.

<table>
<thead>
<tr>
<th>Input setting unit</th>
<th>Command range based on address X</th>
<th>Command range based on address P</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01/0.001mm</td>
<td>0.001 to 999999.999 (s)</td>
<td>1 to 999999999 (s) x0.001 s</td>
</tr>
<tr>
<td>0.0001inch</td>
<td>0.001 to 999999.999 (s)</td>
<td>1 to 999999999 (s) x0.001 s</td>
</tr>
</tbody>
</table>

(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

```plaintext
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;**

---

36
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 Hh1 ; z + (λh1) Offset in + direction by tool offset amount

G44 Zz Hh ; z − (λh) Offset in − direction by tool offset amount

G49 Zz ; z −(+ (λh1) Offset amount cancel.

λh1 : Offset amount for offset No. h1

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 ;
Program Support Functions

(a) G81 (Drilling, spot drilling)
Program
G81 Xx1 Yy1 Zz1 Rr1 Ff1 ;

(b) G82 (Drilling, counter boring)
Program
G82 Xx1 Yy1 Zz1 Rf1 Ff1 Pp1 ;
P : Dwell designation
(c) G83 (Deep hole drilling cycle)

Program

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 ;

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

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 q1>m.
The operation stops at after the (1), (2) and (n) commands during single block operation.

(d) G84 (Tapping cycle)

Program

G84 Xx Yy Zz Rr Ff Pp Rr (or S1, S2)

P : Dwell designation

![Diagram](image)

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 Yy Zz Rr Ff

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

(f) G86 (Boring)
Program
G86 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1;

![Diagram of G86](image)

Fig.4.2-15

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.

(g) G87 (Back boring)
Program
G87 Xx1 Yy1 Zz1 Rr1 Iiq1 Jiq1 Ff1;

(Note) Take care to the z1 and r1 designations.
(The z1 and r1 symbols are reversed).
There is no R point return.

![Diagram of G87](image)

Fig.4.2-16
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.

(Notice) 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.

(b) G88 (Boring)
Program
G88 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1;

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

(i) G89 (Boring)
Program
G89 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1;
G73 (Step cycle)

Program

G73 Xx1 Yy1 Zz1 Qq1 Rr1 Ff1 Pp1;

P : Dwell designation

When executing a second and following cutting in the G73 as shown above, the movement will return several 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 Xx1 Yy1 Zz1 Rr1 Pp1 Rr2 (or S1,S2) ;
P : Dwell designation

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 G00Drm 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 Xx1 Yy1 Zz1 Rr1 Iq1 Jq2 Ff1 ;
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.

---

**Fig.4.2-22**

![Diagram of Fig.4.2-22](image-url)

(1) G0 Xx₁ Yy₁
(2) G0 Zz₁
(3) G1 Zz₁ Ff₁
(4) M19 (Spindle orient)
(5) G1 Xq₁ (Yq₂) Ff₁ (Shift)
(6) G98 mode G0Z - (zi₁+r₁)
(7) G99 mode G0Z - zi₁
(8) G0 X - q₁ (Y - q₂) Ff₁
(9) M3 (Spindle forward rotation)
(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 Y y 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 YY. 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 x 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 x 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 the 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 YY. 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 data even 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.