CAK50d SERIES CNC LATHES
CAK61d SERIES CNC LATHES
FTL50d SERIES CNC LATHES
FTL61d SERIES CNC LATHES

INSTRUCTION BOOK

(For Electrical Unit)

CNC System：FANUC 0i (Mate)-TD

IT IS NECESSARY FOR YOU TO READ THIS BOOK CAREFULLY AND THOROUGHLY BEFORE OPERATING THE MACHINE.

THE CHINESE VERSION OF THIS TECHNICAL DOCUMENT IN ENGLISH IS REGARDED AS FINAL.

Explanations for product models mentioned in this Instruction Book:

CAK50d series: CAK5060di, CAK 5085di, CAK 50135di, CAK 50186di, CAK 5085dj, CAK 50135dj, CAK 50186dj

CAK61d series: CAK 6160di, CAK 6185di, CAK 61135di, CAK 61186di, CAK 6185dj, CAK 61135dj, CAK 61186dj

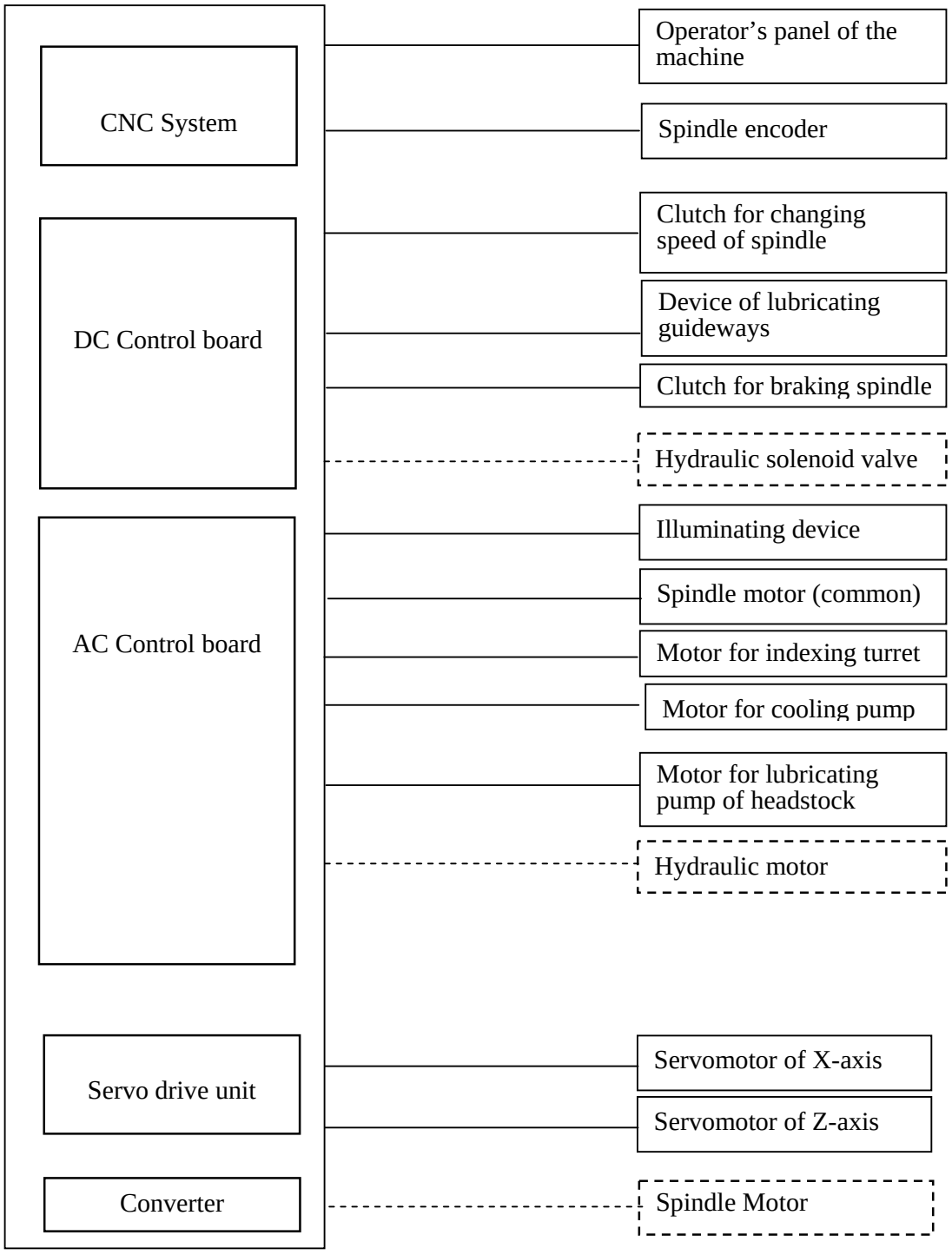
FTL50d series: FTL5060di, FTL 5085di, FTL 50135di, FTL 50186di, FTL 5085dj, FTL 50135dj, FTL 50186dj

FTL61d series: FTL 6160di, FTL 6185di, FTL 61135di, FTL 61186di, FTL 6185dj, FTL 61135dj, FTL 61186dj

CONTENTS

1 OVERVIEW OF ELECTRIC SYSTEM

1.1 Arrangement Diagram of Electrical Equipment



1.2 Table of Basic Performances of CNC System

Main Performance	Target
CNC system	FANUC 0i-Mate TD
Min. command unit	0.001mm
Max. programming size	9999.999mm
Capacity of workpiece program	512 kB
Absolute/Incremental program	X,Z / U,W
Straight/Circular interpolation	●
Metric thread	●
End of threading	●
Fix cycle	●
Combination cycle	●
Subprogram call	●
Reversion of Metric and Inch	●
Tool compensation	●
Tool tip radius compensation	●
Feed/rev., feed/min.	●
Control of constant linear speed	●
Clearance compensation	●
Compensation of pitch error	●
Parallel shift of workpiece coordinate	●
Return reference point	●
Protection of software over-travel	●
RS232C Communication	●
Display device	LCD
Feed drive device	FANUC β iS Servomotor

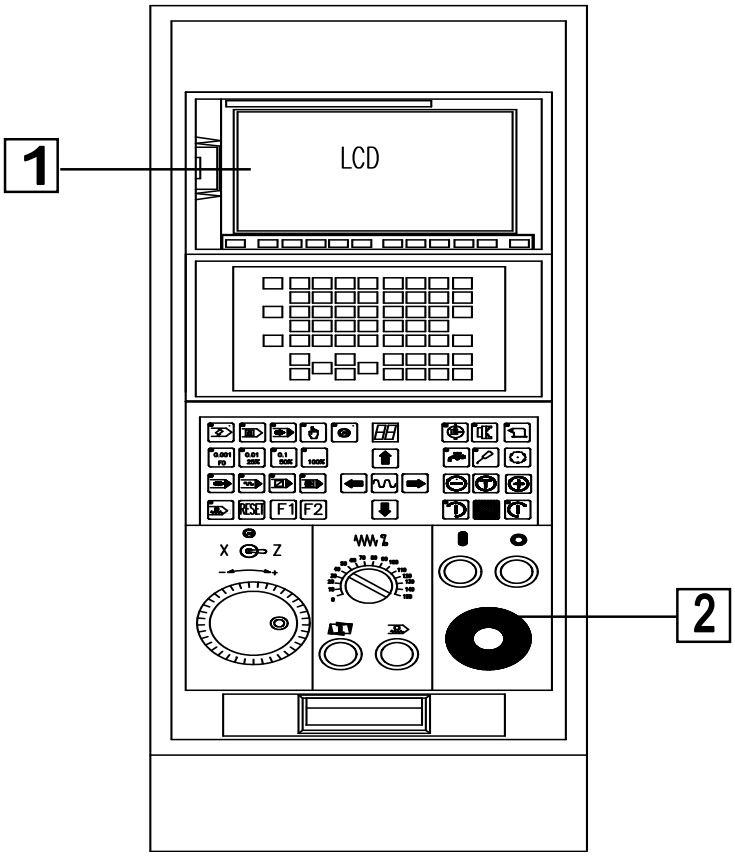
● : Basic function

1.3 List of Motors Used for the Machine

No.	Name	Specification	Remarks
1	Spindle motor	YD132M-2/4,6.5kW/8kW 1450/ 2800 r/min	For spindle with step change speed
		YL160M-4 11 kW 1450 r/min	For spindle with changing speed by converter
		YL160M-4 7.5 kW 1500 r/min	For electric-changing step
2	Motor for indexing 4-station turret	YLJ-1-4 1 Nm 1500 r/min	
	Motor for indexing 6-station turret	YLJ-3-6 3 Nm 1000 r/min	
3	Motor for lubricating pump of headstock	A0-5624 120 W 2800 r/min	
4	Motor for cooling pump	A0B-25 90 W 2800 r/min	
5	Hydraulic motor	Y90L-6 1.1k W 1000 r/min	Optional
6	Servo motor for X-axis	FANUC β iS 8 / 3000 1.2 kW	
7	Servo motor for Z-axis	FANUC β iS 8 / 3000 1.2 kW	For 600 / 850
8	Servo motor for Z-axis	FANUC β iS 12 / 2000 1.4 kW	For 1350 / 1860

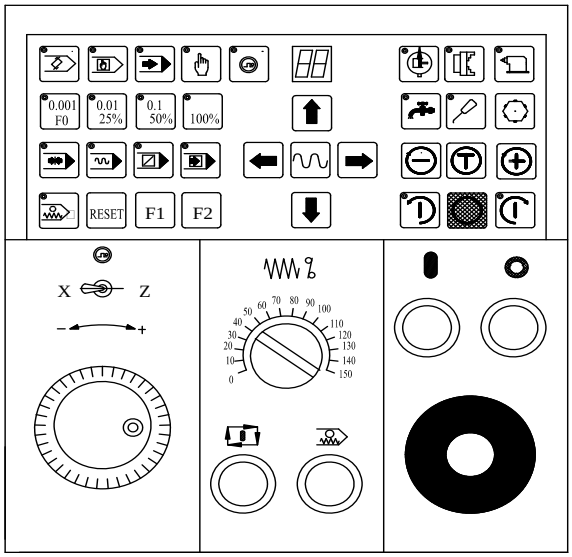
1.4 Operator′s Panel of the Machine

1.4.1 Arrangement Diagram of Operator′s Panel



1. Display and keys board of CNC device Operation keys board of the machine
2. Feed override switch (Refer to “1.4.2”and “1.4.3”)

1.4.2 Arrangement Diagram of Operation Keys Board



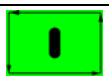
1.4.3 Table of Functions of Operation Keys Board

No.	Symbol	Name
1		Edit mode
2		MDI mode
3		Auto mode of storage program
4		Manual feed mode
5		Handwheel pulse feed mode
6		
7		Feed hold key II
8		
9	X1 F0	Min. unit of handwheel pulse G00 speed F0
10	X10 25%	Handwheel pulse unit 0.01mm G00 speed override 25%
11	X100 50%	Handwheel pulse unit 0.1mm G00 speed override 50%

No.	Symbol	Name
12	100%	G00 speed override 100%
13		
14	F2	Door switch
15		Single-program segment
16		Optional segment skip
17		Dry run
18		Locking of machine
19		Negative jogging of X-axis
20		Positive jogging of X-axis
21		Negative jogging of Z-axis
22		Positive jogging of Z-axis
23		Manual rapid speed

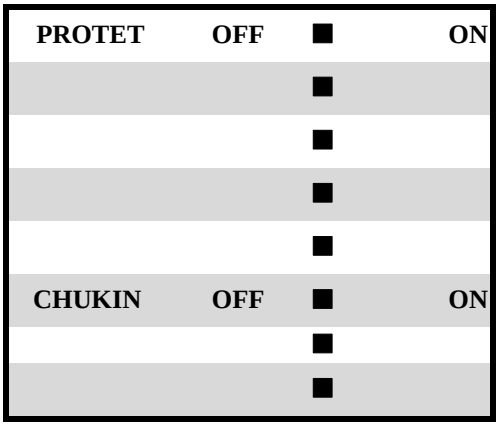
No.	Symbol	Name
24		
25		Manual tool selection
26		ON/OFF of coolant manually
27		ON/OFF of manual lubrication
28		Chucking/unchucking of chuck
29		Advancing/withdrawing of tailstock
30		Start/stop of hydraulic device
31		Manual spindle reverse
32		Manual spindle forward
33		Manual jogging spindle
34		Manually stop spindle rotating
35		Manually raising spindle speed
36		Manually reducing spindle speed
37		(Left) Display of spindle speed step (Right) Display of current tool No.
38		

1.4.4 Buttons on the Operator′s Panel

No.	Symbol	Name	No.	Symbol	Name
1		Power-on of CNC system	7		Feed override switch
2		Power-off of CNC system	8		Emergency stop button
3		Cycle start	9		 Handwheel X-axis
4		Feed hold			 Handwheel Z-axis
5			10		Handwheel pulse generator
6			11		

1.5 Soft Keys

1.5.1 Screen of Soft Keys



Screen of soft keys

Program Protection

Internal / External clamp of chuck

Name of soft keys

1.5.2 Operation of Soft Keys

① Press key 

, then  on the LCD screen to select key [OPR] , and following, press key 

, coming into soft key screen.

② Press key CURSOR ↓ , to select some switch.

③ Pressing → K I H key can right move the mark "■" to the front of ON, showing this switch has been switched on.

④ Pressing ← B C key can left move the mark "■" to the front of OFF, showing this switch has been switched off.

1.5.3 Functions of Soft Keys

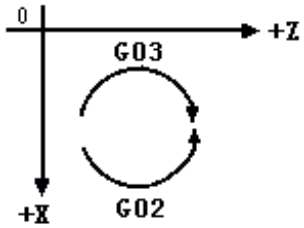
No.	Acronym	Name	Functions
1	**PROTECT**	Program protection	Program edit and parameter setting can be done on the position OFF.
2	**CHUKIN**	Selection of chuck	When on the position ON, hydraulic chuck is under status of internal clamp and on the position OFF, the hydraulic chuck under status of external clamp.

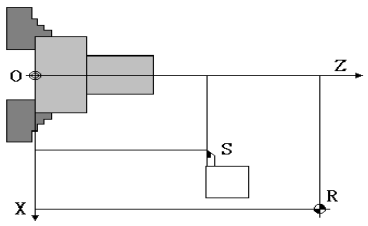
1.6 Coordinate Axes and Setting of Coordinate System

- The machine has two feed axes and adopts transmitting structure for AC servomotor to drive ball leadscrew to obtain plane continuous path movement.

Longitudinal feed axis is in parallel with spindle and it is called Z-axis (feed), and direction pointing at tailstock is positive direction.

Traverse feed axis is perpendicular with Z-axis and it is called X-axis (feed), direction of away from workpiece is positive direction.





- In order to describe moving position and moving path of tool tip, first, it is necessary to establish a coordinate system on a certain point on workpiece which is chucked on the machine, then, edit program workpiece to be turned according to certain rules and on the basis of tool movement path. The two axes of workpiece coordinate system are separately parallel with two feed axes of the machine. The axis parallel with longitudinal feed axis is called Z (coordinate) axis, and that parallel with traverse axis is called X (coordinate) axis. Direction of coordinate axis is always in keeping with that of feed axis.

Point coordinate (X、Z) is referred to as absolute coordinate. Using absolute coordinate to program is called absolute programming. To use absolute programming, first, it is necessary to set coordinate system, that is, set the origin of coordinate system to a specified position. In normal case, z-axis is set on the rotary center of spindle and X-axis is set to position on chuck endface, workpiece endface, etc..

Point S shown in the right lower figure on last page is a original position for turning. Coordinate value set by G50 is the coordinate value of the point S. Point R is reference point of the machine.

- Programming mode of not using absolute coordinate and displacement amount of using new aim position to present position is called incremental programming. In general, there are two methods to obtain incremental programming. The standard method of the machine is (U, W) address mode; U is increment of x-direction; W is increment of X-direction.
- For this machine, both the absolute programming and the incremental programming can be used, and combination programming is also used.
- Diameter programming is used in the direction of X-axis of the machine, therefore, programmed value X (U) is two times of practical coordinate value, coordinate value of X-axis displayed on screen is also two times of practical value, but movement amount of tool is only half of programmed value.
- Direction commanded by circular interpolation G02, G03 is shown by the figure in the last page.
-

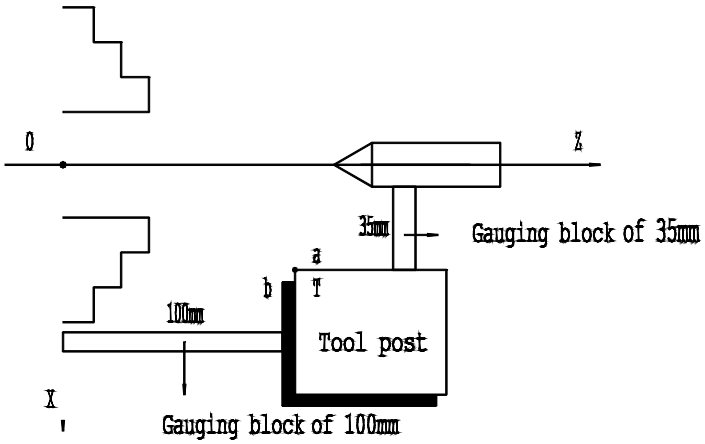
Setting method of mechanical coordinate system:

Owing to the machine′s servo systems of two axes use absolute position encoders, these encoders possesses memory function, the reference point return had been completed before delivery of the machine and the mechanical coordinate system had been also established, the coordinate system is kept by memory function of the encoder. When the machine is switched off, therefore, user needn′t to perform the job of reference point return after power-on every

time when using the machine. If the reference point is lost due to mal-voltage of the batteries or the relative position of the servo motor shaft to X/Z leadscrews is changed to result in the position of the reference point of the machine is also changed when the machine is maintained, although the system does not give alarm, the reference point should be also reset.

Practical setting method and the position of reference point are as follows:

First of all, set prm1815.4 to 0.



Setting method of Z-axis reference point:

❶ Move the carriage of the machine manually to the position shown in the figure to make the one end of gauge block of 100mm fully touch with the endface of chuck and the other end touch with the endface “b” of tool post, the filler gauge of 0.04mm is not inserted into.

❷ Take out the filler gauge, then move the carriage of the machine along +Z direction for 400mm, entry 500000 into the set position of Prm.1240 Z-axis.

❸ Set the set value of Z-axis in Prm.1815.4 to 1. thus, after power-off, then power-on of NC till this, setting of the reference point of Z-axis is completed.

Setting method of X-axis referent point:

❶ Move the carriage of the machine manually to the position shown in the figure to make the one end of gauging block of 35mm fully touch with the endface of chuck and the other end touch with the endface “a” of tool post, the filler gauge of 0.04mm is not inserted into.

❷ Take out the filler gauge, then, move the carriage of the machine along +X direction for 305mm (diameter value), entry 450000 into the set position of Prm.1240 X-axis.

❸ Set the set value of X-axis in Prm1815.4 to 1, thus, after power-off, then power-on of NC till this setting of the reference point of X-axis is completed.

- Coordinate system is not set by command G50 after help of this method establishes it.

Only without accompanying or trouble, the position anywhere the tool moves tallies with the position displayed by absolute coordinate, thereby, as long as without interference, the tool can start program to turn workpiece at any position.

- If set coordinate system by G50 after it is automatically set through returning reference point, the coordinate system set by G50 is priority.

Setting method of coordinate system of workpiece:

Edit command G50 in program.
Format: G50 X α Z γ .

❶ Select reference tool (Ex. Tool No.1)

❷ Move the tool No.1 to nearby workpiece manually to cut endface of workpiece, then move it away from the workpiece along X-axis direction. Reset (clear) the position coordinate W.

❸ Turn O.D. of workpiece manually. And then, move the tool away from the workpiece along Z-axis direction. Reset the position coordinate U and measure value D of O. D. of workpiece.

❹ Move the carriage manually until U= α -D, W= γ -l are displayed.(“L” is a distance from coordinate′s origin to trial-turning endface of workpiece.)

1.7 Code Table of G Commands

Code	Group	Function	Remarks
G00	01	Positioning of rapid travel	
G01		Linear interpolation	
G02		Circular interpolation (CW)	
G03		Circular interpolation (CCW)	
G04	00	Dwell	
G18	16	Selection of ZX plane	
G20	06	Inch input	
G21		Metric input	
G27	00	Check of reference point return	
G28		Reference point return	
G30		Second reference point return	
G32	01	Thread cutting	
G40	07	Cancel compensation of tool tip radius	
G41		Left compensation of tool tip radius	
G42		Right compensation of tool tip radius	
G50	00	Coordinate system setting/setting of spindle speed limitation	
G70	00	Finishing cycle	
G71		Roughing cycle of external cylinder surface	
G72		Cycle of roughing end face	
G73		Canned turning cycle	
G74		Cycle of drilling deep-hole on end-face	

Code	Group	Function	Remarks
G75		Grooving cycle of on OD, ID.	
G76		Combination cycle for thread cutting	
G90		Traverse canned cycle cutting	
G92	01	Canned cycle cutting of threads	
G94		End-face canned cycle cutting	
G96	02	Constant linear speed control	
◤ G97		Cancel of constant linear speed	
G98	05	Feed / min.	
◤ G99		Feed / rev.	

Notes: 1. G codes in Group 00 are non-model and they are valid only blocks in which they are Specified.

2. Alarm occurs if G codes which are not listed in the table mentioned above are specified.

3. Codes in a few different groups can be specified in the same block. The last code is valid if codes in many different groups are specified.

4. The system is under this G code state when symbol"◤" shows switch-on.

1.8 Code Table of M Commands

Code	Function	Remarks
M00	Unconditional stop of program	
M01	Conditional stop of program	
M02	End of program	
M03	Spindle forward	
M04	Spindle reverse	
M05	Spindle stop	
M08	Coolant ON	
M09	Coolant OFF	
M10	Chuck releasing	
M11	Chuck clamping	
M30	End of program and return to program beginning	
M40	Spindle neutral step	
M41	Spindle step Ⅰ	
M42	Spindle step Ⅱ	

Code	Function	Remarks
M43	Spindle step Ⅲ	
M44	Spindle step Ⅳ	
M50	Error test valid	Sharp angle transition between program segments, and hold up during cutting process.,
M51	Error test cancel	
M52	Relief at the end of cutting threads valid	It is only valid when executing G92 and G76 cycle of cutting threads.
M53	Relief at the end of cutting threads cancel	
M98	Subprogram call	
M99	Subprogram return	

1.9 Table of T Commands

T commands are comprehensive commands of tool selection and tool compensation, and they consist of T+4 digits.

Format：**T▲▲●●**

▲▲:Tool No., from 01 to max. tool number of turret, more than this max. number is not allowed.

●●:Unit No. of tool compensation, from 01 to 32. 00 can be specified, which presents cancel of tool compensation. Tool No. and tool compensation No. can be freely combined. Each tool can use tool compensation unit of many groups.

Code	Function	Remarks
T0100	Selection of tool No.1	
T0200	Selection of tool No.2	
T0300	Selection of tool No.3	
T0400	Selection of tool No.4	
T0500	Selection of tool No.5	For turret with 6 stations
T0600	Selection of tool No.6	

For example, select No.2 tool and ready for using tool compensation unit of group 3 to compensate No.2 tool, it is necessary to edit statement T0203 at proper position of program.

2 FEEDING (POWER-ON) OF THE MACHINE

2.1 Major Technical Requirements of Electrical Unit of the Machine

No.	Name of Equipment		Specification	Remarks
1	General power of the machine		15kW	
2	General current of the machine	Power supply of 380V	40A	For more than 380V the electrified wire netting is the same as that of 380V
		Power supply of 220V	60A	
3	General fuse of user′s power supply	Power supply of 380V	80A	
		Power supply of 220V	120A	
4	Wire system		3-phase 4-wire	
5	Voltage of electrified wire netting		Basic 3-phase380V	Optional voltage 220V/420V/440V/460V
6	Allowed fluctuating range of voltage of electrified wire netting		Voltage: Stable voltage value is 0.9~1.1 times of rated volt.	
			Freq.: Stable freq. value is 0.99~1.01 times of rated freq. (continuous working and for the short time working, the value is 0.98~1.02 times of rated freq (short time working).	
7	Harmonic		Sum of distorted harmonic through 2-5 times is not more than 10% of root mean square value of line volt.; for the sum of distorted harmonic through 6-30 times, not more than 2%	
8	Unbalancing voltage		Composition of negative order and that of zero order of 3-phase main voltage are all not more than 2% of that of positive order.	
9	Interruption of voltage		Continued time in interrupting of power supply or zero voltage at any time during the periodic of cycle of the power supply shall not be more than 3ms, the interval time of phase square shall be more than 1second.	
10	Voltage drop		Voltage drop shall not be more than 20% of peak voltage of one period and dropping interval time of phase distance should be more than 1second.	
11	Frequency of electrified wire netting		50Hz	Optional frequency: 60Hz
12	Allowed fluctuating range of frequency of electrified wire netting		±1%	
	Temperature of working environment		0℃～45℃	
13	Relative humidity		Less than 75%	
14	Vibration（when operating）		Less than 0.5g	

No.	Name of Equipment		Specification	Remarks
15	Control voltage	AC	110V	
		DC	24V	
16	Voltage for illuminating		AC 220V	
17	Voltage for CNC system		AC 200V	
18	Voltage of servo amplifier		AC 200V	

2.2 Basic Requirements before Power-on of the Machines

- Feeding (power-on) of electrical system of the machine must employ 3-phase, 4-wire (3 phase wires and 1 PE wire) AC power supply. Section of feeder shall not be less than recommended section, the end must be connected pressingly and firmly by cold pressing terminal whose capacity is specified.
- The special-purpose earth bolt set on the bed body of the machine must be connected firmly and reliably with ground wire.
- In order to prevent CNC system from interfering normal working high frequency equipment, some equipments, such as welding machine, etc, are not allowed to be connected nearby the machine.
-

2.3 Check before Power-on of the Machines

First switch-on：

- It is necessary to confirm if the power supply of feeding of the machine conforms with requirements of “section 2.1” and “section 2.2”.
- It is necessary to confirm if protection ground wire is connected with earth bolt, firmly and reliably, specified by the machine. And earthing resistor shall be less than 10 ohms.
- Check whether any of contactors, relays and connectors on the AC board and the DC board is loose or flick off.
- Check whether any of modules, insertion connectors of CNC system is loose or flick off.
- Check whether the breakers on the AC distribution board in the electrical cabinet are all closed.
- Check whether all electrical devices, cables and control pendant are loose, flick away or damaged.
- Check whether the belt cover door is well closed otherwise, the breaker QF0 of the general power supply of the machine cannot be well closed.

2.4 Power-on of the Machine

The machine has been provided with feeding conditions after all checking jobs mentioned above are completed and are confirmed no any trouble. Feeding procedures are as follows:

Switch-on of the breaker general power supply, after the motor for headstock lubricating pump starts, the working lamp lights on.

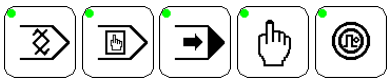
 **It is necessary to confirm the phase sequence of the power supply at the first time feeding, erroneous sequence of the power supply may result in a series of troubles which should not occur, for example, tool post does not index, cooling pump does not pump water. Headstock is not lubricated, hydraulic system has no pressure, etc. , even component(s) may be damaged.**

Simple method to judge the phase order**:**

Observe whether in the oil window on the front of headstock lubricating oil flows on. If there is oil flow, it shows the phase order is correct, if there is no oil flow and the lubricating motor runs on normally, it shows the phase order is often not correct in the first time feeding. In this case you should switch off user's switch of power supply (Attention: Not the general switch of the machine) Correcting method is to change positions of two phases at the terminal of leading wires of external power supply.

2.5 Power-on of CNC System

Press the NC start key " 1" on the Operator′s Panel of the machine, a few seconds later, the screen will be lighting and display concerned positions and command information; All of the indicating lamps on the Operator′s Panel of the machine are lighting, five seconds later, the display of tool No. and the display of step position will be alternatively displaying, the indicating lights of the other keys charge to normal displaying, the lubricating pump for lubricating guideways comes into work and starts to timing, power-on of CNC system is completed and the CNC system comes into operational able status. Wherever as long as NC switch-off button " 0 " is pressed, the system is immediately switched off.

3 OPERATION OF THE MACHINES 

3.1 Selection of Operation Mode

Five keys shown in above figure are selection keys of operation mode and they are used to select five operation modes of the machine, in any case, only one mode can be selected, the indicating lamp whose operation mode is selected lights on. Only one indicating lamp lights on in any case, others are not normal status.

3.1.1 Edit Mode 

This mode is an operation mode to input, modify, cancel, inquiry and call turning program of workpiece. Open the switch of program protection before inputting, modification, cancel of work turning program. In this mode, work program does not run.

For the operating procedures in detail of edit mode, refer to the《BEIJING FANUC 0i-MATE OPERATION MANUAL》, please. For command codes used for programming, please refer to “sections 1.6、1.7 and 1.8” of this Instruction Book.

3.1.2 Manual Data Input (MDI) Mode 

Under this mode, entry the paragraph of program from the keys board of CNC system, then, pressing the cycle start key to execute it. In general case, this mode is used to execute simple measuring operation.

Operating steps of MDI are as follows:

① Press  key, with indicating lamp lighting, and coming into MDI operation mode.

② Press《 PRGRM 》key on the CNC key board.

③ Press《 PAGE 》displaying page with MDI on the left upper on screen.

④ Press the 《 INRUT 》keys through word character keyboard of CNC, displaying input command words on the right half part of screen.

⑤ Press the Cycle Start key  after all command words are input, the indicating lamp of this key will light on. Programs come into executing status. The indicating lamp goes out after the programs are completely executed, program commands are canceled with them.

⑥ It is necessary to reentry the commands in the same if they need to be executed again. Only one program segment block can be executed one time.

⑦ During executing, it is necessary first to execute operation of turning reference point if there is displacement command.

 **It is necessary to modify parameters of the system under this operation mode!!**

3.1.3 Auto Operation Mo 

Auto. operation mode is a mode which controls the machine to perform continuous and

automatic turning according to commands of programs.

This operation mode is also referred as stored program operation mode because the programs executed by automatic operation (that is, work programs) are stored in the storage of CNC system before cycle start.

It is necessary to measure compensation value of each tool accurately by help of correct correcting tool method before starting automatic operation cycle, and then, set the measured compensation value in tool compensation unit which is specified by program.

It is necessary to move the turret exactly to the start position specified by work program before starting automatic operation cycle.

If returning reference point operation is correctly executed before starting automatic operation cycle (the lamp for reference point is not flashing), it is recommended that operator record the machine coordinate at turning start point, this is very useful to deal with possible emergency of sudden dump, tool insert damaged and rapidly back to turning.

Basic operating precedures of automatic operation are as follows:

① Press  key to select operation mode

② Select program to be executed

③ Press the Cycle Start key  ,with the indicating lamp of this key lighting on, automatic turning cycle starts.

④ After programs are executed to complete, the indicating lamp of cycle start key goes out, turning cycle ends, programs return to the beginning to prepare next execution.

If there is alarm information display of “PS000” on screen during operation, it shows there is any mistake in the programs or set data. Refer to the《BEIJING FANUC 0i-MATE OPERATION MANUAL》please.

Automatic running of programs may stop under following cases：

① Executing command M02 and M30（normal stop）;

② Reset key on the CNC keys board is pressed; Emergency stop button has been pressed.

③ Wrong alarm of program.

④ Servo alarms.

Automatic running of programs may dwell under following cases：

① Feed Dwell key  has been pressed, the indicating lamp of the feed dwell key lights on. In this time, as long as the Cycle Start key is pressed, program recovers to automatic running.

② Operation mode is out of automatic operation mode.

In this case, the machine returns to automatic operation mode as soon as press the key of automatic operation mode, then, press the Cycle Start key , program recovers to automatic running immediately.

Program executed command M00;

① Press the Cycle Start key , program recovers to automatic running immediately.

② Program executed command M01(Selection stop button of program is under ON status)

③ Press the Cycle Start key , program recovers to automatic running immediately.

Single block switch has been switched on.

Press the Cycle Start key , program runs continuously, but, it is necessary to press the Cycle Start key once after one block is executed as long as single switch is not switched off.

3.1.4 Manual Operation Mode

Press key, the indicating lamp of this key lights on, the machine and comes into manual operation mode. Under this operation mode, all the manual operation functions can all be executed.

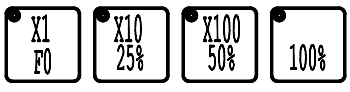
3.1.4.1 Jog and Jog Rate of X-axis and Z-axis

- Pressing key, the carriage is moving to negative direction of X-axis, releasing the key, the carriage stops moving.
- Pressing key, the carriage is moving to positive direction of X-axis, releasing the key, the carriage stops moving.
- Pressing key, the carriage is moving to negative direction of Z-axis, releasing the key, the carriage stops moving.
- Pressing key, the carriage is moving to positive direction of Z-axis, releasing the key, the carriage stops moving.

Moving rate of feed axis is determined by position of feed override switch 10％corresponds to the lowest rate of 2mm/min and 150% to the highest rate of 1260mm/min. Please refer to the 《BEIJING FANUC 0i-MATE OPERATION MANUAL》for details.

The feed override switch is not set to the zero position in normal case, otherwise, the feed axis does not move.

3.1.4.2 Rapid Jog and Jog Override



Rapid travel of the carriage can be conducted by pushing Rapid Selection key

together with the job key of certain direction. Release the Rapid Selection key, the indicating lamp of that key lights off, moving of the feed axis recovers to jog speed.

Rapid moving rate = rapid moving rate set by Prm.1422 × rapid override.

Rapid override has four selections：F0,25％,50％,100％ .

To use four 

keys to select them. Any one of them is pressed, its indicating lamp lights on, the other three key's indicating lamps light off. The percentage of the key is current rapid override.

Rapid overrides are also available for program rapid commands (G00, G27, G28 and G30, rapid travel block of canned cycle). And it is valid for rapid travel of manual return reference point, too.

3.1.4.3 Change Speed of Spindle 

Basic form : When mechanical changing steps and changing speed of double-speed motor and clutch are used, change speed by mechanical handle of 3 steps × Automatic electrical change speed of 4 classes = 12 kinds of revolutions with classes.

Unit：r/min

Class / Step	1	**2**	**3**	**4**
L. step	40	56	80	112
M. step	170	236	335	475
H. step	640	900	1320	1800

To the mechanical changing steps machine,under condition of no running of spindle,you can pressing the changing steps foot-pedal to let the spindle run in low speed, 3 steps, i .e. L. M. H. Steps can be selected by means of shifting the handle on the headstock.

Under Manual Mode, revolution speed class of spindle can be optionally selected by the spindle increasing speed key  and the spindle decreasing speed key  .

Under Auto. Mode or MDI Mode, revolution class of spindle can be directly specified by commands M41, M42 and M43.

Class number of spindle can be manually or automatically changed under spindle running.

The left digit of numeral display  displays current class of change speed.

 No changing steps while spindle running (without in the situation that pressing the changing steps foot-pedal to let the spindle run in low speed)(Shift the mechanical change speed handle is shifted)!

This form employs way by which universal 3–phase AC non-synchronous motor directly starts spindle. Because the reason of this kind of motor's intrinsic characteristic, frequent starting and stop spindle may result in overheat of spindle motor to form overheat protection, therefore, minimum time interval of start and stop should not be less than 30 seconds when operating this machine.

Selection forms:

When mechanical changing step and regulating speed by changing frequency change speed by mechanical handle of 3 steps × common converter to drive universal AC non-synchronous motor in each step to realize stepless regulating speed.

Change speed range of 3 steps is shown by following table:

Step of Handle	Numeral Display	Range of Speed (r/min)
L (Low step)	1	72 ～ 370
M (Middle step)	2	175 ～ 905
H (High step)	3	425 ～ 2200

- Under condition of no running of spindle, 3 steps, i. e. L. M. H. steps can be selected by means of shift the handle on the headstock.
- The left digit of numeral  display displays current class of change speed.
- Speed in every step is commanded by command Snnnn, and "nnnn" represents speed value of spindle. The system will automatically limit it to max. speed specified by this step when speed given exceeds the max. speed of spindle of this step.
- Under Manual Mode to use the increasing key  or the decreasing key  of spindle can increase or decrease speed of spindle in range of the step, jogging once changes one ΔS(set by K3+K4),if the pressing time exceeds five seconds, the changing is continuous. The upper limitation and the lower limitation of speed can be set by K5+K6, K7+K8.The speed is still kept no change until new s command comes out after the speed under manual made changes into the speed under Auto. mode. And for the spindle speed commanded under Auto. mode to be switched into the speed under manual mode, the result is as the same, too. Too large load inertia on spindle (with larger fixture) may make the converter give alarm when retarding or stop of the machine, in this case, it is necessary properly to prolong retarding time of the converter, refer to the Table of Parameters for Converter, please.

No changing steps while spindle running (the mechanical change speed handle is shifted)!

Form of power-driven changing step:
Stepless regulating speed of converter motor driven by converter with in every step

Unit：r/min

Step of spindle / Speed Range	M40	M41	M42	M43
Speed Range （r / min）	Neutral step	22-246	66-725	203-2200

- Under Manual Mode or Auto. Mode to use the increasing key ⊕ or the decreasing key ⊖ of spindle can increase or decrease speed of spindle in range of this step.
- Under Auto. Mode or MDI Mode, the steps1, 2, 3 and 4 as well as neutral step of spindle can be selected by commands M41, M42, M43, M44 and M40.
- To change steps of spindle are not allowed manually or automatically under running status of spindle.
- The left digit of numeral 88 display displays current step position.

3.1.4.4 FWD, REV and Stop of Spindle

- Select changing step and required speed of spindle according to the Changing Step Chart for spindle and by M commands.
- Press the Spindle FWD key ,its indicating lamp lights on, having spindle forward.
- Press the Spindle REV key , its indicating lamp lights on, having spindle reverse.
- Press the Spindle Stop key, indicating lamps which indicate spindle forward and reverse light off, spindle stops rotating.
- In the same time when spindle executes to stop rotating, spindle has a braking, the spindle is changing speed by double-speed motor and clutch is forced to make its own rapidly be stopped running through electromagnet breaker attracting on. Braking time is set by parameter T16. User may properly adjust it if necessary. The spindle of changing frequency-regulating speed realizes braking by means of energy consumption of resistor connected outside of converter. The converter determines the braking time. Refer to the chapter 3.1.4.4 change speed of spindle.

Under Auto mode or MDI mode:

- After command（M03）of spindle forward is executed, the indicating lamp of spindle forward lights on just as the same of manual operation.
- After command（M04）of spindle reverse is executed, the indicating lamp of spindle reverse lights on just as the same of manual operation.
- If spindle stop command（M05）is executed, the indicating lamps for forward and reverse are all off.

No starting Spindle under neutral position!

It is necessary to close the protection door well before start of spindle!

It is strictly forbidden to open the protection door while the spindle is running.

Do not start spindle at high speed in case of workpiece is not clamped for the machine equipped with non-hydraulic chuck!

3.1.4.5 Jog of Spindle

Pressing the Spindle Jog key, spindle is jogging, releasing it, spindle stops rotating.

3.1.4.6 ON/OFF of Coolant

- Press the "ON/OFF of coolant" key , its indicating lamp lights on, the cooling pump is working. Open the cock of coolant, coolant is gushing. If press it again, the lamp is off, the cooling pump is off and coolant stops gushing.
- Under Auto. Mode or MDI mode, if command (M08) of coolant ON is executed, its indicating lamp also lights on. Command（M09）of coolant OFF is executed, or press this key again, its indicating lamp is off， coolant is off.

3.1.4.7 Manual Tool Selection

- Pressing the Manual Tool Section key , turret is automatically releasing, then, it is counterclockwise indexing and searching required tool station. After the Tool Selection key is released, the turret counter leans against the seat automatically, then, it is locked on the target position. The right number of the numeral display displays current tool station number.

- Lightly to press the Tool Selection key can realize pressing one time to select one tool station.

Non-enough of delay of tool post locking can affect locking rigidity of tool post, but ov but too long delay of tool post locking can make the tool post motor overheat and resulting in damage. Delay time of tool post locking is set by parameter T7. And the parameter had been well set before delivery, it should not be changed as own will. If you find that locking of tool post is not enough and affecting machining accuracy, properly increasing time setting value is allowed. Pay attention to temperature of tool post motor when increasing the Value. Record the time data in the Parameter Table after adjustment.

■ Manual correcting tool method

① Select a reference tool (for example, Tool 1#)

② Move the tool 1# to near workpiece to turn the endface of workpiece, then, move the tool away from the workpiece along x-axis direction. Reset the position coordinate W.

③ Turn external cylinder of workpiece manually, then, move the tool away from the workpiece along Z-axis direction. Reset the position coordinate U.

④ Move the turret to safe area to change the second tool.

⑤ Make the second tool tip touch lightly with endface of workpiece, press the Z key, then, press the (WR) key.

⑥ Make the second tool tip touch lightly with O.D of workpiece, press the X key, then, press the INPUT key. Up till now, tool compensation of the second tool is automatically input into 2# unit.

⑦ Repeat ④-⑥ steps to complete tool compensation input of all tools.

3.1.5 Feed Mode of Manual Pulse Generator 

Press  key, its indicating lamp lights on, the machine is under feed mode of handwheel pulse generator. After select x-axis or z-axis by means of directions of  , operator can turn the handwheel of handwheel pulse generator to make turret moving of all sides (forwards, backwards, left and right). Its speed can be adjusted as will. It is very available for correcting tool operation in short distance, etc.

Operating procedures are as follows:

① Select the handwheel Pulse Override  keys .

② Handwheel pulse overrides have three: 0.001/0.01/0.1mm. You can select any kind of them according to rapid, slow, finishing and rough. The indicating lamp of selected override lights on, thus, equivalent value per scale on the handwheel can be determined.

③ Selection of manual feed axis:
Turn the switch  to the position X , with x-axis selected; Turn the switch to the position Z, with z-axis selected.

④ Turn the hand wheel clockwise or counter-clockwise.

■ Manual feed mode can execute manual operations, such as spindle changing speed, manual spindle start and stop, manual coolant ON and OFF, manual tool selection, etc.

3.2 Cycle Start and Feed Hold  

- Cycle start key 
  Starting program operation can be performed under either Auto. mode or MDI mode. During program being performed, the indicating lamp on the left-upper corner of the key lights on.

- Feed hold key 
  During program being performed under Auto. mode or MDI mode, press the key,

having the indicating lamp on the left-upper corner of the key lights on. Program being performed is held. Repress the feed hold key; its indicating lamp is off. The program is continuously performed.

3.3 Feed Override Switch 

During program being performed, you can use this switch to adjust feed speed that is specified in the program at any time to obtain optimum turning result. Practical feed speed after adjustment can be observed through display screen. Adjusting range：10％～150％.

3.4 Trial-running (Dry Running 

Trial- running is also referred as dry run, it is a operation which is to test, check the newly input programs of turning workpiece under non-turning condition. In order to short debugging time, feed rate is forced to max. value by the system during trial - running.

Operating procedures are as follows:

① Select Auto. mode to call out program to be tested.

② Press the Trial-running key ，in this time, its indicating lamp lights on, showing that trial-running status is valid.

③ Press the Cycle Start key, its indicating lamp lights on, trial - running operation begins.

3.5 Lock Operation of the Machine 

- Press  key, its indicating lamp lights on, showing lock of the machine is valid. Press it again, having the indicating light off, showing lock status of the machine is released.
- Under locking status of the machine, moving operation (jog, handwheel feed) of each axis under manual mode can only change displayed values of position and not move every axis of the machine, but, spindle, coolant and turret work normally.

Under locking status of the machine, the programs under Auto. and MDI are normally running, position display values are changing, but every axis of the machine doesn't move, and spindle, coolant and turret work normally.

3.6 Operation of Optional Skip of Block 

- Press  key, its indicating lamp lights on, block skip function is valid. Press it again, its indicating lamp goes out, block skip function invalid.
- Under Auto. mode, during valid period of block skip function, all of blocks which are preceded by symbol "/" （erase character）in front of block number all skip

over not to be executed. During invalid period of block function, all of blocks are all executed normally.

- Functional application： Edit some special blocks, such as trial-turning, measurement and correcting tool in program. Precede block numbers with symbol "/" .To use this kind of block skip function can control the machine to execute these blocks by selection.

3.7 Operation of Single Block 

- Press  key in Auto. mode, its indicating lamp lights on, single block function is valid. Press it again, the indicating lamp goes out, single block function is canceled. The Single Block Function key is allowed to be switched over while program is continuously running.
- During valid period of single block in Auto. mode, the Cycle Start key is pressed every time, executing one block, it is pressed again, executing next block.
- Functional application：It is mainly used to measure program, and it also combined with trial-run, lock of the machine and block skip functions can be used with together according to practical conditions.

3.8 Limitation and Release of Over travel of the Machine

Limitation of Store Travel (Soft limitation)：

During operation, the carriage of the machine may move out of safe area set by Prm.1320, 1321 in some direction due to some reasons (operation fault, programming data error, etc.), CNC system alarms and stops moving of carriage. Move the carriage out of forbidden area in opposite direction, then, you can do normal operation.

After the coordinate system of the machine is established when manual reference point is executed after power-on of the machine, only can soft limitation function be valid.

3.9 E-stop Operation 

- A red mushroom emergency stop button is on the right- upper corner on the Operator's panel. If emergency case occurs, all actions of the machine stop immediately as soon as the emergency stop button is pressed and the button is self-locked automatically. To turn the button to a certain angle clockwise can make it be recovered after emergency-shooting or trouble-shooting.
- After the emergency stop button is pressed, spindle may run for 3-5 seconds due to inertia. The turret can also rotate a little.

3.10 Operation of Lubricating Guideways

The machine has automatic intermittent lubricating function of guideways.

The machine comes automatically into travel lubricating state of guideways after power-on. The travel lubricating controls “stop” and “start” (ON and OFF) of the lubricating pump of guideways through calculating the moved distance of the servo axis. Start time (Filling oil time) is set by T21 (ms), and upper limit of travel is set by D152 (cm). Operator may adjust them by himself according to practical requirement.

If you keep pressing the button for lubricating the guideways, the lubricating pump will be continuously working.

4 OPERATION OF HYDRAULIC SYSTEM , PNEUMATIC AND HYDRAULIC CHUCK AS WELL AS HYDRAULIC TAILSTOCK

4.1 Start of Hydraulic System

① Press the auto. air-switch "QF5" on in the electrical cabinet.

② Press the button of Hydraulic Start/Stop with its indicating lamp lighting, the hydraulic pump will be starting. Press it again; the indicating lamp goes out, hydraulic pump will stop. After there is emergency stop operation, the hydraulic pump stops. Hydraulic pump should be restarted.

③ Adjust the pressure of hydraulic system and the pressure relay to the pressure required by chucking workpiece.

4.2 Operation of Hydraulic Chuck

- Using can all complete chucking and unclamping of chuck. Also using the foot-pedal switch of hydraulic chuck complete chucking and unclamping of chuck. Press the key or step the foot-pedal switch one time, chuck is chucking, the key's indicating lamp lights on. Repress the key or step the pedal switch again, chuck is unclamping, the key's indicating lamp goes out.
- Chucking of hydraulic chuck is indicated by pressure relay, user according practical condition, also to select travel switch to indicate it, may adjust chucking force.

In order to avoid accidents, to operate the chuck is not allowed when the spindle is under running status. It is necessary to make spindle′s speed retarding to zero, and you can be allowed to operate the hydraulic chuck.

4.3 Operation of Hydraulic Tailstock

Using the key can complete advancing and withdrawing of the hydraulic tailstock's quill, also using the foot-pedal switch SQ50 to complete action of the tailstock quill. Press the key (or step the foot-pedal switch) once, the quill of tailstock is advancing with the indicating lamp lighting; and press it (or step it) again, the quill is withdrawing, the indicating lamp goes out. The lowest tightening up pressure of advancing of the hydraulic tailstock quill is indicated by the pressure relay. The tightening up pressure may be adjusted by user according to the practical situation, also to select travel switch to indicate it.

In order to avoid accidents, to operate the tailstock is not allowed when the spindle is under running status. It is necessary to make spindle's speed retarding to zero, you can be allowed to operate the hydraulic tailstock.

5 OPERATING FLOW DIAGRAM AND USER'S PARAMETERS

5.1 Operating Flow Diagram

Close the general power supply breaker, and then the machine gets normal power-on

Switch on the push-button, CNC system gets power-on

Input turning programs, check and confirm their rightness.

Lock the machine and dry-run programs to verify rightness of the programs, especially, carefully observe whether he coordinate sizes of every segment are correct. After complete those, it is necessary to cancel dry-run operation.

Release the machine and load the workpiece for trial-running, and select every tool manually, an then measure the tool compensation of every tool in method of trial-running and input the tool compensation unit specified by the program. At last, also pay attention to the positive and negative sign and decimal point.

Taking the trial-turning workpiece as reference, move the carriage manually to coordinate position set by command G50. Unload the trial-turning workpiece.

Call out program of current turning workpiece, and select Auto. operation mode and proper feed override as well as rapid override. Press the Cycle Start key to start the automatic cycle turning. For turning first workpiece, it is better to select lower rapid override and using single block function can reduce troubles resulting from program and error from correction tool.

After first workpiece is completed, measure sizes of each turning position and modify tool compensation value of each tool. Then, turn the second workpiece. Recover rapid override of 100% to turn all of workpieces after confirming all sizes are correct.

5.2 Table of User′s Parameters

Prm.No.	Meaning of Prm.	Prm. Value	Remarks
1420	x/z-axis rapid speed		
1851	Clearance compensation of x/z-axis	-9999—+9999	This Prm. should be adjusted after the machine is used for a certain period.
1240	Coordinate of x/z axis reference point		
5132	G71/G72 Deep turning	1-9999999M	
5133	G71/G72 Relief	1-9999999M	
5135	G73 X-axis relief	1-9999999M	
5136	G73 Z-axis relief	1-9999999M	
5137	G73 Fragmenting digit		
5139	G74/75 Return amount	1-9999999M	
5140	G76 Minimun turinting depth		
5141	G76 Permissible error for finishing		
5142	G76 Repeat digit		
5143	G76 Tool tip angle	0/29/30/55/60/90	
5013	Allowing worn limit of tool		
5014	Max. wearing value of tool		

6 ELECTRICAL MAINTENANCE AND ADJUSTMENT OF THE MACHINE

When alarm information appears on the display screen, some PLC alarms can be canceled by outer reset key [F1] and some must be deleted by the reset key of the system, please do trouble-shooting according to remedy of relative alarm given by《BEIJING FANUC 0i-MATE OPERATION MANUAL》.

Batteries under dump status protect the parameters and turning programs of CNC system. The screen displays alarm warn when energy of the batteries is not enough, in this case, it is necessary for user to change them immediately under status of power-on of CNC system, otherwise, the parameters and the programs may be lost.

It is necessary for you to read this INSTRUCTION BOOK and《BEIJING FANUC 0i-MATE OPERATION MANUAL》, carefully and thoroughly before operating the machine to avoid operation failure to result in accidents.

Appendix: Alarm Information and Remedy Method

Alarm Code	**Displayed Information**	**Meaning of Alarm Information**	**Alarm Condition or Cause**	**Consequence Resulted from Alarm or Triggered Action**	**Remedy Method Alarm**
#2004	EMG OFF	Emergency stop; switch switched off	Emergency stop button is pressed or circuit is broken.	The machine is under emergency stop status.	Shoot the trouble and press the outer reset key F1.
#2005	MAIN MOTOR OVERHEAT	Heat-protection of main motor is off	The built-in overheat protection switch of the main motor is off.	Intermittent stop, spindle has stopped.	
#2006	BED-HEAD LUBRICATE OFF	Switch for lubricating headstock is off,	The air-switch for lubricating headstock is off.	Restarting the turning programs is forbidden.	
#2010	HYDRAULIC NOT RUN	Hydraulic device is not started.	Hydraulic device is not started after the machine is satrted.	Restarting the turning programs is forbidden.	
#2012	HYDRAULIC CHUCK PRESS LOW	Pressure of the hydraulic chuck is tool low.	Pressure of the hydraulic chuck is too low	Intermittent stop, the spindle has stopped.	
#2013	HYDRAULIC TAIL PRESS LOW	Pressure of the hydraulic tailstock is too low.	Pressure of the hydraulic tailstock is too low.		
#2020	TURRET CODE ERROR	Turret code is error	More than practical tool No. or tool No. 0 has been instructed.	The programs have stopped and the system does not response.	

Alarm Code	Displayed Information	Meaning of Alarm Information	Alarm Condition or Cause	Consequence Resulted from Alarm or Triggered Action	Remedy Method Alarm
#2021	TURRET RUN OVERTIME	Rotating o the turret is overtime.	Time of arriving at the instructed tool station is overtime specified.	. The programs have stopped and the system does not response	Shoot the trouble and press the outer reset key F1.
#2022	TURRET LOCK OVERTIME	Locking of the turret is overtime.	Back-leaning and locking time of the turret is overtime specified.		Start the hydraulic device and press the outer reset key F1.
#2023	TURRET NOT LOCK	The turret has not been locked.	Signal is not sent when back-leaning and locking the turret.		Shoot the trouble and press the outer reset key F1.
#2040	TRANSDUCER ALARM	The transducer fault	The transducer fault or the outer fault.	Starting the spindle is forbidden	Shoot the trouble and press the outer reset key F1.
#2041	TRY TO RUN SPINDLE WHILE CHUCK NOT LOCK	The spindle is tried to be started while the chuck is not firmly clamped.	The spindle is tried to be started while the chuck is not firmly clamped.	Starting the spindle is forbidden	Tighten up the tailstock and press the outer reset key F1.
#2042	TRY TO RUN SPINDLE WHILE TAIL NOT LOCK	The spindle is tried to be started while the chuck is not tightened up.	The spindle is tried to be started while the chuck is not tightened up.		Shoot the trouble and pres the reset key f1.
#2043	SPINDLE GEAR ABNORMAL	The spindle step is abnormal.	The switch of spindle step has mistaking action.	Restarting the turning program is forbidden	Shoot the trouble and press the outer reset key F1
#2045	SPINDLE CHANGE GEAR OVERTIME	The spindle change gear abnormal.	The spindle change gear action abnormal.	Restarting the turning program is forbidden	Shoot the trouble and press the outer reset key F1
#2053	SAFETY DOOR BE OPENED	The safety door has been opened.	The safety door has been opened during normal turning.	Intermittent stop; the spindle stops.	Close the safety door and pres the outer reset

Alarm Code	**Displayed Information**	**Meaning of Alarm Information**	**Alarm Condition or Cause**	**Consequence Resulted from Alarm or Triggered Action**	**Remedy Method Alarm**
#2054	SAFETY DOOR NOT CLOSE	The safety door is not closed.	The spindle or the turning programs are attempted to be started when the safety door is closed.	Starting the spindle or the turning programs is forbidden.	key F1.

Other matters needing attention:
If actions of the machine can not be realized due to voltage fluctuation or overload making the air breaker OFF, please check if overload occurs to circuits and electric elements or short circuit exists, after trouble shooting, reset the corresponding air breakers, and for correspondence of the motors and each air breaker is shown below:

Name	Air Breaker	Portion Cause or Phenomenon
General switch	QF0	Door of electric cabinet is not closed, short circuit occurs to electric elements.
Lubricating motor	QF2	Overload or short circuit of motor, or short circuit of cables occurs.
Cooling motor	QF3	
Motor for turret	QF4	
Hydraulic motor	QF5	

Reliable work of travel switch is essential for machine, and therefore, periodically check if there is foreign matters blocked on its contact, and if there is, clear it in time. If alarm occurs when the machine has not been over travel, and it is unable to carry out the operation of return zero point, check if there are foreign matters blocked on the contactor.

If over-travel switch of the machine is pressed or it is needed to change cables of motor encoder, etc., it is necessary for you to reset the reference point of the machine (refer to the method described above).

Modification of parameters of the machine.

FANUC control system: press key “OFS/SET”, and then soft key “SETTING”, selecting MDI mode. Set “Parameter Read-in” to 1, No. 100 alarm will occur to the system, press key “SYSTEM” and input the parameter No. to be modified. After pressing the horizontal soft key “SEARCH”, the system will skip automatically to the parameter to be modified to carry out the modification. Set “Parameter Read-in” to 0, the alarm of the system will disappear. If No. 000 alarm occurs after modification of the parameter, it is necessary to re-power on the system.

Parameters of the machine can influence the accuracy and actions of the machine greatly, so

the modification of parameters of the machine shall be carried out by professional. Erroneous input of parameters will bring great damage to the machine and injury to operation personnel.

Adjustment of electro brake: braking and stopping speed of spindle differs because the loading of spindle is different, and the braking force and braking time (left of the brake shows the braking force and right shows the braking time) can be regulated according to actual loading status. In addition, it is available to observe the status of brake through the four indicating lights “TROUBLE” “POWER SUPPLY” “BRAKING” AND “ALLOWANCE” on the brake.

While short circuit occurs to the machine, it is necessary to check if the cable is broken by protection or other components, and normally, there is standby cable to take the place of the broken cable, and if there is not, you should change the cable.

Matters needing attention to use of air conditioner:

Filter net of air conditioner of electric cabinet shall be cleaned periodically, and otherwise, the normal work of the system will be influenced because the temperature in the electric cabinet is too high. The cleaning of filter net shall be carried out according to working environment, and normally, twice for every month. For light dirt, clear it by dust collector or flapping, if the dirt is too much, clear it out by neutral detergent or wash it by clean water. After complete cleaning, dry it and then re-install it.

The temperature inside the cabinet has been well set, and un-profession is forbidden to do any regulation.

For maintenance, when it is necessary to open air conditioner, it is necessary to cut off the power.

The temperature shall be properly regulated higher in baiu seasons and wet seasons.

Every day, you should check if the drainpipes are blocked or the water in water receiver overflows. Every day, at least, the water receiver shall be poured once, and once for every 4 hours during baiu seasons and wet seasons.