4. Code & Programming

ISO code is the language used by numerical control machine all over the world, which is also adopted by this controller. In this chapter, the basic principle of numerical control code for the controller will first be introduced, and then several programming examples will be given. After carefully reading the contents of this chapter, you will be able to do some actual programming by yourself.

4.1. ISO code

4.1.1. Introduction

ISO code file comprises several instruction lines. Each instruction line consists of one or several instructions. According to their functions, instructions can be divided into several categories such as preparation, move and auxiliary. Its structure is as follows:

instruction = instruction code + instruction data

The instruction code defines its function and is represented by a letter from A to Z. The instruction code is followed by the instruction data, whose meaning is determined by the instruction code.

4.1.2. Instruction code

The following are the instruction codes used by the controller and their definitions.

Table 4-1   Instruction code and definition

Instruction code	Definition
A	Angle of taper
B	Radius of reference Surface
E	Machining condition
F	Pause time
G	Preparation
H	Serial number for offset
I, J	Center of arc
L	Times of calling subprogram
M	Auxiliary

N	Serial number
P	Subprogram number that has been called
R	Rotary angle
S	Reduction/enlargement ratio
X,Y,U,V,Z	Moving distance of axis

4.1.3. Instruction code and instruction data

a. A represents the angle of taper to be processed.

A number from –999999.9999 to +999999.9999 can be entered. If the number contains decimal point, the unit will be degree. If the number is an integer, the unit will be 0.001 degree. For instance, A1000 represents one degree; A1.0 also represents one degree.

b. B represents the radius of reference surface.

Number from 0 to 99999.999 can be entered. In metric system, if the number contains decimal point, the unit will be mm; if the number is an integer, the unit will be μm. In British system, if the number contains decimal point, the unit will be inch; if the number is an integer, the unit will be 0.00001 inch. 1 inch = 25.4mm

c. E represents machining condition.

An unsigned 4 digit integer larger than 0 can be entered, for instance, E0001. The machining condition must be stored in the system's technological data library, otherwise the system will display the fault information.

d. F represents the pause time.

An unsigned integer less than 1000 can be entered (in sec).

e. G represents preparation.

The corresponding instruction code can be entered.

f. H represents the serial number for offset.

An unsigned integer larger than 0 can be entered, for instance, H001.

g. I and J represent the coordinate of the center of arc.

A number from –999999.999 to +999999.999 can be entered.

h. L represents the times the subprogram has been called.

An unsigned integer larger than 0 and less than 1000 can be entered.

i. M represents auxiliary.

An unsigned integer can be entered.

j. N represents serial number.

An unsigned integer from 0 to 10000 can be entered.

k. P represents the serial number of subprogram that has been called.

An unsigned integer from 0 to 10000 can be entered.

l. R represents the rotary angle.

A number from –999999.999 to +999999.999 can be entered.

m. S represents the reduction/enlargement ratio.

A number from 0.1 to 10.000 can be entered.

n. X, Y, U and V represent the moving distance of axes.

A number from –999999.999 to +999999.999 can be entered. For X, Y, U, V, I and J, if the number contains decimal point, the unit will be mm in metric system and inch in British system; if the number is an integer, the unit will be μm in metric system and 0.00001 inch in British system. 1 inch = 25.4mm

4.1.4. Instruction line

Each numerical control file comprises several instruction lines. Each instruction line may consist of the following kinds of instructions:

a. N serial number

b. E code

c. Part of the M codes: M00

d. Part of the G codes:

G05、G06、G08、G09

G04

G11、G12

G16

G20、G21

G26、G27

G29、G36、G37

G30、G31

G40~G42、G50~G52、G83、G90、G91

e. Part of the G codes:

G00~G03，G92

f. Part of the M codes: M98、M99、M02

Each instruction line must be in accordance with the following rules:

a. The instructions in the instruction line are executed from a to e. (Instruction priority decreases from a to e.)

b. If there are several instructions with same function that belong to the same category, the last one will be executed.

c. Code listed in f can be used only once in an instruction line.

d. Code listed in b~f cannot be used together with that listed in g in one instruction line.

e. When rotation, scale and symmetry appear in the same instruction line, the execution sequence is rotation, scale to symmetry. In the same instruction line,

"(" can be used to express the beginning of a note, ") " the end of a note and ";
" the neglecting of the contents after "; ".

f. When an instruction line contains " / ", it will not be executed if G11 has been defined. It will still be executed if G12 has been defined or G11 has not been defined.

4.1.5. ISO file

Before cutting, you must draw up the ISO file according to the workpiece drawings. Each ISO file comprises three sections:

a. Head of file

Head of file must include the machining parameters and offset.

b. Main program

Main program consists of several instruction lines, ended with M02.

c. Subprogram block

Subprogram block must contain the subprogram called by the main program, but the subprogram cannot be a nested one.

4.2. G code

Table 4-2   G Code

Code	Function	Default
G00	Rapid positioning	
G01	Straight insertion	
G02	clockwise arc insertion	
G03	Counterclockwise arc insertion	
G04	Pause	
G05	Conversion of X axis	G09
G06	Conversion of Y axis	
G08	Simultaneous conversion of X and Y	
G09	conversion of X and Y cancelled	
G11	jump allowed	G12
G12	jump cancelled	
G16	reduction/enlargement	
G20	unit in British system	G21
G21	unit in metric system	
G26	Rotation allowed	G27
G27	Rotation cancelled	
G29	reference point specified	
G30	Clear function cancelled	G30
G31	Clear function allowed	

G36	"Radius of reference surface specified" cancelled	G36
G37	Radius of reference surface specified	
G40	Offset cancelled	G40
G41	left offset	
G42	right offset	
G50	taper cancelled	G50
G51	taper tilting to left	
G52	taper tilting to right	
G54~ G61	Define multiple reference frames	G54
G83	recording machining time	
G90	programming in absolute mode	G90
G91	programming in incremental mode	
G92	Defining the initial point of coordinate	
G98	Defining the start point of machining in absolute coordinate	
G140	Upper and lower irregular shape cancelled	G140
G141	upper and lower irregular shape	

4.2.1. Introduction of G code

G codes can be divided into following categories according to their functions:

a. Move: G00、G01、G02、G03

b. Pause: G04

c. Symmetry: G05、G06、G08、G09

d. Jumping: G11、G12

e. Reduction/enlargement: G16

f. Unit: G20、G21

g. Rotation: G26、G27

h. Reference point: G29

i. Clear function: G30, G31

j. Taper radius: G36、G37

k. Offset: G40、G41、G42

l. Taper: G50、G51、G52

m. Recording: G83

n. Programming mode: G90、G91

o. Initial point: G92、G98、G54~G61

p. Upper and lower irregular shape: G140、G141

The instructions listed in Pause and Recording are effective only in the current instruction line. Other instructions will be always effective until that belonging to the same category appears.

4.2.2. Move code

a. G00: rapid positioning

Format: G00　Xx Yy Uu Vv

Turning off the machining power supply, rapidly moving from the current point to the terminal point.

u and v are the offset of U and V table relative to X and Y table respectively.

In the incremental mode, x and y are the coordinate of terminal point relative to the current point.

In the absolute mode, x and y are the coordinate of terminal point.

b. G01: straight insertion

Format: G01 Xx Yy Uu Vv

Running from the current point to the terminal point in the machining speed. X, Y, U and V move simultaneously to perform the straight insertion. The meaning of x, y, u and v are the same as G00.

c. G02: clockwise arc insertion

Format: G02 Xx Yy Ii Jj

Running from the current point to the terminal point. i and j are the coordinate of center relative to the current point.

In the incremental mode, x and y are the coordinate of terminal point relative to the current point.

In the absolute mode, x and y are the coordinate of terminal point.

When the arc insertion is to be carried out, you have to compare the current radius with the terminal one. If their difference is larger than 2μm, the system will show you the hint that the code radius is wrong.

d. G03: counterclockwise arc insertion

It is similar to G02, but in the reverse rotating direction.

Examples for insertion instruction line:

Absolute programming mode　　G90G92X10.Y10.

G01X4.Y3.

The above cutting path and direction are shown in Fig.4-1. Numbers in the brackets are the coordinates of initial point and terminal point.
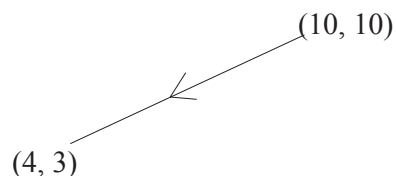
(10, 10)

(4, 3)

Fig.4-1

Incremental programming mode   G91G92X10.Y10.

G01X4.Y3.

The above cutting path and direction are shown in Fig.4-2. Numbers in the brackets are the coordinates of initial point and terminal point.
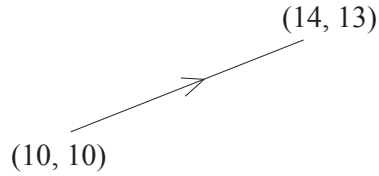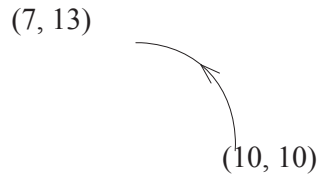
(14, 13)

(10, 10)

Fig.4-2

Absolute programming mode   G90G92X10.Y10.

G03X7.Y13.I-3.J0

The above cutting path and direction are shown in Fig.4-3. Numbers in the brackets are the coordinates of initial point and terminal point.

(7, 13)

(10, 10)

Fig.4-3

Incremental programming mode   G91G92X10.Y10.

G03X-3.Y3.I-3.J0

The above cutting path and direction are shown in Fig.4-4. Numbers in the brackets are the coordinates of initial point and terminal point.
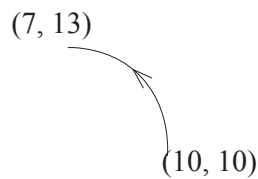
(7, 13)

(10, 10)

Fig.4-4

4.2.3. Pause

G04: pause

Format: G04Ff, f : 0 ~ 999 sec

The table's move will be stopped temporarily for f seconds.

Example: G04F10; the table's move will be stopped for 10 seconds.

4.2.4. Symmetry

The default is G09.

Symmetry refers to the sign reversion of instruction value when axes are moving. Meanwhile, the offset direction and taper direction will also be changed accordingly.

a. G05: symmetry about Y axis, i.e. conversion of X axis

Format: G05

b. G06: symmetry about X axis, i.e. conversion of Y axis

Format: G06

c. G08: symmetry about origin, i.e. conversion of X and Y axes

Format: G08

d. G09: symmetry cancelled

Example for G05:

```
G91G92X0Y0
M98P1000L1
G05
M98P1000L1
M02
N1000
G01X5.0Y5.0
Y5.0
X5.0
G02X0Y-5.0I0J-2.5
G01X-4.0
Y-5.0
M99
```

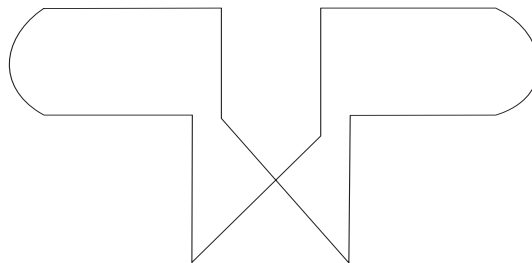The graphic drawn according to the above ISO file is shown in Fig.4-5.

Fig.4-5

Example for G06:
  G91G92X0Y0
  M98P1000L1
  G06
  M98P1000L1
  M02
  N1000
  G01X5.0Y5.0
  X5.0
  Y5.0
  X-10.0
  Y-10.0
  M99

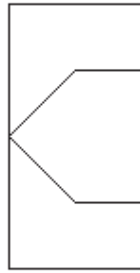The graphic drawn according to the above ISO file is shown in Fig.4-6.

Fig.4-6

Example for G08:
  G91G92X0Y0
  M98P1000L1
  G08
  M98P1000L1
  M02
  N1000
  G01X5.0Y5.0
  Y5.0
  X5.0
  G02X0Y-5.0I0J-2.5
  G01X-4.0
  Y-5.0

G01X5.0Y-5.0
M99

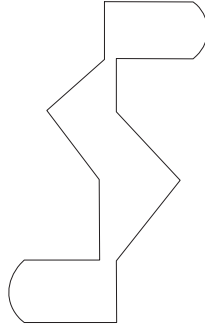The graphic drawn according to the above ISO file is shown in Fig.4-7.

Fig.4-7

4.2.5. Jump

The default is G12
a. G11: jump allowed
   Format: G11
   The instruction line containing "/" will not be executed.
b. G12: jump cancelled
   Format: G12
   The instruction line containing "/" will still be executed.
   Example:
   G92X0Y0
   G91G01X10.0
   G11
   /G41 H000   (A)
   G01Y10.0
   G12   (B)
   /G01X10.0
   G01Y-10.0
   M02
In the above ISO file, A line will not be executed and B line will still be executed.
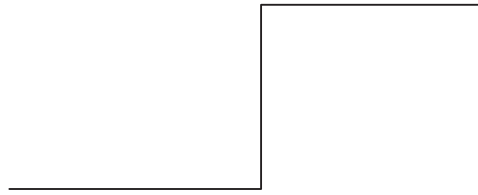The graphic drawn according to the above ISO file is shown in Fig.4-8.

Fig.4-8

4.2.6. Reduction/enlargement

G16: reduction/enlargement according to the scale

Format: G16Ss

The graphic will be reduced or enlarged by s (0.1 ~ 10.0) from the defined position. Example: 15.000*1.33333=19.99995≈20.000 (G16S1.33333)

4.2.7. Unit

The default is G21

a. G20: British system

Format: G20

In British system, if the number contains decimal point, the unit will be inch, otherwise the unit will be 0.00001 inch. 1 inch = 25.4mm

b. G21: metric system

Format: G21

In metric system, if the number contains decimal point, the unit will be mm, otherwise the unit will be μm.

Example:

G91

G21

G01X10.0   ;   X moves 10mm.

G01X1000   ;   X moves 1mm.

G20

G01X1.0   ;   X moves 2.54cm

G01X10000   ;   X moves 0.254cm

4.2.8. Rotation

The default is G27.

a. G26: graphic rotation allowed.

Format: G26Rr

The graphic will rotate r degrees from the defined position.

Format: G26

The rotation angle of graphic will increase by r degrees from the defined position.

b. G27: graphic rotation cancelled

Format: G27

Example:

G92X0Y0

G91 G01X10.0Y10.0

G26R90

G01X10.000

G26

G01X10.0

G26

G01X10.0

M02

The graphic drawn according to the above ISO file is shown in Fig.4-9.

Fig.4-9

4.2.9. Reference point

G29: reference point specified

Format: G29

The reference point is specified. The wire-threading will be conducted at the reference point after wire break.

4.2.10. Clear function

The default is G30.

a. G30: "clear function" is cancelled.

Format: G30

b. G31: "clear function" is allowed.

Format: G31

When machining to a corner, the controller automatically machines a short distance along the rightabout of the angle bisector of the corner, then return to the corner along the former track.

If two lines connect, the track of clear is the rightabout of the angle bisector. If a line connects with an arc, the rightabout of the bisector of the angle in the line and the tangent of the arc is the clear track. In the Run\Option menu, user can specify the length and angle of clear.

4.2.11. Taper radius

The default is G36.

a. G36: "radius of reference surface specified" is cancelled.

Format: G36

b. G37:   radius of reference surface specified

Format: G37Bb

During the machining of taper, the arc radius in the reference surface corresponding to that in the programming surface will be specified as b.   If b is omitted, the arc radius in the reference surface will be considered to be equal to that in the programming surface (equal-radius mode).
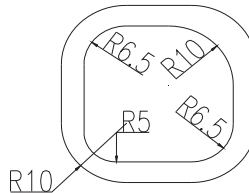
Example:

Fig.4-10

```
G92X0Y0
G91
G01X10000Y0
G52 A5000
G01X0Y15000
G02X10000Y10000I10000J0
G37
G01X30000Y0
G02X10000Y-10000I0J-10000
```

G01X0Y-30000

G36

G02X-10000Y-10000I-10000J0

G37 B5000

G01X-30000Y0

G02X-10000Y10000I0J10000

G01X0Y15000

G36

G50

G01X-10000Y0

M02

If HB = 0, HC = 40 during the slope cutting, the graphic and the reference surface drawn according to the above ISO file are shown in Fig.4-10.

4.2.12. Offset

In order to acquire the dimension specified by the program, the path of wire center relative to the programmed path will be adjusted to the right or to the left by taking into account the wire radius and discharge gap (See Fig.4-11).
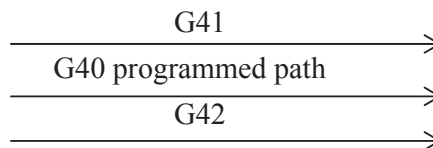

Fig.4-11

The default is G40

a. G40: offset cancelled

Format: G40

b. G41: left offset from the machining path

Format: G41 Hxxx, where xxx is the code of predetermined offset.

c. G42: right offset from the machining path

Format: G42 Hxxx, where xxx is the code of predetermined offset.

4.2.13. Taper

The default is G50.

a. G50: taper cancelled

Format: G50

b. G51: tilting to left

Format: G51Aa

Looking along the machining path, the wire electrode tilts to the left by "a" degrees.

c. G52: tilting to right

Format: G52Aa

Looking along the machining path, the wire electrode tilts to the right by "a" degrees.

4.2.14. Recording

G83: recording the machining time

Format: G83

Recording the machining time from the initial point to the end of current instruction.

4.2.15. Programming mode

The default is G90.

a. G90: absolute programming mode

Format: G90

b. G91: relative programming mode

Format: G91

4.2.16. Start point

a. G92 defines the workpiece coordinate value

Format: G92 Xx Yy Uu Vv

Define the current values as the coordinate (X, Y, U, V) of the operating coordinate system. Default values are (0, 0, 0, 0).

b. G98 specifies that machining is start from absolute coordinate (x, y). If before the machining the absolute coordinate is (x_0, y_0), controlling machining moves to (x, y) and then start machining.

Format: G98XxYy

Use this function and [,], { and } can conveniently carry out machining several moulds in a program. In the program user use [ ] specify the moulds that will not be machined and use { } specify the moulds that need machining. Between two numbers of the moulds the blank is used to separate a mould from another and the "," is used to means all the moulds between the two numbers is included. For example: [10, 13 17 19] is means the moulds which numbers are from 10 to 13 and 17 and 19 will not be

machined. {10, 13 17 19} means the moulds which numbers are from 10 to 13 and 17 and 19 will be machined.

Note: [ ] and { } cannot appear in a ISO file at the same time. Controller automatically specifies the number of mould. When a G00 appears in an ISO file, controller will machine the next mould. The number of machining mould is displaying below the Z coordinate on the screen.

c. G54~G61 define multiple reference frames

Format: G54↙ G92...↙ ...↙ G55↙ G92...↙ ...

Define the current point as the start point in a new reference frame. All following coordinates are in this new reference frame. The default is G54.

4.2.17. Upper and lower irregular shape

The default is G140.

a. G140: upper and lower irregular shape cancelled

Format: G140

b. G141: upper and lower irregular shape allowed

Format: G141

Example:

G91G92X0Y0

G01Y-6000

M00

G01 Y-200

M98P100L1

M02

N100

G01Y-1000

G141

Fig.4-12

G02X10.Y-10.I0J-10.	:	G01X10.Y-10.
X-10.Y-10.I-10.	:	X-10.Y-10.
X-10.Y10.J10.	:	X-10.Y10.
X10.Y10.I10.	:	X10.Y10.

G140

M99

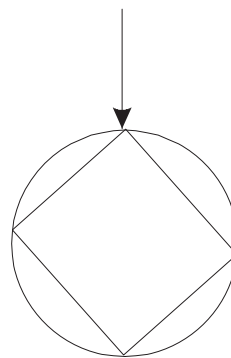According to the above ISO file, the resulting workpiece will have an upper square and a lower circle (See Fig.4-12).

4.3. M code

4.3.1. M02: termination of machining

After the machining is terminated, the operations of wire, fluid, pulse power, and motors will all be stopped.

4.3.2. M00: pause

Pulse power, fluid, and wire are turned off.
After "Start" is clicked, the system will automatically resume the machining
If "Stop" is clicked, the machining will be terminated.

4.3.3. M98, M99: call subprogram and return

M98 Pxxxx call for a subprogram, xxxx=0001~9999, is the number of subprogram.

M99   end of a subprogram, returns to process the main program.

(Nxxxx beginning of a subprogram, xxxx=0001~9999, is the number of subprogram. Beginning of the subprogram, used to define subprogram. Subprograms are defined after the M02 code.)

4.4. E code and technological data library

During machining, selecting the machining parameters (also called electric parameters) is very important for machining workpiece. This system provides not only a powerful pulse generator, but also the machining technological data library. This chapter introduces the basic rules of selecting machining parameters for Wire-cut EDM. Then tells how to use the technological data library in this system. The basic rules are provided for the unfamiliar operator, while the practician can select the machining conditions according to experiences.

4.4.1. Basic rules of machining parameter selection

The machining parameters for Wire-cut EDM include: pulse width (Pul_on), the ratio of pulse pause to pulse width (P_ratio), unloaded peak voltage (Voltage), auxiliary function (Auxiliary), peak current limit (I_max.), the system also includes feed speed