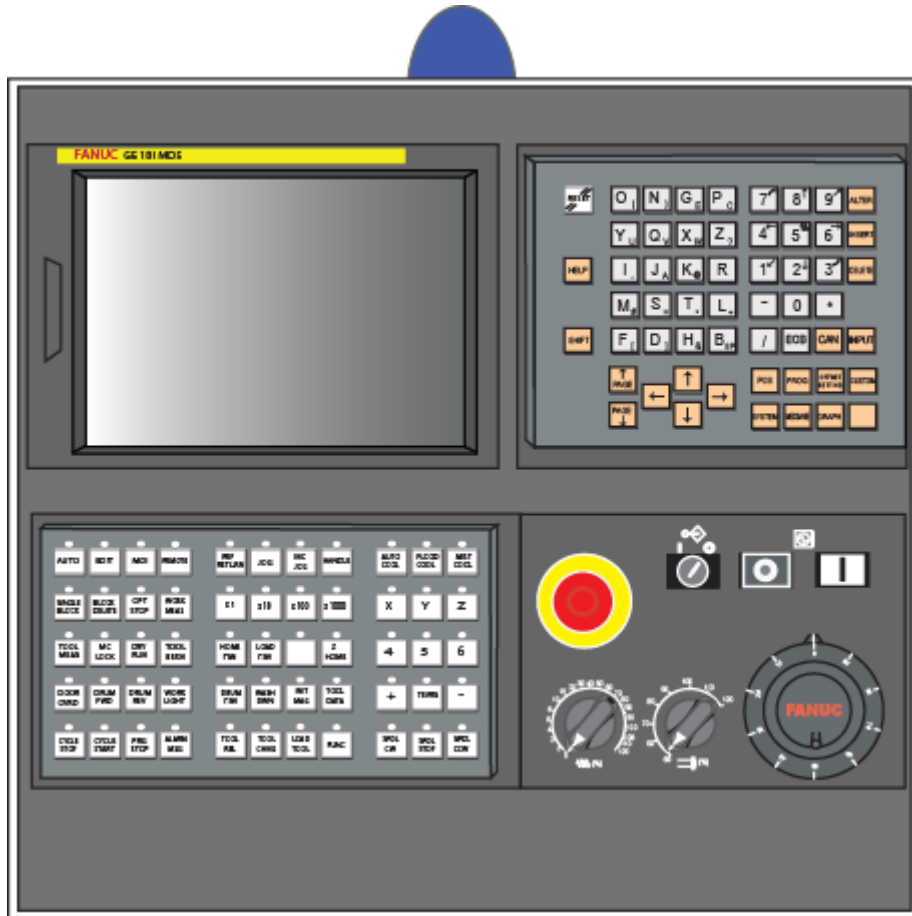


# GE FANUC 18i CONTROL OPERATOR MANUAL



FADAL MACHINING CENTERS, LLC

Corporate Office .....phone (818) 407-1400..... fax (818) 407-0020  
 Service / Parts .....phone (818) 727-2100..... fax (818) 407-1004  
 Programming Support.....phone (818) 727-2100..... fax (818) 407-0061

support@fadal.com  
 20701 Plummer Street, Chatsworth, California 91311 USA



---



---

## TABLE OF CONTENTS

---



---

<b>1.0 POWER ON/OFF .....</b>	<b>1</b>
1.1 PRE-START CHECKING STEPS .....	2
1.1.1 OIL RESERVOIR .....	2
1.1.2 AIR PRESSURE .....	2
1.1.3 WATER RESERVOIR .....	3
1.1.4 FLOOD COOLANT .....	3
1.1.5 COOL POWER RESERVOIR .....	4
1.2 POWER ON/OFF .....	4
1.2.1 POWER ON .....	4
1.2.2 POWER OFF .....	5
<b>2.0 OPERATOR PANEL LAYOUT .....</b>	<b>7</b>
2.1 MAIN OPERATOR PANEL .....	8
2.2 SAFETY FUNCTIONS .....	8
2.2.1 EMERGENCY STOP SWITCH .....	8
2.2.2 DOOR OVRD .....	8
2.2.3 WORK LIGHT .....	9
2.2.4 ALARM MSG .....	9
2.3 OPERATION MODE SELECTION .....	9
2.3.1 AUTO .....	9
2.3.2 EDIT .....	9
2.3.3 MDI .....	9
2.3.4 REMOTE .....	9
2.3.5 REF RETURN .....	9
2.3.6 JOG .....	10
2.3.7 INC JOG .....	10
2.3.8 HANDLE .....	10
<b>3.0 MANUAL OPERATION .....</b>	<b>11</b>
3.1 MANUAL OPERATION MODES .....	12
3.1.1 MANUAL REFERENCE POSITION RETURN .....	12
3.1.2 JOG OPERATION .....	13
3.1.3 INC JOG OPERATION .....	14
3.1.4 HANDLE OPERATION .....	15
3.1.5 JOG FEED RATE .....	16
<b>4.0 AUTOMATIC OPERATION .....</b>	<b>17</b>
4.1 AUTOMATIC OPERATION MODES .....	18
4.1.1 MDI OPERATION .....	18
4.1.2 MEMORY OPERATION (AUTO MODE) .....	19
4.1.3 DNC OPERATION .....	20

4.1.4 MEMORY PROTECTION KEY .....	21
<b>5.0 TEST FUNCTION .....</b>	<b>23</b>
5.1 TEST FUNCTION MODES .....	24
5.1.1 SINGLE BLOCK .....	24
5.1.2 BLOCK DELETE .....	24
5.1.3 OPTION STOP .....	24
5.1.4 PROGRAM STOP .....	24
5.1.5 MC LOCK .....	24
5.1.6 DRY RUN .....	24
<b>6.0 MACHINE SETUP FUNCTION .....</b>	<b>27</b>
6.1 SETUP MODES .....	28
6.1.1 WORK MEAS .....	28
6.1.2 TOOL MEAS .....	28
6.1.3 FUNCTION .....	28
6.1.4 HOME PSN .....	28
6.1.5 LOAD PSN .....	28
6.1.6 Z HOME .....	29
<b>7.0 TOOL OPERATION .....</b>	<b>31</b>
7.1 MANUAL OPERATION .....	32
7.1.1 DRUM FWD .....	32
7.1.2 DRUM REV .....	32
7.1.3 DRUM PSN .....	32
7.1.4 TOOL REL .....	32
7.1.5 TOOL CHNG (DATC ONLY) .....	34
7.1.6 TOOL LOAD .....	34
7.1.7 REFERENCE THE DRUM (ONLY FOR DATC) .....	36
7.2 AUTO OPERATION .....	36
7.2.1 T-COMMAND (TOOL NUMBER) .....	36
7.2.2 TOOL CHANGER COMMAND .....	36
7.3 TOOL MANAGEMENT .....	38
7.3.1 CLR MAG .....	38
7.3.2 TOOL DATA .....	38
7.3.3 TOOL BRKN .....	39
7.3.4 EDIT THE TOOL/MAGAZINE LIST .....	39
7.3.5 TOOL ORDER PROCEDURE (ONLY FOR ATC) (M11) .....	39
7.3.6 INITIALIZING ATC TOOL MAGAZINE .....	41
7.3.7 TOOL LOADING (ATC) .....	41
7.3.8 TOOL CHANGER (RECOVERY ATC) .....	42
7.3.9 DATC 32-TOOL WITH TOOL CAGE DOOR (6535 40T/50t) .....	42
<b>8.0 COOLANT OPERATION .....</b>	<b>45</b>
8.1 COOLANT CONTROL PUSH BUTTONS .....	46
8.1.1 AUTO COOL .....	46
8.1.2 FLOOD COOL .....	46
8.1.3 MIST COOL .....	46

<b>9.0 SPINDLE OPERATION .....</b>	<b>47</b>
9.1 SPINDLE OPERATION PUSH BUTTONS .....	48
9.1.1 SPDL STOP .....	48
9.1.2 SPDL CW .....	48
9.1.3 SPDL CCW .....	48
9.1.4 MANUAL SPINDLE SPEED .....	49
9.1.5 M CODE / S CODE .....	49
9.1.6 SPINDLE SPEED RANGE .....	49
9.1.7 SPINDLE OVERRIDE .....	50
9.1.8 SPINDLE ORIENTATION .....	50
<b>10.0 HMOP OPERATION GUIDE .....</b>	<b>51</b>
10.1 HMOP OPERATION ON/OFF SWITCH .....	52
10.2 OVERRIDE SWITCH .....	52
10.3 MEMORY OPERATION PROCEDURE .....	53
10.4 MANUAL HANDLE FEED PROCEDURE .....	53
10.5 MANUAL JOG FEED PROCEDURE .....	54
10.6 MANUAL REFERENCE RETURN .....	54
10.7 MANUAL SPINDLE CONTROL .....	55
10.8 KEY SHEET LAYOUT .....	55
10.9 MESSAGE KEY .....	56
<b>11.0 ROTARY TABLE .....</b>	<b>57</b>
11.1 TR 165 .....	58
11.1.1 MANUAL REFERENCE RETURN .....	58
11.1.2 B AXIS TILT TABLE .....	58
11.1.3 G90 ABSOLUTE MODE .....	58
11.1.4 G91 INCREMENTAL MODE .....	59
11.1.5 FEED RATE .....	59
11.2 V300 .....	59
11.2.1 REFERENCE RETURN .....	59
11.2.2 A AXIS DIRECTION .....	60
11.2.3 G90 ABSOLUTE MODE .....	60
11.2.4 G91 INCREMENTAL MODE .....	60
<b>12.0 APPENDIX .....</b>	<b>61</b>
12.1 M CODE LIST .....	62
12.2 ALARM LIST .....	64
12.3 MACHINE COLD START PROCEDURE .....	70
12.4 FANUC FILES BACKUP PROCEDURE .....	72
<b>13.0 INDEX .....</b>	<b>77</b>



---

---

## 1.0 POWER ON/OFF

---

---

## 1.1 PRE-START CHECKING STEPS

### 1.1.1 OIL RESERVOIR

Examine the oil levels. Both levels should be filled up to one inch from the top of the reservoir. The spindle oil reservoir may have oil in it for up to six months. The way lube oil reservoir may run out of oil in one week.



Figure 1-1: Spindle Oil Reservoir (opt)



Figure 1-2: Way Lube Oil Reservoir

### 1.1.2 AIR PRESSURE

The inlet air must not exceed 120 PSI. This supplies air to tool IN-OUT cylinder and it is used for air blast during tool change. Visually inspect the air pressure gauge to verify that it is set to at least 80-100 PSI. Air is used to operate:

- belt change



- spindle orient
- way lube pump
- spindle air/oil pump
- spindle air seal
- tool changer



Figure 1-3: Air Pressure

### 1.1.3 WATER RESERVOIR

Most new VMC models release water collected in the water reservoir automatically. It is advisable to place an additional water trap in the air line going to the machine.

### 1.1.4 FLOOD COOLANT

Replenish the flood coolant level to avoid running out of coolant during execution of the program.



Figure 1-4: Flood Coolant

## 1.1.5 COOL POWER RESERVOIR

Examine the cool power reservoir once a month.



Figure 1-5: Cool Power Reservoir

## 1.2 POWER ON/OFF

### 1.2.1 POWER ON

1. Turn On the main breaker.

To power on the machine, press the safety lock and turn the power switch in the clockwise direction.

2. Press the CNC ON push button located on the operator panel. CNC will boot up and enter operating status.



Figure 1-6: Power ON

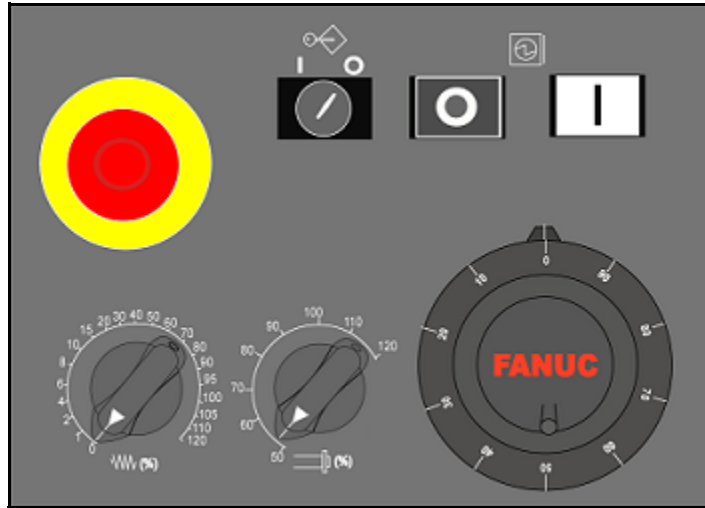


Figure 1-7: Operator Panel A layout

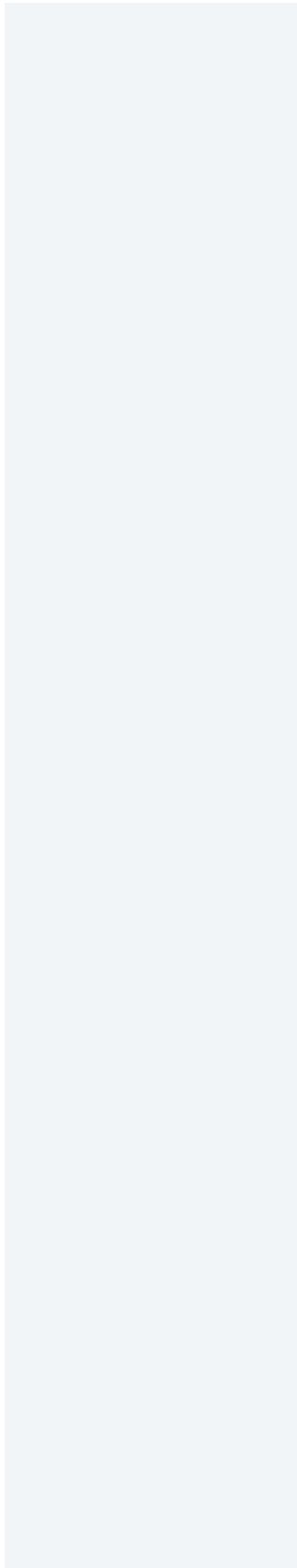
## 1.2.2 POWER OFF

1. Press the CNC OFF push button. This button shuts down the controller.
2. Turn OFF the main breaker. It isolates the machine from external power source.

To power off the machine, turn the power switch in the counterclockwise direction.



Figure 1-8: Power OFF



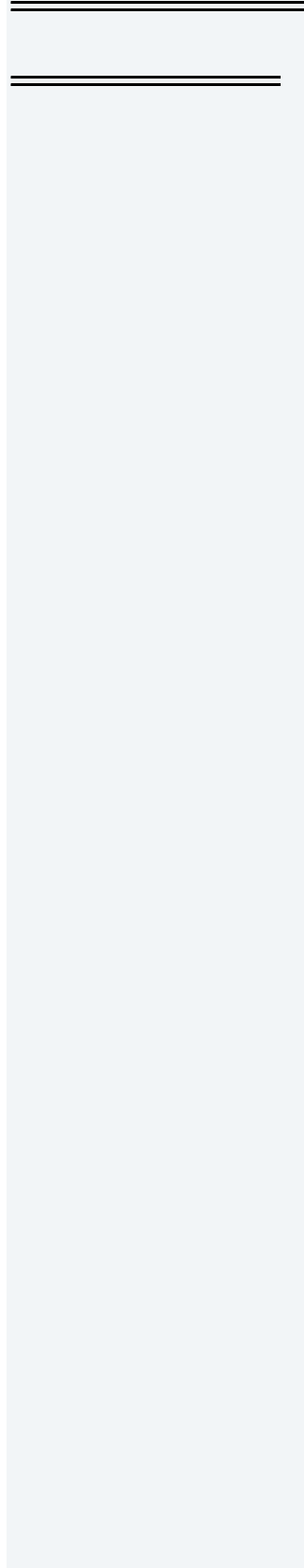
---

---

## 2.0 OPERATOR PANEL LAYOUT

---

---



## 2.1 MAIN OPERATOR PANEL

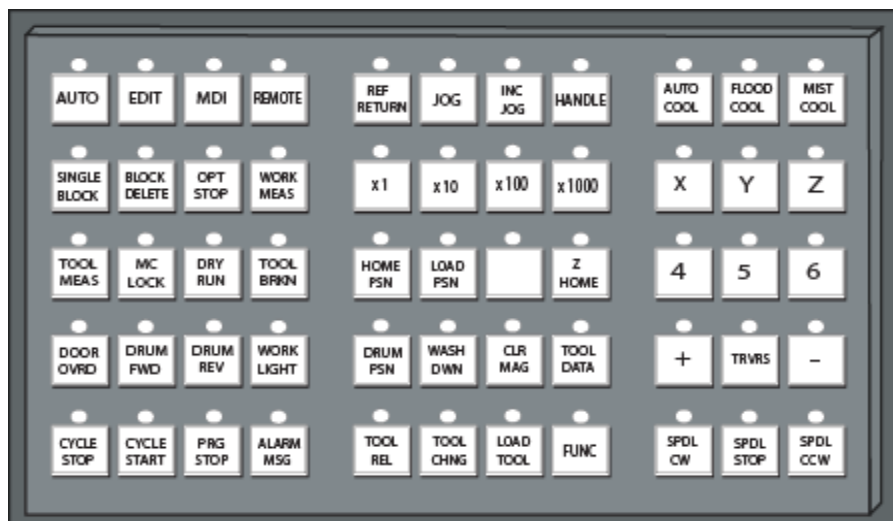


Figure 2-1: Main B Operator Panel layout

GE Fanuc 18i-MB5 control is equipped with “Main B” Operator Panel. 55 Push button layout is pictured above.

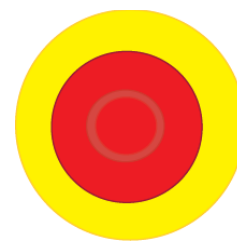
Clear key caps and custom legends have been installed as presently defined for machine use. Unless otherwise noted, buttons fit with blank key caps (no legend) are not defined for end-user use.

## 2.2 SAFETY FUNCTIONS

### 2.2.1 EMERGENCY STOP SWITCH

If you press Emergency Stop button on the machine operator’s manual, machine movement stops in a moment. This button is locked when it is pressed, and can be unlocked by twisting it CW.

When the emergency stop is pressed, the emergency stop command is applied to the machine, and the CNC is reset, spindle and XYZ axes are stopped, and the other actions are interrupted. CNC will display EMG STOP message on the screen.



### 2.2.2 DOOR OVRD

This push button enables opening machine doors without generating Feed Hold & Cycle Start inhibits. While door override is applied, LED is flashing & Operator Message (2006 Door open override is active) is displayed to remind that override is active.



Operation of Door Override push button is momentary. Momentary function requires operator to hold push button while door override is required.

## 2.2.3 WORK LIGHT

Operator control of machine's work light(s). Pressing push button toggles work light On/ Off.

Machine work light is set to automatically turn on with power-up of machine.



## 2.2.4 ALARM MSG

Pressing this button will reset the CNC and clear the PMC message.



## 2.3 OPERATION MODE SELECTION

### 2.3.1 AUTO

Auto (Memory) Mode (Fanuc operator manual GFZ-63534EN Section 4.1)

Auto Mode is also called Memory Mode. Automatic operation of part program selected from program files registered in control's program directory.



### 2.3.2 EDIT

Edit Mode (Fanuc operator manual GFZ-63534EN, Section 9)

Edit mode enables entering and editing of part programs stored in control's part program directory. Part programs stored on optional Data Server or memory card inserted in PCMCIA card slot are not available for editing. Programs must be edited before loading to storage media.



### 2.3.3 MDI

Manual Data Input Mode (Fanuc operator manual GFZ-63534EN, Section 4.2)

MDI mode is used for simple test operation. In the MDI mode, operator can create and execute a program consisting up to 10 lines from the MDI panel, which is in the same format as the normal program.



### 2.3.4 REMOTE

Remote Mode (Fanuc operator manual GFZ-63534EN, Section 4.3, 4.13)

Remote mode is also called DNC mode. In the mode, it is possible to perform machining while a program is being read in via reader/puncher interface, or remote buffer. Operator can, also, perform machining with execution of the program in the memory card, which is installed in the memory card interface located on the left side of the screen.



### 2.3.5 REF RETURN

Reference Return Mode (Fanuc operator manual GFZ-63534EN, Section 3.1)

Reference Return gives an opportunity to return all the axes to the machine zero position.



## 2.3.6 JOG

Continuous Jog Mode (Fanuc operator manual GFZ-63534EN, Section 3.2)

In the continuous jog mode, pressing the direction switch on the operator's panel moves the tool along with the selected axes in the selected direction.



## 2.3.7 INC JOG

Incremental Jog Feed Mode (Fanuc operator manual GFZ-63534EN, Section 3.3)

In the incremental mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction.



## 2.3.8 HANDLE

Handle (MPG) Mode (Fanuc operator manual GFZ-63534EN, Section 3.4)

In the handle mode, operator can move the axes by rotating the manual pulse generator on the operator's panel.





---

---

## 3.0 MANUAL OPERATION

---

---

## 3.1 MANUAL OPERATION MODES

### 3.1.1 MANUAL REFERENCE POSITION RETURN

#### Reference Zero:

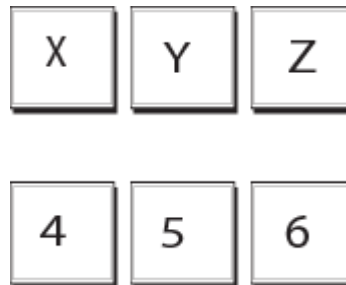
Manual reference position return is to move the tool to the reference position using switches and push buttons located on the operator's panel. Fadal machines are equipped with Absolute Encoders. Stored position information from the encoders will initialize the current machine position at power-up. When position is established by absolute feedback, REF RETURN MODE will simply command axes to Home position.

#### Reference Procedure (for the machines without scale option)

1. Select REF RETURN Mode.



2. Select one axis to be referenced by X, Y, Z, 4, or 5 axes button.



Machine axes will move at traverse rate to home position. When each axis has completed finding the reference position, the related machine coordinate will become to zero. Press the JOG, JOG INC, HANDLE or AUTO, MDI, and EDIT mode keys to exit the Reference mode.

Pressing the POS key on the MDI panel will switch to different position display which includes machine coordinate, absolute coordinate, and relative coordinate.

For referencing of machine axes, including initialization of absolute encoder positions, please see separate maintenance procedure in Appendix.

#### NOTE

FADAL machines do not use reference deceleration switches for establishing machine home position. Dog-less reference operation is provided for referencing axes without reference switches. Maintenance instructions for dog-less reference must be followed exactly, otherwise machine home position will not be repeatable.

**Reference Procedure (for the machines with distance-coded scale option)**

If VMC is equipped with **Distance Coded Scale**, operator needs to do manual reference position return after powering ON the VMC according to the procedure below:

- Press the **REF RETURN** key.
- Press **X/Y/Z/4/5** push button on the operator panel A and wait until the reference moving is finished.
- Press **JOG** key.
- Press the **REF RETURN** key.
- Press **X/Y/Z/4/5** push button on the operator panel A and **X/Y/Z/4/5** axes will move to the machine reference position.

## 3.1.2 JOG OPERATION

1. Select **JOG** Mode.



2. Select one axis to be manually jogged by **X, Y, Z, 4** or **5** axes buttons. Axis selection button does not have to be held down. LED will light indicating selected axis. Pressing any axis button will automatically de-select previously selected axis. It is implemented to prevent jogging of more than one axis simultaneously.



3. Press and hold **+** or **-** direction button for desired direction of travel. Axis motion will continue until direction button is released. Direction LED will be lit during motion.



Machine axes will travel at jog rate 200 inch per minute. Feedrate Override rotary switch may be used to vary jog rate.

4. Pressing **TRVRS** during continuous jog move will increase axis jog rate to traverse speed (MAX 900IPM for XY axis, 700IPM for Z axis). Feed rate rotary switch is also used to override traverse rates.

## 3.1.3 INC JOG OPERATION

### NOTE

If the feedrate is at 0%, no motion will occur when using the "+/-" jog button. 6030, 8030 have the Max 400IPM for X, Y, Z.

Each time a +/- button is pressed, the tool moves by the predetermined distance.

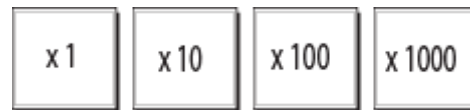
1. Select **INCR JOG** Mode.



2. Select one axis by **X, Y, Z, 4** or **5** axes buttons, as in JOG mode.



3. Select desired increment of travel by using **X1, X10, X100** or **X1000** push buttons.



4. Press **+** or **-** direction button for desired direction of travel. Incremental axis motion of selected distance will be commanded per each press of direction button. Direction LED will be lit during motion.



Incremental travel selection by multiplier setting (X1, X10, X100 or X1000) is based on the minimum command increment used by FADAL machine, 0.0001 inch or 0.001 mm. Selection of X1 will command 0.0001" or 0.001mm for each press of direction button. X10 commanding 0.001" or 0.01mm, (10 x 0.0001" or 10 x 0.001mm) and so on.

- X1: 0.0001 inch (0.001mm)
- X10: 0.001 inch (0.01mm)
- X100: 0.01 inch (0.1mm)
- X1000: 0.1 inch (1.0mm)

## 3.1.4 HANDLE OPERATION

Handle mode is also referred to as MPG Mode (Manual Pulse Generator). By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

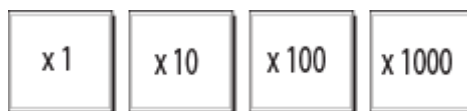
1. Select **Handle** Mode.



2. Select one axis by **X, Y, Z, 4** or **5** axes buttons, as in JOG mode.



3. Select desired increment of travel by using **X1, X10, X100** or **X1000** push buttons.



X1: 0.0001 inch (0.001mm)

X10: 0.001 inch (0.01mm)

X100: 0.01 inch (0.1mm)

4. Rotate Manual Pulse Generator (MPG) clock-wise or counter clock-wise for plus or minus motion, respectively. For each detent or "click" of MPG handle, one selected increment or travel will be commanded. Direction LED will be lit during motion.



### NOTE

Increment travel selection by multiplier settings are used same as during INCR JOG mode. For example, selection of X10 increment with clock-wise rotation of 15 "clicks" on MPG will command 0.0015" or 0.015mm of travel in plus direction.

When MPG hand wheel is rotated at a rate fast enough to exceed the axis traverse rate, the axis speed is clamped at the traverse rate and excess rotation of the hand wheel is ignored. In this case, the distance of axis travel will not equal the amount of hand wheel rotation.

## 3.1.5 JOG FEED RATE

When the machine is in the JOG mode, the jog feedrate varies from 0 IPM to 240 IPM since the feedrate switch changes the feedrate from 0% to 120%. For example, if Jog Feedrate Switch is at 100%, the feedrate changes to 200IPM.

This switch affects the rapid speed since it is rapid speed override switch also.

When the Handy Machine Operator's Panel (HMOP) operation ON/OFF switch is turned off, the effective feed override/ feedrate is only related with main operator panel feed override switch.

When the operation ON/OFF switch is turned on, the effective feed override/feedrate value is that HMOP override switch's value multiply main operator panel feed override switch's value.



### NOTE

If the feed override/feed rate is at 0%, no motion will occur when using the "+/-" jog button or executing the program.

---

---

## 4.0 AUTOMATIC OPERATION

---

---

## 4.1 AUTOMATIC OPERATION MODES

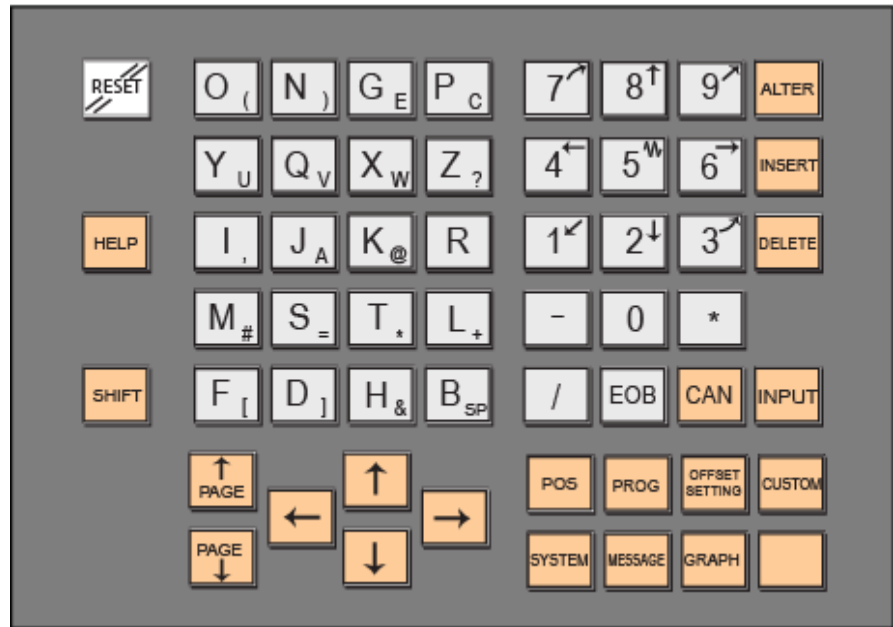


Figure 4-1: MDI Panel

### 4.1.1 MDI OPERATION

In the MDI mode, a program consisting of up to 10 lines can be created in the same format as normal programs and executed from the MDI panel. MDI operation is used for simple test operations.

In order to complete this operation, follow the next procedure:

1. Press the MDI mode selection switch.
2. Press the **PROG** function key on the MDI panel to select the program screen.
3. Prepare a program to be executed by an operation similar to normal program editing. M30, specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode.
4. To entirely erase created program in the MDI mode, use one of the following methods:
  - Enter address, and then press the **DELETE** key on the MDI panel.
  - Alternatively, press the **RESET** key.
5. To execute a program, set the cursor on the head of the program. Push **CYCLE START** button on the operator's panel. By this selection, the prepared program will start. When the program end (M02, M30) or ER (%) is executed, the program will



#### 4.1.2 MEMORY OPERATION (AUTO MODE)

be erased and the operation will end. By command of M30, control returns to the head of the prepared program.

6. To stop or terminate MDI operation in midway through, follow the next steps:

- Stop MDI operation.

Press the CYCLE STOP switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off.

- Terminate MDI operation.

Press the reset key on the MDI panel. Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.

When the CYCLE STOP switch in the machine operator's panel is pressed during automatic operation, this operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.

When the **RESET** key on the MDI panel is pressed, automatic operation terminates and the reset state is entered.

To complete this operation follow the next procedure:

1. Press the **MEMORY (AUTO)** mode selection switch.
2. Select the program from the registered programs doing the following steps:
  - Press **PROG** to display the program screen.
  - Press **“+”** soft key.
  - Press **“DIR”** soft key and the program library will be displayed.
  - Enter a program number using the numeric keys.
  - Press the **O SRH** soft key.
3. Press the CYCLE START switch on the machine operator's panel. Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.
4. To stop or cancel memory operation midway through, follow the steps below:
  - Stopping memory operation.

Press the CYCLE STOP switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:

4.1.3 DNC  
OPERATION

When the machine was moving, feed operation decelerates and stops.

When dwell was being performed, dwell is stopped.

When M, S, or T was being executed, the operation stopped after M, S, or T is finished.

- Terminating memory operation.

Press the **RESET** key on the MDI panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

By activating automatic operation during the DNC operation mode (**REMOTE**), it is possible to perform machining (DNC operation) while a program is being read in via reader/puncher interface, or remote buffer.

To use the DNC operation function it is necessary to set the parameters related to the reader/punch interface and remote buffer in advance.

To complete this procedure follow the next procedure:

1. Search for the program to be executed.
2. Press the **REMOTE** switch on the machine operator's panel to set **REMOTE** mode, then press cycle start switch. The selected file is executed.

During DNC operation, the program currently being executed is displayed on the program check screen and program screen.

The number of displayed program blocks depends on the program being executed.

#### DNC Operation procedure with memory card:

1. Set the parameter of No.0020 to 4 in the setting screen in advance.
2. Change to **REMOTE** mode.
3. Press **PROG** function key on the MDI panel.
4. Push "+" soft key twice.
5. When "**DNC-CD**" soft key is pressed, the following screen is displayed.

The screen can be scrolled by page key. An arbitrary file number is input, and "**F SRH**" soft key is pressed. Then the arbitrary file name is displayed at the top of DNC operation (memory card) screen.

6. Input the file number which is going to be executed.

#### 4.1.4 MEMORY PROTECTION KEY

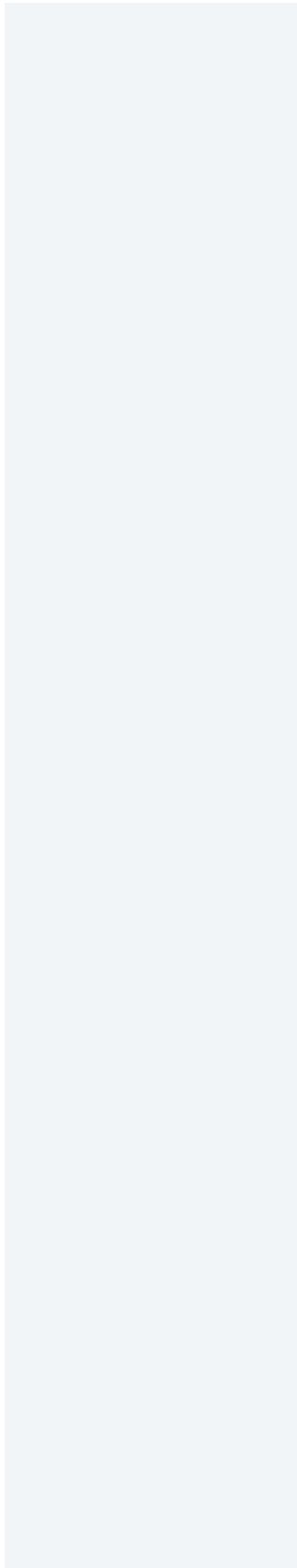
7. Press the “**DNC-ST**” soft key, the file name will be displayed in the right side of **DNC FILE NAME:**

For example, DNC FILE NAME:O0053

8. Press the **CYCLE START** button to execute the program selected.

This key is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously.

When the key is in the On position “**I**”, the memory is protected and operator can't change any data in the CNC. When the key is in the Off position “**O**” operator is able to register/modify/delete the part programs, offset values, parameters and setting data.



---

---

## 5.0 TEST FUNCTION

---

---

## 5.1 TEST FUNCTION MODES

### 5.1.1 SINGLE BLOCK

Single Block (Fanuc Operator Manual GFZ-63534EN, III, Section 5.5)

Single Block Mode executes part program block by block. Single block mode is implemented to toggle On/Off with press of button.



Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after executing a single block in the program. Check the program in the single block mode by executing the program block by block.

### 5.1.2 BLOCK DELETE

Block Delete (Fanuc Operator Manual GFZ-63534EN, II, Section 12.2)

Skip execution of program Block (/). Multi level Block Skip is not supported. Block Delete is implemented to toggle On/Off with press of button.



### 5.1.3 OPTION STOP

Optional Stop on M01 (Fanuc Operator Manual GFZ-63534EN, II, Section 11.1)

Option Stop is implemented to toggle On/Off with press of button. Executing program will stop at M01 when option stop button is On. Operator needs to push CYCLE START button to restart the program. It does not affect the program when the option stop button is Off



### 5.1.4 PROGRAM STOP

Program Stop (Fanuc Operator Manual GFZ-63534EN, III, Section 40.1)

Program Stop indicator LED is lit when part program execution is stopped by M00 or M01 part program.



### 5.1.5 MC LOCK

Machine Lock (Fanuc Operator Manual GFZ-63534EN, III, Section 5.1)

Machine Lock (Test Mode) enables execution of part program without axis motion, but M/S/T command still is able to execute. This button is for test purposes. Machine lock is implemented to toggle On/Off with press of button.



### 5.1.6 DRY RUN

Dry Run Feed Rate (Fanuc Operator Manual GFZ-63534EN, III, Section 5.4)

This feed rate forces program feedrate to fixed "dry run" rate to speed non-cutting testing of part programs. Dry Run is implemented to toggle On/Off with press of button except AUTO and REMOTE mode. If the machine is in AUTO or REMOTE mode, operator can turn On DRY RUN by pressing **FUNC + DRY RUN** buttons and turn it Off by pressing **DRY RUN** button. The tool is



moved at the feed rate specified by a parameter regardless of the feed rate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table. Press the Dry Run switch on the machine operator's panel during automatic operation. The tool moves at the feed rate 900/700 IPM when the feed rate is overridden by 100%. The rapid traverse switch can also be used for changing the feed rate. The Dry Run feed rate changes as shown in the table below according to the rapid traverse switch and parameters.

Table 5-1: For X and Y axis, when the DRY RUN button is "ON".

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	700 IPM	500 IPM
OFF	900* Feedrate override IPM	500* Feedrate override IPM *1

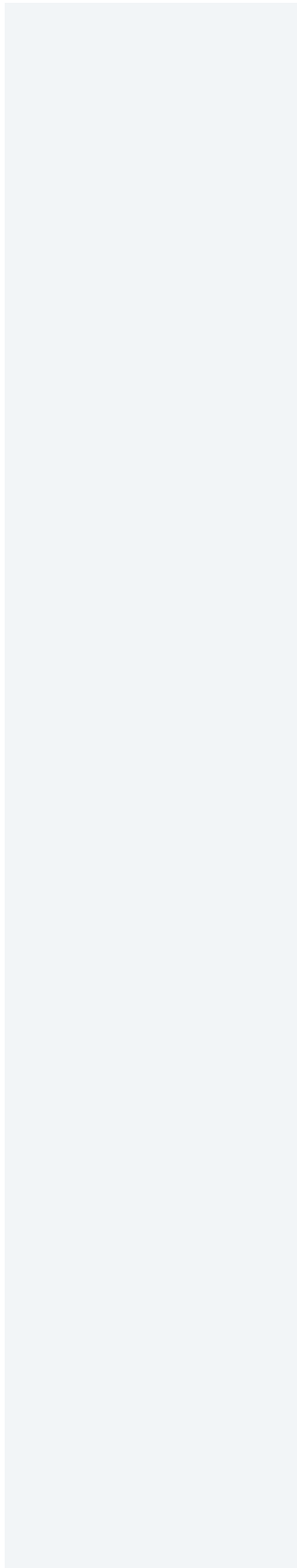
Table 5-2: For Z axis, when the DRY RUN button is "ON"

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	700 IPM	500 IPM
OFF	700* Feedrate override IPM	500* Feedrate override IPM *1

\*1 Speed is Clamped to maximum feedrate 500 IPM.

**NOTE**

When the machine is in the Auto/Remote mode, operator can only turn on Dry Run by pressing Dry Run & Func key at the same time.





---

---

## 6.0 MACHINE SETUP FUNCTION

---

---

## 6.1 SETUP MODES

### 6.1.1 WORK MEAS

Work Offset Setting (Fanuc Operator Manual GFZ-63534EN, III, Section 11.4.14)

This measurement simplifies setup of work/fixture offsets using Measurement B feature. Please refer the Fanuc operator manual for the details.



### 6.1.2 TOOL MEAS

Tool Length (Fanuc Operator Manual GFZ-63534EN, III, Section 11.4.14)

This measurement simplifies setup of tool length offsets using Measurement B feature. Please refer the Fanuc operator manual for the details.



### 6.1.3 FUNCTION

Function button

Horizontal mounting of Operator Panel in Fadal pendant exposes operator buttons to a more varied environment. To prevent accidental activation of selected operator panel operations, the **FUNC** button is required to be pressed in combination with desired operator panel function.



Operator Panel functions currently requiring **FUNC** button are:



### 6.1.4 HOME PSN

Macro Program Positioning Axes to Home Position

Pressing this button will call the macro program which commands axes to machine position of X0.0 Y0.0 Z0.0 using G53 traverse moves. Program will command Z-axis to 0.0 first, before X & Y axes motion are commanded.



**FUNC** button must be pressed simultaneously with **HOME PSN** to execute program.

### 6.1.5 LOAD PSN

Macro Program Positioning Axes to Table Load PSN

Pressing this button will call the macro program which commands axes to machine position of X0.0, Z0.0, and Y plus limit position using G53 traverse moves. Table position is in



## 6.1.6 Z HOME

front and center. When control is set for metric operation, macro program will automatically command to X0.0, Y+Limit, Z0.0 in mm units.

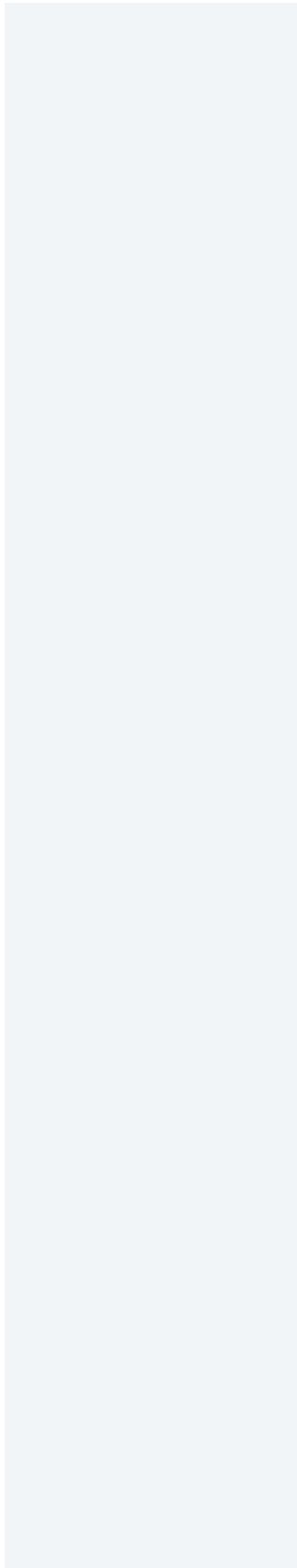
**FUNC** button must be pressed simultaneously with **LOAD PSN** to execute program.

Macro Program Positioning Axes to Z axis home position.

Pressing this button will call the macro program which commands Z-axis to machine position of Z0.0 using G53 traverse move.



**FUNC** button must be pressed simultaneously with **Z HOME** to execute program.



---

---

## 7.0 TOOL OPERATION

---

---

## 7.1 MANUAL OPERATION

### 7.1.1 DRUM FWD

Manual Index of Tool Drum in Forward Direction

Commands are manually jogging or indexing of the tool drum in forward direction. Forward direction is defined as indexing of tool pots in rising order:... 23, 24, 1, 2, 3,... Rotation is counter clock-wise when looking at the back of tool drum from the operator station.



Jog Mode is required. Operator may hold down button for indexing of multiple tool pots, or press momentarily to index one tool. **DRUM PSN** button can be used to display tool pot at tool change (bottom) position. Operator message will update while tool carousel is being manually rotated.

### 7.1.2 DRUM REV

Manual Index of Tool Drum in Reverse Direction

Commands are manually jogging or indexing of the tool drum in reverse direction. Reverse direction is defined as indexing of tool pots in decreasing order:... 3, 2, 1, 24, 23,... Rotation is clock-wise when looking at the back of tool drum from the operator station.



Jog Mode is required. Operation is same as used for **DRUM FWD**.

### 7.1.3 DRUM PSN

Display of Tool Drum Position by Operator Message

Use of this button activates display of Operator Message containing tool pocket number at tool change (button) position of carousel.



**DRUM PSN** message is set to automatically turn on with power-up of machine. Position of Tool Drum is maintained in battery backed PMC Data table while machine is powered down.

Operator may turn the message On/Off at any time using **DRUM PSN** push button.

### 7.1.4 TOOL REL

Tool Release (Draw Bar Open)

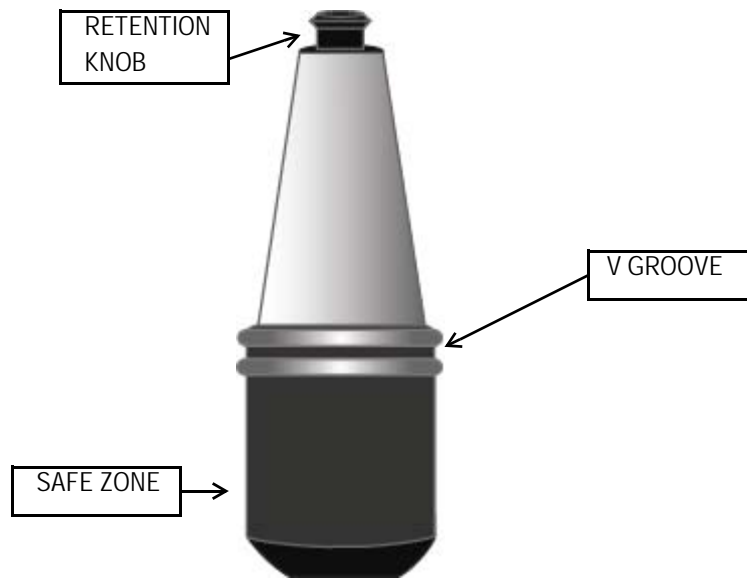
This button is for manually loading/uploading a tool holder. Operator must be prepared to catch tool from spindle cartridge immediately upon pressing button. Spindle air is turned on along with release of tool. Draw bar will open after pressing the button more than 1 second and remain open while push button is still held.



Spindle must be stopped with control in JOG or HANDLE modes to enable tool release.

**Unloading tool procedure:**

1. The tool holder must be held in the left hand with the thumb and the first finger grasping the holder below "V" groove. No other fingers should have contact with the holder or the tool in the holder. The area below the "V" groove is called the safe zone. The safe zone is the only place where the tool holder should be held.



2. Press the **TOOL REL** button. Keep the **TOOL REL** button pressed until the tool is completely out of the spindle.

**Loading tool procedure:**

1. The tool holder must be held in the left hand with the thumb and the first finger grasping the holder below the "V" groove. No other fingers should have contact with the holder or the tool in the holder. The area below "V" groove is called the safe zone. The safe zone is the only place where the tool holder should be held.
2. Place the holder into the spindle after pressing the **TOOL REL** button, not before. The keys on the nose of the spindle must fit into the key-ways on the tool holder flange.
3. Release the **TOOL REL** button to lock the tool into the spindle.

**NOTE**

When loading a holder into the spindle, inspect the taper for chips. Remove any chips from the taper with a flat stone. Confirm the retention knob is securely tightened before placing the tool in the spindle.

After loading a tool holder into the spindle, operator needs to edit the magazine management to write the spindle tool number into the SPDL1 column, and then use the **TOOL CHNG** to load the spindle tool into the magazine/carousel.

7.1.5 **TOOL CHNG**  
(DATC ONLY)

Macro Program Execute Tool Change

**FUNC** button must be pressed simultaneously with **TOOL CHNG** to execute program.



Pressing this push button will execute macro program O8988. Program executes swap of tool between spindle cartridge and current position of **TOOL DRUM**.

This button is for manually unloading of tools between spindle and tool drum, or loading of tool to specific tool drum pot. Operator can position tool drum prior to use of **TOOL CHNG** button.

During tool loading, operator may input tool number into spindle window on Tool Magazine screen. Tool Change cycle will move tool number to magazine. Alternatively, operator may edit magazine table once tools are loaded. Upon removal of tool from spindle, operator should delete tool number from spindle window on Tool Magazine screen.

7.1.6 **TOOL LOAD**

Macro Program for Loading Tools

**FUNC** button must be pressed simultaneously with **LOAD TOOL** to execute program



**Machine with DATC:**

Pressing this push button will execute the macro program O8989. Program searches for and positions to empty tool pot in tool drum. Then executes swap of new tool from spindle cartridge into empty tool pot.

This function is provided for manual tool loading from to tool drum. Operator to input tool number from spindle window on Tool Magazine screen so that tool number can be loaded to magazine, or edit magazine and spindle tool data after tools are loaded.

**FUNC+LOAD TOOL** procedure

- Manually load the tool to the spindle according the **TOOL REL** procedure.
- Press **JOG** or **HANDLE** mode.
- Press **OFFSET/SETTING** key on the MDI panel.
- Press + soft key below the LCD screen.
- Press **TL-MNG** soft key.
- Press **MAG** to enter the tool data management screen.



- Press **OPRT** soft key.
- Press **EDIT** soft key, the EDITING will be displayed on the screen.
- Move the cursor to the most right column: **SPDL1**.
- Input the tool number of the current spindle tool into the **Spindle Tool** column.
- Press **INPUT**, the tool number will be displayed in the **SPDL1** column.
- Press **OPRT**.
- Press **END** to end the tool management editing.
- Press **FUNC+LOAD TOOL**, tool will be loading the magazine automatically, and the **MAG** management will be updated.

**NOTE**

The Operator needs register the tool into the management first before load into the new tool.

**Machine with ATC:**

Press the **TOOL LOAD** push button with **FUNC** button will execute the macro program O8989. Program positions to specific tool pot in tool drum, and then load the spindle tool into the specific tool pot.

This function is provided for manually loading tools from spindle to tool carousel. Operator input tool number from spindle window on Tool Magazine screen so that tool number can be loaded to magazine, or edit magazine and spindle tool data after tools are loaded.

**FUNC** button must be pressed simultaneously with **LOAD TOOL** to execute program.

**FUNC+LOAD TOOL** procedure

- Manually load the tool to the spindle according to the **TOOL REL** procedure.
- Press **JOG** or **HANDLE** mode.
- Press **OFFSET/SETTING** key on the MDI panel.
- Press + soft key below the LCD screen.
- Press **TL-MNG** soft key.
- Press **MAG** to enter the tool data management screen.
- Press **OPRT** soft key.
- Press **EDIT** soft key, the EDITING will be displayed on the screen.
- Move the cursor to the most right column: **SPDL1**.
- Input the tool number of the current spindle tool into the **Spindle Tool** column.
- Press **INPUT**, the tool number will be displayed in the **SPDL1** column.
- Press **OPRT**.
- Press **END** to end the tool management editing.
- Press **FUNC+LOAD TOOL**, tool will be loading the magazine automatically, and the **MAG** management will be updated.

**NOTE**

Operator need register the tool type into the management first before load the new tool. If the tool management screen, MAG has the same tool already, operator should clear this magazine pot using EDIT and take the tool out from that magazine manually first before try to load another tool.

7.1.7 REFERENCE THE DRUM (ONLY FOR DATC)

1. Press **REF RETURN** button.
2. Press **X1** button. The drum will automatically move to the number 1 pot in the 6 o'clock ready position.

7.2 AUTO OPERATION

7.2.1 T-COMMAND (TOOL NUMBER)

Eight digit T-Word may be programmed in block with or without M06 code.

T-Word programmed in block by itself will position tool carousel with position tool carousel with programmed tool at 6 o'clock position (DATC) for next tool change. Rotation of carousel will not inhibit continued execution of part program (FIN) as carousel is away from work area. For DATC, this enables cycle time reduction of part program execution, by allowing next tool to be immediately ready for tool change.

Programming T-code of tool that is already at tool change (6 o'clock) position will command no activity by tool carousel, as requested tool, is already at tool change position.

T-Word may be used to position tool carousel by pocket number when combining with M100 miscellaneous code.

M100T5: Position carousel to tool pocket #5. Pocket #5 will be selected regardless of tool number in pocket, or if pocket empty.

7.2.2 TOOL CHANGER COMMAND

**ATC:**

M06 Txx command from part program executes tool change cycle for carousel type ATC tool changer. M06 command calls macro program O9021, which executes required Z-axis motion commands & special Miscellaneous Codes (M- Codes) for execution of the tool change cycle.

*M06 T10: Proper Command Syntax*

*M06: Illegal, Macro Error 3020 will result*

*T05 M06: Illegal, Macro Error 3020 will result*

**WARNING!**

1. Special M-Codes contained in macro program O9021 must never be used in end-user part programs without specific permission of Fadal Machining Centers. Any use other than in O9021 macro cycle is the responsibility of the machine user/operator/programmer. Machine damage can occur.
2. End-user should always refer to Fadal Machining Centers Operator Manual: applicable to this machine, for proper M-Codes to be used during part programming.
3. Macro program O9021 must never be edited without permission of Fadal Machining Centers. Machine damage can occur.

**DATC:**

M06 command from part program executes tool change cycle for "dual arm" type DATC tool changer. M06 command calls program O9020, which executes required Z-axis motion commands & special Miscellaneous Codes (M-Codes) for execution of the tool change cycle.

T-Code may be programmed alone, in previous block to provide positioning of TOOL DRUM prior to tool change. This method improves part program cycle time.

*T10: Tool Drum positions, following operations will continue  
while drum.*

*G01 X . . ., Positions to Tool from pocket containing T10 tool.*

*. . . ,*

*. . . ,*

*M06: Tool change. T10 will be inserted into spindle, old tool to  
empty pocket. D and/or H codes must be added after tool  
change for offsets as required.*

Programming M06, without programming a new tool number, will not command a swap of tools between the carousel and spindle. When no new tool number has been specified, and the active tool is already in the spindle, it is expected that no tool change is required.

**WARNING!**

1. Special M-Codes contained in macro program O9020 must never be used in end-user part programs without specific permission of Fadal Machining Centers. Any use other than in O9020 macro cycle is the responsibility of the machine user/operator/programmer. Machine damage can occur.
2. End-user should always refer to Fadal Machining Centers Operator Manual: applicable to this machine, for proper M-Codes to be used during part programming.
3. Macro program O9020 must never be edited without permission of Fadal Machining Centers. Machine Damage could occur.

## 7.3 TOOL MANAGEMENT

### 7.3.1 CLR MAG

Macro Program Initial Tool Magazine Data

**FUNC** button must be pressed simultaneously with **CLR MAG** to execute program.



Pressing this push button will execute macro program O8986 which initiates tool data from magazine. Tool numbers in all tool drum pots are initialized to the related pocket number. Tool number in the spindle is initialized to "0" indicating no tool in the spindle.

So, operator should take the tool off from the spindle after execution of this function.

Operator can only execute this function in the JOG/INC JOG/HANDLE mode and when the door is closed or door override is ON.

**NOTE**

Please take out the spindle tool after execute this function since this function clears the spindle tool number to zero. Ignoring this note to take out the tool from the spindle will cause tool changer crash.

### 7.3.2 TOOL DATA

Macro Program Loads Tool Management Data

**FUNC** button must be pressed simultaneously with **TOOL DATA** to execute program.



Pressing this push button will execute macro program O8987 which demonstrates loading of tool management data by part program command. Tool numbers 1-24 (1-maximum of tool pocket numbers) are loaded as well as tool groups demonstrating tool life setting by count and by time. The default setting is not tool management for those tools of this function. Operator can only execute this function in the JOG/INC JOG/HANDLE mode and when the door is closed or door override is ON.

**NOTE**

User may setup the tool management function by manually setup of the tool management screen according their requirement.

## 7.3.3 TOOL BRKN

## Tool Broken

**FUNC** button must be pressed simultaneously with **TOOL BRKN** to execute program.



Pressing **TOOL BRKN** and **FUNC** button at the same time is a shortcut to mark active tool as broken/ damaged in the tool management. This function enables operator to mark tool as damaged without editing Tool Management Data. Tool management will not select a damaged tool for use, from the tool group.

7.3.4 EDIT THE TOOL/  
MAGAZINE LIST

## Edit the tool data:

- Press **MDI** button.
- Press **OFFSET/ SETTING**.
- Press "+" soft key.
- Press **TL-MNG** soft key.
- Press **TOOL** soft key to enter the tool data page.
- Press **OPRT** soft key.
- Press **EDIT** soft key to start editing the tool data.
- Press **OPRT** soft key gain after editing is finished.
- Press **END** soft key to exit tool data the editing mode.

## Edit the Magazine data:

- Press **MDI** button.
- Press **OFFSET/SETTING**.
- Press "+" soft key.
- Press **TL-MNG** soft key.
- Press **MAG** soft key to enter the magazine data page.
- Press **OPRT** soft key.
- Press **EDIT** soft key to start editing the magazine data.
- Press **OPRT** soft key again after editing is finished.
- Press **END** soft key to exit magazine data the editing mode.

7.3.5 TOOL ORDER  
PROCEDURE  
(ONLY FOR ATC)  
(M11)

M11command is to define current pocket as pocket 1 for ATC.

In JOG mode, use DRUN FWD/REV button position carousel to new tool position 1, then operator can used M11 to difinite pocket as new pocket position 1.

When you execute M11 command, you may face these two situations:

- I. If spindle has a tool number in the tool management screen, CNC will display an alarm on the screen as soon as you execute M11. If you press MESSAGE button

on the MDI panel, screen will display the following alarm and messages. M11 will stop by alarm 1081 and operator has to return spindle tool to ATC and re-execute this command again.

ALARM	1081 M11 ALM, RETURN/CLR SPDL TOOL
MESSAGE	2056 M11, USING FUNC+LOAD TOOL BUTTON TO RETURN SPDL TOOL TO ATC FIRST
	2058 M11 ALM, TAKE SPDL TOOL OFF, CLR SPDL TOOL MANAGMENT TO 0.

**Table 7-1: Actions**

METHOD 1	METHOD 2
<ol style="list-style-type: none"> <li>1. Press RESET button to reset alarm (mentioned above) first.</li> <li>2. Press JOG button on the operator panel to enter JOG mode.</li> <li>3. Press TOOL REL button to unload spindle tool.</li> <li>4. Edit tool management to change spindle tool number to 0.</li> <li>5. Make sure spindle does not have a physical tool</li> <li>6. Execute M11 in the MDI mode and CNC will enter situation 2 as following:</li> </ol>	<ol style="list-style-type: none"> <li>1. Press RESET button to reset alarm (mentioned above) first.</li> <li>2. Press JOG button.</li> <li>3. Presss FUNC+LOAD TOOL buttons to load spindle tool back to ATC carousel.</li> </ol> <p><b>NOTE</b> Spindle tool number of management screen will display 0 after spindle return the tool to ATC carousel. Operator should make sure these actions are correct and spindle does not have a physical tool after this step.</p> <ol style="list-style-type: none"> <li>4. Make sure spindle does not have a physical action tool.</li> <li>5. In JOG mode, use DRUN FRWD/REV button position carousel to new tool position 1.</li> <li>6. Execute M11 in the MDI mode and CNC will enter situation 2 as following:</li> </ol>

II. If spindle has no tool number in the tool management screen, CNC will execute M11 command and setup current pocket to pocket 1. Also, CNC will display following messages on the screen as soon as you execute M11. If you press MESSAGE button on the MDI panel, you will see following messages:

MESSAGE	2055 M11, ATC IS SET POCKET 1
	2059 M11 ALM, OPERATOR MUST EDIT TOOL MANAGEMENT ACCORDING TO REAL CAROUSEL

### 7.3.6 INITIALIZING ATC TOOL MAGAZINE

Operator should go to the tool management screen to edit tool management according physical tool on the carousel. If operator ignore this action, M6 Txx command will pick up wrong tool.

#### Procedure

1. Take spindle tool out.
2. Press DRUM FWD/DRUM REV to move carousel to pocket 1.
3. Press FUNC+CLR MAG.
4. Press FUNC+TOOL DATA.
5. Execute the M11 in the MDI mode.
6. Cycle CNC power.
7. Double check if Tool Management Table is same as Physical ATC.
8. Test the tool changer to make sure M6 Txx command picks up the right tool.

### 7.3.7 TOOL LOADING (ATC)

#### Turret (ATC) open for loading:

1. Change the machine to the **MDI** mode and execute the M150.  
M150 M-Code is provided to emulate Fadal control operation for manual tool change. M150 will return tool in the spindle to carousel pocket and raise Z-axis to clear tools. Carousel in left "out" position, enables operator to rotate carousel to next tool.

#### NOTE

M151 command may be used to pick up next tool, if desired. M150 will position above carousel when no tool is in the spindle to be returned.

2. Change the mode to Jog, and use the **DRUMFWD/DRUM REV** button to rotate the turret.

Operator can press **FUNC+DRUM PSN** button to display the drum current position.

3. Change the machine to the **MDI** mode and execute the M151.

M151 M-Code is provided to emulate Fadal control operation for manual tool change. M151 will proceed from position machine left by M150 cycle. After operator has manually indexed tool carousel to desired new tool, executing M151 will pick up operator-selected tool from carousel and return carousel to home position.

## 7.3.8 TOOL CHANGER (RECOVERY ATC)

### NOTE

When user loads tools into the turret, they can place tools in the turret and use DRUM FWR/ DRUM REV button to put the next tool in the turret until all the tools are placed in the turret. Rotate the turret until the first tool is in the number one position before setting the tool order (using M11 to establish tool order).

### WARNING!

1. M150 is provided for MDI use only to return tool to carousel. Use of the M150/M151 commands is responsibility of the machine operator. Machine damage may occur in unintended use.
2. M151 is provided for MDI use only after M150 command for return of active tool to carousel, and positioning of Z-axis above the carousel. Use of M150/M151 is the responsibility of the machine operator. Machine damage can occur during unintended use.
3. M150/M151 commands should never be used during Automatic execution of part programs.

1. Press the **RESET** to disable the alarm.

### NOTE

If the alarm can not be disable, operator may cycle the power to clear the alarms.

2. Execute the M186 in the MDI mode to get the turret to the home position if the Z axis still is in the home position.

### NOTE

User may move the Z axis to the safe area before return of the slide according to the real situation.

3. Move to the cold start position using either **REF RETURN** or G28.
4. Use the **DRUM FWD/DRUM REV** button to rotate the turret to the number 1.
5. Execute the M11 to establish the drum number 1. Cycle the power.
6. Double check the tool management to see if it is right. Edit it to make it right if it needs.

## 7.3.9 DATC 32-TOOL WITH TOOL CAGE DOOR (6535 40T/ 50T)

Tool Cage door option enables to manually load tool through magazine side. Operator can only operate magazine using remote button on the tool cage door when machine is in the JOG mode and tool cage door is open.

Tool Cage door operation procedure (JOG mode):

1. Stop the machine first.
2. Press the **JOG** button on the operator panel A to enter JOG mode.



3. Turn the Tool Cage door lock to the right 90 degree to open the tool cage door.
4. Press Tool Door Override button and hold it.
5. Press **DRUM FWD/DRUM REV** button to move the magazine to the desired position and release **DRUM FWD/DRUM REV** button.
6. Release Tool Door Override button.
7. Repeat steps above if needed.
8. Close the Tool Cage door.

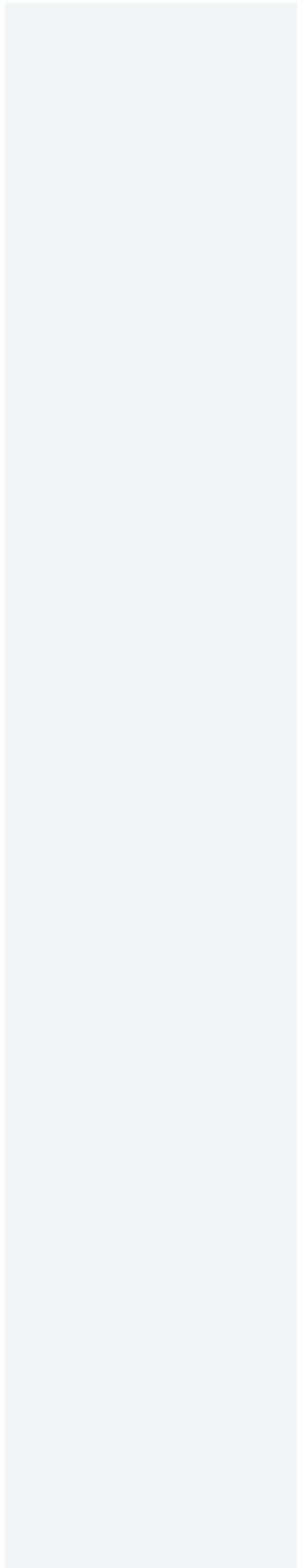
If operator opens the Tool Cage door while machine is running, machine will enter the feed hold mode and magazine movement will be stopped by hardware. Also, screen will display the following message:

[2016 TOOL DOOR IS OPENED, PLEASE CLOSE IT, FEED HOLD IS ON]

Once the Tool Cage door is closed, screen will display another message:

[2068 USING FUNC+CYCLE START BUTTON TO RESTART MAGAZINE]

Then, operator can press **FUNC** and **CYCLE START** buttons to continue the machine cycle or finish the tool change sequence.



---

---

## 8.0 COOLANT OPERATION

---

---

## 8.1 COOLANT CONTROL PUSH BUTTONS

### 8.1.1 AUTO COOL

AUTO COOL push button selects automatic control of coolant by programmed M-Code.



AUTO COOL button toggles on/off, as indicated by LED. While AUTO COOL is active, manual coolant control buttons MIST COOL AND FLOOD COOL are

inhibited. AUTO COOL "OFF" enables control of coolant. AUTO COOL is defaulted to "ON" condition at power-up.

When AUTO COOL is on, the following M-Code is used to control the coolant system:

M-Code:

M7: Mist coolant ON; Coolant through spindle ON

M8: Flood coolant ON

M9: Mist coolant, Flood coolant, coolant through spindle OFF

### 8.1.2 FLOOD COOL

FLOOD COOL turns on/off the flood coolant pump when the AUTO COOL is off.



### 8.1.3 MIST COOL

MIST COOL turns on/off the mist coolant pump and coolant through spindle solenoid when the AUTO COOL is off.



#### NOTE

FLOOD COOL and MIST COOL may be turned on simultaneously. MIST COOL also turns on the coolant through spindle pump if the machine has this option.

---

---

## 9.0 SPINDLE OPERATION

---

---

## 9.1 SPINDLE OPERATION PUSH BUTTONS

### 9.1.1 SPDL STOP

#### Manual Spindle Stop

SPDL STOP applies stop to spindle running in either CW or CCW direction. SPDL STOP button is inhibited when CYCLE START LED is on, indicating automatic cycle is active. SPDL STOP does not cancel active spindle speed "S" word command. SPDL STOP may be used from any automatic or manual mode.



### 9.1.2 SPDL CW

#### Manual Spindle Start Clock-Wise & Spindle Jog CW

**FUNC** button must be pressed simultaneously with **SPDL CW** to execute program.



SPDL SW provides dual functions, based on operating mode of control.

### 9.1.3 SPDL CCW

#### Manual Spindle Start Counter Clock-Wise & Spindle Jog CCW

**FUNC** button must be pressed simultaneously with **SPDL CCW** to execute program.



SPDL CCW, also, provides dual function, based on operating mode of control.

	JOG Mode	MDI, AUTO or REMOTE Mode	
SPDL CW	X		Jog Spindle in CW direction
		X	Spindle Start in CW direction
SPDL CCW	X		Jog Spindle in CCW direction
		X	Spindle Start in CCW direction

#### 9.1.4 MANUAL SPINDLE SPEED

Spindle jog speed is defined in PMC data table. Spindle Speed Override rotary switch is applied to spindle jog speed. Spindle will jog while FUNC+ SPNDL CW/CCW button is held. Spindle will stop when CW/CCW button is released.

#### 9.1.5 M CODE / S CODE

M command:

- M3: Spindle CW rotate.
- M4: Spindle CWW rotate.
- M5: Spindle Stop.
- M41: Spindle low gear range.
- M42: Spindle high gear range.

S command: Spindle rotation speed command

The spindle speed can be specified directly by address S followed by a maximum five-digit value (min-1). The unit is rotation per minute (RPM).

The S command must be specified with M3/M4 command in the same block to run the spindle. Using M5 command or pressing RESET can stop the spindle.

Example: S10000; spindle rotation speed is 10000RPM.  
*M3 S2000 (Spindle CW, 2000RPM).*  
*M4 S2000 (Spindle CCW, 2000RPM).*  
*M5 Spindle Stop*

#### NOTE

Use of FUNC+SPDL CW/CCW as spindle start in MDI, Auto & Remote Modes is inhibited when Cycle Start LED is on, and also by state of door interlock. Programmed "S" word must be active for use of buttons for spindle start. Care should also be used in CW/CCW rotation.

#### 9.1.6 SPINDLE SPEED RANGE

The spindle speed of the Fadal VMC has different configurations.

For EMCD, 2016, 2216, 3016 machines with maximum spindle speed 7.5K, the spindle has a single range 30-7500 RPM.

For 2216, 3016 machines with maximum spindle speed 15K, the spindle has a single range 30-15000 RPM.

For 3020, 4020, 4525, 5020, 6030, 8030, 6535 40T machines with maximum spindle speed 15K with a electrical high/low, the spindle range is as follows:

Low range: 30-2500

High range: 2501-15000

The VMC machines with mechanical Hi/Lo and the maximum speed is 10K, the spindle ranges are as follows:

Low range: 30-2500

High range: 2501-10000

#### 9.1.7 SPINDLE OVERRIDE

Spindle override 50%-120%

This switch will adjust the commanded spindle speed by 50% to 120%.

#### 9.1.8 SPINDLE ORIENTATION

Spindle orientation command for tool changer:

*M190 : Spindle orientation*

Execution of the M5/M3/M4 command or pressing the RESET key will release the spindle orientation command.

*M19: Spindle special orientation command for cycles*

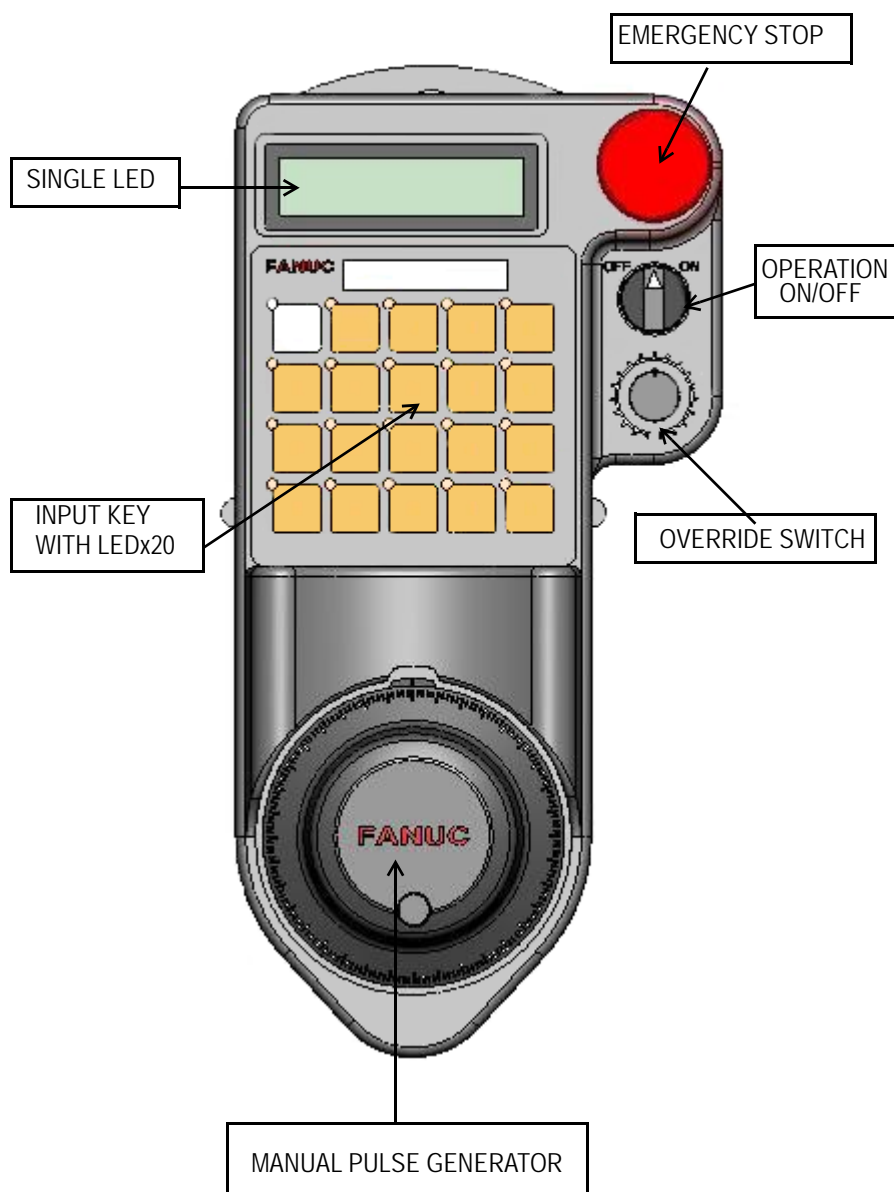
M19 positions spindle aligned with X & Y plane to provide for proper tool alignment during M19 orient cycles which is used during Fine Boring and Back boring canned cycled.

**NOTE**

M19 should be used for the Fine Boring and Black Boring canned cycles only. Using M19 for other purposes may cause machine crash.



10.0 HMOP OPERATION GUIDE



**WARNING! (REFER TO FANUC HMOP MANUAL)**

Please hold the handy machine operator's panel through belt. When operator holds the handy machine operator's panel without using belt, the handy machine operator's panel may be dropped. This switch is not enabled. When operator is surprised with unexpected motion, he should release the enabling switch.

**10.1 HMOP  
OPERATION ON/  
OFF SWITCH**

Turn ON this switch for the purpose of selecting the operation by handy machine operator's panel. Turn off this switch for operation by main operator's panel. When HMOP ON/OFF switch is turned on, the operator is permitted to use machine using HMOP only. If HMOP ON/Off switch is turned off, the operator is permitted to use machine using pendant.

**WARNING!**

When the operation ON/OFF switch is turned OFF, the operation from the main operator's panel is possible, but it is extremely dangerous.

**10.2 OVERRIDE  
SWITCH**

The function of this switch is same as the feed rate switch on the main operator panel. When the operation ON/OFF switch is turned on, the effective override value is that HMOP override switch's value multiply main operator panel feed override switch's value.

Example: The operation switch is on.

Main operator panel feed override is 80%  
 HMOP feed override is 50%  
 The effective feed override for the machine is 40%  
 $80\% \times 50\% = 40\%$

When the operation ON/OFF switch is turned off, the effective feed override is only related with main operator panel feed override switch.

The keys for each of the next procedures are shaded.

## 10.3 MEMORY OPERATION PROCEDURE

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

- Refer the FANUC operator manual to select the program and call it out to memory to run.
- Turn on the HMOP operation ON/OFF switch.
- When the **[MEM]** key is pushed, mode is changed to MEM operation mode.
- When the **[CYCLE START]** key is pushed, automatic operation is started.
- When the **[FEED HOLD]** key is pushed, automatic operation is stopped.
- When the **[RESET]** key is pushed, CNC is reset. Reset and rewind signal should be on for rewinding part program.
- Override can be applied to feed rate speed by the override switch on handy machine operator's panel. Set feed rate override signal from 0% to 120% according to the override switch position.
- For test operation **[DRY RUN]** and **[SINGLE BLOCK]** are available.
- One axis position is displayed normally. When the **[XYZ456]** key is pushed next axis position is displayed.

## 10.4 MANUAL HANDLE FEED PROCEDURE

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

- When the **[HANDLE]** key is pushed, mode is changed to manual handle feed mode.
- The axis, which is moved by MPG, is selected by pushing the **[XYZ456]** key several times.
- When **[Xn]** key is pushed, manual handle feed amount selection is applied. Please select **X1-X10-X100** by pressing several times of this key.

## 10.5 MANUAL JOG FEED PROCEDURE

- When the **[RESET]** key is pushed, the moving axis is stopped.

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

- When the **[JOG]** key is pushed, mode is changed to manual continuous feed mode.
- The axis, which is moved by the **[JOG]** key, is selected by pushing the **[XYZ456]** key several times.
- When the "+" key is pushed, the desired axis moves to positive direction. When the "-" key is pushed, the desired axis moves to negative direction.
- When the **[RAPID]** key is pushed, at the same time, rapid traverse is selected.
- Override can be applied to manual feed rate by the override switch on the handy machine operator's panel. Set manual feed rate override signal from 0% to 120% (0 IPM to 240 IPM) according to the override switch position.
- When the **[RESET]** key is pushed, the moving axis are stopped.

## 10.6 MANUAL REFERENCE RETURN

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

- When the **[REF]** key is pushed, mode is changed to manual continuous feed mode and manual reference position return selection signal "ZRN" is asserted.
- The axis, which is moved for manual reference position return, is selected by pushing the **[XYZ456]** key several times.
- When the "+" key or the "-" key is pushed, the axis moves to the direction of reference position.

















## 10.7 MANUAL SPINDLE CONTROL

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HADNLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

- When [SPINDL START] key is pushed, spindle motor is rotated to the direction which is designed with the speed at the decided override.
- When [SPINDL STOP] key is pushed, spindle motor is stopped.
- When [SPINDL INC] key or the "SPINDL DEC" key is pushed, override of speed is changed. The changing rate is 50% - 120%.

## 10.8 KEY SHEET LAYOUT

FANUC standard key sheet A is as follows. This key sheet is put on the standard unit.

XYZ 456	 REF	 SPINDL START	 SPINDL STOP	 RESET
 HANDLE	 JOG	 SPINDL INC	 SPINDL DEC	+
xn	 MEM	 F1 FUNC TION	 MESSAG	 RAPID
 SINGLE BLOCK	 DRY RUN	 CYCLE START	 FEED HOLD	-

Tool Release Button (F1 Function)

## 10.8.1 TOOL REL

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HADNLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

Same function as the TOOL REL button on the operator panel.

It is used for manually loading/unloading a tool holder use HMOP when the HMOP switch is on. Operator must be prepared to catch tool from spindle cartridge immediately upon pressing button. Spindle air is turned on along with release of tool. Draw Bar will open after pressing the button more than 1 second and remain open while push button is still held. Spindle must be stopped, with control in JPG or HANDLE modes to enable tool release.

**NOTE**

Please refer the operator panel A section for more detail about release function.

## 10.9 MESSAGE KEY

When the "MESSAGE" key is pressed, the screen is changed to user message screen. The next message screen is displayed after another pressing the "MESSAGE" key. When the "XYZ456" key is pressed, the screen returns to the screen according to each mode.

---

---

## 11.0 ROTARY TABLE

---

---

## 11.1 TR 165

11.1.1 MANUAL  
REFERENCE  
RETURN

Once machine is powered ON, execute the following steps:

1. A axis reference procedure is same as X/Y/Z axes because A axis comes with absolute encoder.
  - Press **REF RETURN** button on the operator panel to change machine into Reference mode.
  - Press **4** button on the operator panel and machine will move the A axis to the zero position automatically.
2. B axis comes with incremental scale and operator should obey the following procedure to reference the axis.
  - Press **JOG** button on the operator panel to change the machine into Jog mode.
  - Press **5** button on the operator panel to select B axis.
  - Press **+** button on the operator panel to move the B axis cross the reference mark and move to the positive side of reference mark over 5 degree.
  - Press **-** button on the operator panel to move B axis cross the reference mark and move to the negative side of reference mark over 5 degree.
  - Press **REF RETURN** button on the operator panel to change machine into Reference mode.
  - Press **5** button on the operator panel and machine will move B axis to the zero position automatically.

11.1.2 B AXIS TILT  
TABLE**B axis tilt table directions**

- (B+) = Counterclockwise work piece rotation (viewing in the Y+ direction)
- (B-) = Clockwise work piece rotation (viewing in the X+ direction)

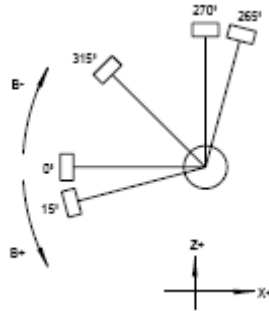
**B axis tilt table limits**

- -95° from the Cold Start Position
- +15° from the Cold Start Position

11.1.3 G90 ABSOLUTE  
MODE

In the absolute mode, the tilt angular value defines the final position between 15° and 265°. The + or - signs define the direction, the table will move to get to the degree of tilt. Care must be used in selecting the proper sign for tilt to prevent over travel.





Example:

In absolute terms B+10 will tilt the B axis counterclockwise to the 10th degree.  
In absolute terms B-270 will tilt the B axis clockwise to the 270th degree.

#### 11.1.4 G91 INCREMENTAL MODE

In the incremental mode (G91) the tilt value defines the direction and number of degrees for the tilt table to move. B+10. causes the table to tilt positively (CCW) 10° from its current position. B-10. causes the table to tilt negatively (CW) 10° from its current position.

#### 11.1.5 FEED RATE

Feed rate is addressed by use of the F word and G01 code. Tilt motion is programmed in degrees per minute.

Example:

G91G01B+45.F50.0 rotates the B axis 45° at 50° pr minute.

### 11.2 V300

#### 11.2.1 REFERENCE RETURN

Execute the following steps after powering ON machine.

1. A axis comes with incremental scale and we should obey the following procedure to reference the axis.
  - Press **JOG** button on the operator panel to change machine into Jog mode.
  - Press **4** button on the operator panel to select the A axis.
  - Press **+** button on the operator panel to move the A axis cross the reference mark and move to the positive side.
  - Press **-** button on the operator panel to move the A axis cross the reference mark and move to the negative side.
  - Press **REF RETURN** button on the operator panel to change machine into reference mode.
  - Press **4** button on the operator panel to select the A axis.
  - Press **+** button on the operator panel to move the A axis to the zero position automatically.

**11.2.2 A AXIS  
DIRECTION****A axis rotary table directions**

- (A+) = Counterclockwise work piece rotation (viewing in the X+ direction)
- (A-) = Clockwise work piece rotation (viewing in the X+ direction)

**A axis rotary table limit**

- A axis has the roll over function and has no travel limit to that axis

**11.2.3 G90 ABSOLUTE  
MODE**

In the absolute mode, the rotary angular value defines the final position between 0° and 360°. The + or - sign defines the direction, the table will move to get to the degree of rotation.

Example:

In absolute terms A+10. moves the A axis counterclockwise to the 10th degree.

In absolute terms A-270. moves the A axis clockwise to the 270th degree.

**11.2.4 G91  
INCREMENTAL  
MODE**

In the incremental mode (G91) the rotary value defines the direction and number of degrees for the rotary table to move. A+10. causes the table to move positively (CCW) 10° from its current position. A-10. causes the table to move negatively (CW) 10° from its current position.

---

---

## 12.0 APPENDIX

---

---

## 12.1 M CODE LIST

Table 12-1: M Codes

M CODES	DESCRIPTION	REMARK
M00	Program Stop	See unit II, Section 11.1 of 16i/18i Operator's Manual
M01	Optional Program Stop	
M02	End of Program	
M30	End of Program	
M98	Sub-Program Call	See unit II, Section 12.3 of 16i/18i Operator's Manual
M99	Sub-Program Returnl	
Machine Operation Called using following M-Codes defined in PMC Ladder interface Logic:		
M03	Spindle Start Forward	
M04	Spindle Start Reverse	
M05	Spindle Stop	
M06	Run Tool Change Cycle	Macro Call O9020. Pass Tool number in macro call Ex: M06T02
M07	Coolant ON-Mist/Thru Spindle	Mist Coolant (Coolant 2) Receptacle
M08	Coolant ON-Flood	Flood Coolant (Coolant 1) Receptacle (K15.0=1 reverse M07-Flood & M08-Mist)
M09	Coolant OFF	
M11	Set Tool Carousel position to 1	(ATC Tool Changer ONLY)
M19	Spindle Orient (for canned cycle use only)	Canned Cycles, spindle orients notrml to X/Y axes for motion off part surface during Boring Cycles
M20	Wash Down, Toggle ON/OFF	Set K15.7=1 when activating Wash Down by M20 otherwise use M68/69 codes listed below.
M22	Chip Conveyor Toggle On/OFF	Set K15.6=1 wjem activating Conveyor by M22 otherwise use M66/M67 codes listed below.
M29	Rigid Tap	M29 Sxxxx in block prior to G84
M41	Low Gear Select	Auto Select of gear range based on programmed Spindle speed will override selection
M42	High Gear Select	

Table 12-1: (Continued) M Codes

M CODES	DESCRIPTION	REMARK
M48	100% Spindle Speed Override Forced	
M49	100% Spindle Speed Override Reversed	
M60	A axis Brake ON	Output may be used as M60/M61 spare when not equipped with rotary axis
M61	A axis Brake OFF	
M62	B axis Brake ON	Output may be used as M62/M63 spare when not equipped with rotary axis
M63	B axis Brake OFF	
M64	M64/M65 Output ON	
M65	M64/M65 Output OFF	
M66	M66/M67 Output ON	M66/M67 used for Chip Conveyor when K15.6=0
M67	M66/M67 Output OFF	
M68	M68/M69 Output ON	M68/M69 used for Wash Down when K15.7=0
M69	M68/M69 Output OFF	
M78	M78/M79 Output ON	M78/M79 used by 2016 & 3016L Machines for Cooled Spindle Pump when K3.6=1
M79	M78/M79 Output OFF	
M85	A axis Rotary Table Enable	
M86	A axis Rotary Table Disable	
M87	B axis Rotary Table Enable	
M88	B axis Rotary Table Disable	
M100	Position Tool Drum to Pocket by T-word Ex: "M100T05" position to pocket #5 (Applies to both ATC tool drum and DATC tool carousel)	
ATC ONLY		
M150	Equivalent to Fadal "TC,1" command for return of tool to ATC carousel	
M151	Equivalent to Fadal "Manual, Start" sequence for transfer of tool from ATC carousel into spindle from M150 position.	
M190	Spindle orient for tool changer	This is a standard command that orients spindle to change position.

## 12.2 ALARM LIST

1. If it is an alarm related with CNC, please refer the CNC operator manual appendix.

For alarms from No.000 through No.253 and from No.5000 and over refer to Fanuc Operator's Manual GFZ-63534EN, Appendix, G.Alarm List).

**Table 12-2: Alarm list**

ALARM NO.	DESCRIPTION
1002	DUAL DOOR SWITCH FAILURE
1005	DOOR OPEN, MDI TOOL CHANGE ALARM
1040	CAN NOT CHANGE TOOL WHEN SPINDLE IS RUNNING
1041	CAN NOT CHANGE TOOL WHEN Z IS NOT IN HOME POSITION
1042	CAN NOT CHANGE TOOL WHEN SPINDLE IS NOT ORIENTED
1050	INIT SHIFT FAIL, CYC STRT INHIBT
1051	LOST LOW GEAR DURING OPERATION
1052	LOST HI GEAR DURING OPERATION
1054	HIGH TO LOW GEAR SHIFT FAILED
1055	LOW TO HIGH GEAR SHIFT FAILED
1056	SPINDLE WINDING SHIFT FAILED
1081	M11 ALM RETURN CLR SPDL TOOL
1083	TOOL UP AT MACHINE ON FAILED
1084	TOOL UP AT DRUN ROTATION FAILED
1086	STORED TOOL ILLEGAL, SEE OPR MSG
1090	NO EMPTY TOOL POCKETS AVAILABLE
1091	INVALID TOOL MGMT DATA FOR SEARCH
1092	OPTIONAL TOOL MGMT SEARCH ILLEGAL
1093	EMPTY POT SEARCH ILLEGAL DATA
1094	EMPTY POT SEARCH INVALID OPT
1095	TOOL MGMT SEARCH PROTECTED
1096	TOOL CHANGER INIT FAULT, SEE MSG
1097	NO TOOL CHGR, CYC STRT INHIBITED
1103	ATC CAN NOT EXTEND DURING SP-OP
1104	ATC TOOL COUNT SWITCH MISSING
1105	INHIBIT SPINDLE CMM, ATC IS NOT AT HOME POS.

Table 12-2: (Continued) Alarm List

ALARM NO.	DESCRIPTION
1106	ATC Z AXIS IS NOT AT REF. POSITION PLEASE REFERENCE Z AXIS
1107	ATC SPINDLE IS NOT ORIENTED. PLEASE ORIENT THE SPINDLE
1110	ATC CAROUSEL EXTEND FAILED
1111	ATC CAROUSEL RETRACT FAILED
1112	ATC CAROUSEL E-STOP ALARM
1113	ATC SLIDE SENSORS ARE BAD
1114	ATC SLIDE STOP IN WRONG PLACE AND TIME OUT
1115	ATC SLIDE EXTENDING IS TOO QUICK OR EXTEND SENSOR IS BAD
1116	ATC SLIDE RETRACTING IS TOO QUICK OR SLIDE HOME SENSOR IS BAD
1117	ATC SLIDE CAN NOT RETRACT BECAUSE DRAW BAR IS OPEN
1120	M85 WINDOW R/W DATA ERROR
1120	M86 WINDOW R/W DATA ERROR
1120	M87 WINDOW R/W DATA ERROR
1120	M88 WINDOW R/W DATA ERROR
1125	DRAW BAR TIME OUT CHECK DRAW BAR SENSOR
1126	ORIENTATION INHIBIT, SLIDE EXTENDED
1130	4TH AXIS BRAKE ON, COMMAND HALTED
1132	5TH AXIS BRAKE ON, COMMAND HALTED
1143	CYCLE CNC POWER
1145	DOOR PARAMETER WRITE ALARM
1152	ATC DISK IS IN WRONG POSITION
1153	CAN NOT EXTEND ATC WHEN ATC IS MOVING
1154	CAN NOT MOVE Z AXIS WHEN ATC IS RUNNING
1163	CYCLE START IS INHIBITED
1166	M11 RESET ACTIVE
1170	M84 LOGR CALIB DATA ZERO ERROR
1171	M84 LOGR SPDL SPEED CALC ERROR
1172	M84 LOW GEAR SHIFT ERROR

Table 12-2: (Continued) Alarm List

ALARM NO.	DESCRIPTION
1173	M84 LOGR WINDR READ SAMPLE ERROR
1174	M84 LOGR ARBITRARY DATA ERROR
1175	M84 LOGR REV COUNT WINDR ERROR
1176	M84 LOGR FDBK COUNT WINDR ERROR
1177	M84 LOGR DATA OUT OF RANGE ERROR
1180	M84 LOGR DISP RATIO MATH ERROR
1181	M84 LOGR RATIO RANGE ERROR
1182	M84 LOGR WINDW PAR SET ERROR 1
1183	M84 LOGR WINDW PAR SET ERROR 2
1184	M84 LOGR CALIB SPDL START ERROR
1186	M84 BELT CALIB CYCLE INHIBITED
1190	M84 HIGH CALIB DATA ZERO ERROR
1191	M84 HIGH SPINDLE CALC ERROR
1192	M84 HIGH GEAR SHIFT ERROR
1193	M84 HIGH WINDR READ SAMPLE ERROR
1194	M84 HIGH ARBITRARY DATA ERROR
1195	M84 HIGH REV COUNT WINDR ERROR
1196	M84 HIGH FDBK COUNT WINDR ERROR
1197	M84 HIGH DATA OUT OF RANGE ERROR
1200	M84 HIGH DISPLAY RATIO MATH ERROR
1201	M84 HIGH RATION RANGE ERROR
1202	M84 HIGH WINDR PAR SET ERROR 1
1203	M84 HIGH WINDR PAR SET ERROR 2
1204	M84 HIGH CALIB SPINDLE START ERROR
1205	NEED TO REFERENCE ALL AXES FIRST
1206	DOOR OPEN/CLOSE REQUIRED
1210	M84 #3741 CALC ERROR
1211	M84 #3742 CALC ERROR



Table 12-2: (Continued) Alarm List

ALARM NO.	DESCRIPTION
1212	M84 MAX LOW WINDW ERROR
1213	M84 #3751 MAX HIGH WINDOW ERROR
1214	M84 #3751 CALC ERROR
1215	M84 SHIFT SPINDLE WINDOW ERROR
1216	M84 #3736 CALC ERROR
1217	M84 #3736 WINDOW ERROR
1220	M84 #3752 WINDOW ERROR
1221	M84 #3762 WINDOW ERROR
1227	DATC/ATC NEEDS SYNCHRONIZATION
2004	OPERATOR FUNCTION INHIBITED CHECK DOOR OVRD, ALRM OPERATIONAL MODE
2005	MACHINE DOOR OPEN. FEED HOLD ON CYCLE START IS INHIBITED
2006	DOOR OVERRIDE IS ACTIVE
2007	CLOSE MACHINE DOOR TO CONTINUE OR RESET TO CANCEL OPERATION
2010	USE FUNC+SPINDLE CW OR CCW TO RESTART SPINDLE
2011	PRESS CYCLE START TO RESUME PROGRAM COMMAND
2012	TOOL CHANGE INTERRUPTED BY DOOR OPEN
2013	PRESS CYCLE START TO RESTART TOOL CHANGE
2015	DOOR UNLOCK INHIBITED
2016	DOOR IS CLOSED. PRESS ALARM PB TO RESET
2020	LOW LUBE OIL LEVEL REFILL VACTRA-2 OR EQUIVALENT
2024	LOW PROBE BATTERY
2026	AIR-OIL FAULT, ILLEGAL TABLE DATA FOR AIR-OIL INTERVALS
2027	AIR-OIL FAULT FEED HOLD SPINDLE STOP FORCED. SEE OTHER MESSAGES
2030	AIR-OIL LOW OIL PRESSUSRE FAULT
2031	AIR-OIL UPPER BEARING PRESSURE FAILED FAULT
2032	AIR-OIL LOWER BEARING PRESSURE FAILED FAULT
2033	AIR-OIL PRESSURE WHILE PUMP TURNED OFF
2034	AIR-OIL FAULT. SAFETY STOP MACHINE OR STOP WILL BE FORCED

Table 12-2: (Continued) Alarm List

ALARM NO.	DESCRIPTION
2035	AIR-OIL SPINDLE AIR/VAC PRESSURE FAILED FAULT
2036	AIR-OIL LEVEL LOW PLEASE REFILL RESERVOIR
2037	CORRECT FAULT & USE ALARM MSG PB TO RESET ALL AIR-OIL MESSAGES
2044	TOOL ARM OVERLOAD TRIP MAINTENANCE MUST RESET OVERLOAD
2045	TOOL MAGAZINE OVERLOAD MAINTENANCE MUST RESET OVERLOAD
2046	WASHDOWN PUMP OVERLOAD TRIP MAINTENANCE MUST RESET OVERLOAD
2055	M11 MSG ATC IS SET POCKET 1
2056	M11 ALM USING FUNC+LOAD TOOL BUTTON TO RETURN SPDL TOOL TO ATC FIRST
2057	PWR DOWN REQ'D TO CLR ALARM 1050. CHECK AIR PRESSURE 80-90PSI
2058	M11 ALM, TAKE SPDL TOOL OFF, CLR SPDL TOOL MANAGMENT TO 0
2059	M11 ALM, OPERATOR MUST EDIT TOOL MANAGEMENT ACCORDING TO REAL CAROUSEL
2060	Z AXIS TOOL CHANGE PROX SWITCH NOT CONFIRMING TOOL CHANGE POSITION
2061	TOOL DRUM POT NOT UP FOR ROTATION OF TOOL DRUM
2062	TOOL DRUM POT NOT IN DOWN POSN FOR ACCESS BY TOOL ARM
2063	TOOL ARM NOT IN POSITION FOR TOOL REMOVAL FROM SPINDLE
2064	DRAWBAR NOT RELEASED FOR REMOVAL OF TOOL FROM SPINDLE
2065	TOOL ARM NOT IN POSITION FOR TOOL INSERTION INTO SPINDLE
2066	DRAW BAR NOT CLAMPED ON NEW TOOL IN SPINDLE
2067	TOOL ARM NOT RETURNED TO PARK POSITION AT END OF TOOL CHANGE
2068	USING "FUNC+CYCLE START" BUTTON TO RESTART MAGAZINE
2070	TOOL CHANGE COMMAND ABORTED TOOL CHANGE SERVICE MODE ACTIVE
2071	TOOL CHANGE COMMAND ABORTED SPINDLE FRWD, REV, ORIENT ACTIVE
2072	ARM NOT AT HOME POSITION
2073	ARM AT SPINDLE
2074	Z MOVE TO THE PROHIBIT AREA
2075	TOOL CHANGE & SPINDLE, E-STOP FORCED ALARM PB TO CLEAR

Table 12-2: (Continued) Alarm List

ALARM NO.	DESCRIPTION
2076	TOOL CHANGE SERVICE MODE SET
2077	TOOL CHNG ALARM, ALARM PB TO CLR CYCLE START INHIBITED
2080	DATC IS NOT AT HOME POS CYCLE START INHIBITED
2081	CURRENT ATC CAROUSEL POSITION RESET TO POCKET #1
2082	TOOL DRUM AT POCKET # (CURRENT POCKET NUMBER) OF # (TOOL POCKET CAPACITY=16, 20, 21, 24, 32, OR 40). PRESS DRUM POSITION TO CLEAR MSG
2087	STORED TOOL VALUE OF # (CURRENT TOOL POCKET NUMBER) IS NOT IN LEGAL RANGE OF # (MINIMUM TOOL POCKET NUMBER = 1) TO (MAXIMUM TOOL POCKET NUMBER=16, 20, 21, 24, 32, OR 40)
2100	NO TOOL CHANGER SELECTED CORRECT K5.X & CYCLE POWER
2101	ILLEGAL VMC TOOL CHANGER SETTING CORRECT K5.X & CYCLE POWER
2102	VMC & HMC TOOL CHANGER BOTH SET CORRECT K5.X & CYCLE POWER
2126	DOUBLE CHECK ATC SYNCHRONIZATION
2127	ORIENTATION INHIBIT, SLIDE EXTENDED
2130	LOW COOLANT LEVEL OR PUMP FAILURE
2131	USE M61 TO RELEASE 4TH AXIS BRAKE PRIOR TO AXIS COMMAND BLOCK
2133	USE M62 TO RELEASE 5TH AXIS BRAKE PRIOR TO AXIS COMMAND BLOCK
2136	POSSIBLE FAILURE OF PCB-0321 OR SPINDLE BELT MAY BE BROKEN
2137	POSSIBLE FAILURE OF PCB-0321
2140	ADAPTIVE WARN LIMIT
2150	ATC IS NOT AT HOME POSITION USE M186 RETURN; Z INHIBITED
2161	TOOL CAGE DOOR IS OPEN, PLEASE CLOSE IT. FEED HOLD IS ON
2165	PRESS UNLOCK BUTTON FOR 60S TO UNLOCK DOOR
2187	M84 BELT CALIBRATION ACTIVE
2224	PB TWICE TO RESET, THEN USE JOG OR HANDLE MODE TO MOVE AXIS
2225	PRESS AND HOLD ENABLE KEY ON THE BACK OF HMOP AND PRESS ALARM
2667	WINDOW WRITE ERROR, CE

### 12.3 MACHINE COLD START PROCEDURE

Fadal Fanuc VMC comes with absolute encoder which is setup in the Fadal already. The machine will remember the position even if it is powered off. We need to make the cold start position again if the alarm No.300 appears on the screen which can be caused by low battery or disconnect the servo motor encoder.

Follow the next procedure:

1. Prepare: In MDI mode input G21 to change machine to Metric display.
  - 1.1 Press the MDI push button on the operator panel A.
  - 1.2 Press PROGRAM key one/two times on the MDI panel until PROGRAM (MDI) screen is displayed.
  - 1.3 Press G21 using MDI panel.
  - 1.4 Press EOB key on the MDI panel.
  - 1.5 Press INSERT key.
  - 1.6 Press CYCLE START push button on the operator panel A.
  - 1.7 Make sure the screen is changed to Metric Mode (X\*\*\*, \*\*\* is displayed).
  - 1.8 Press REF RETURN push button on the operator panel A.
  - 1.9 Press Z push button on the operator panel A (Manual Z axis zero return).
  - 1.10 Press X push button on the operator panel A (Manual X axis zero return).
  - 1.11 Press Y push button on the operator panel A (Manual Y axis zero return).
  - 1.12 Preparation is done.
2. Reference all the axes.
  - 2.1 Press REF RETURN and select reference mode, then push X/Y/Z key to move X/Y/Z axis to zero position. CNC screen will display X0Y0Z0.
3. Install the tool.
  - 3.1 Press JOG push button on the operator panel A.
  - 3.2 Press TOOL REL push button on the operator panel and keep it. (Draw Bar will be released)
  - 3.3 Put the tool inside of spindle nose.
  - 3.4 Release the TOOL REL push button.
4. Find out X axis zero position manually, make the cold start.
  - 4.1 Press HANDLE and select MPG mode, press X1/X10/X100 to select MPG resolution, and press X select to select X axis.
  - 4.2 Use TAPE MEASURE find out the X axis middle point of the table, and make a mark.
  - 4.3 Use MPG move the X axis to the X axis middle position mark.
  - 4.4 Record the X coordinate displayed on the CNC screen as A.
  - 4.5 Press MDI push button on the operator panel A.
  - 4.6 Press SYSTEM key on the MDI panel.

- 4.7 Press PARAMETER soft key below the LCD screen.
  - 4.8 Press 1850 using MDI panel.
  - 4.9 Press NO.SRH soft key below the LCD screen (Parameter 1850 will be displayed).
  - 4.10 Move the cursor to X.
  - 4.11 Press the X coordinator as following format: 34560 (3.456x10000).
  - 4.12 Press INPUT key on the MDI panel (34560 will be displayed on the parameter 1850X column)
    - \*\*\*Alarm 000 will pop up on the alarm window.
  - 4.13 Power OFF the CNC (Press "O" button on the operator panel B).
  - 4.14 Power ON the CNC (Press "I" button on the operator panel B).
    - \*\*\*Alarm 300X will be displayed on the screen (it is normal).
  - 4.15 Press JOG push button on the operator panel A.
  - 4.16 Press X push button on the operator panel A.
  - 4.17 Press "-" minus push button to move the X axis 55.0mm or more (press "+" push button if machine is 2016, 3016, EMCD, 2216).
  - 4.18 Press "+" plus push button to move the X axis middle position (press "-" push button if machine is 2016, 3016, EMCD, 2216).
  - 4.19 Press REF RETURN push button.
  - 4.20 Press X push button. (X axis will automatically move to the zero position)
  - 4.21 In case if zero position is wrong, repeat the previous steps starting with step 4.5
  - 4.22 Double check the cold start position using tape measure.
  - 4.23 Press RESET key on the MDI panel. That will reset the 300 alarm.
5. Find out Y axis zero position manually, make the cold start.
    - 5.1 Install the indicator on the spindle.
    - 5.2 Repeat the same procedure as 4.1-4.23 to find out the Y axis cold start position.
  6. Find out Z axis zero position manually, make the cold start.
    - 6.1 Install tool on the arm (DATC) or Magazine (ATC).
    - 6.2 Measure the height of the tool.
    - 6.3 Install the tool on the spindle.
    - 6.4 Use MPG move the Z axis to the same height as the tool on the ARM/Magazine.
    - 6.5 Repeat the same procedure as 4.5-1.23 to find out the Z axis cold start position.
  7. Repeat the same procedure for A/B axis.
  8. Cold Start setup is finished.

## 12.4 FANUC FILES BACKUP PROCEDURE

Please prepare the compact flash card.

### I. Step

(Prepare operation)

1. Smoothly put the **COMPACT FLASH CARD** inside the M-CARD slot of LCD.
2. Press "**EDIT**" button to change to edit mode.
3. Press "**SYSTEM**" key on the MDI panel until parameter screen comes out.
4. Press "+" twice using the soft key until ALL IO displays below the LCD.
5. Press "**ALL IO**".

(Backup the parameter)

6. Press "<" to return to the previous page.
7. Press "**PARAM**" soft key on the LCD screen.
8. Press "**OPRT**" soft key.
9. Press "**PUNCH**" soft key.
10. Press "**EXEC**" soft key (OUTPUT will be display on the screen, all the parameter will output to the Flash Card as **CNCPARAM.DAT**).

(Backup the MACRO)

11. Press "<" to return to the previous page. Backup of data files procedure:
12. Press "+" soft key to change to the next page.
13. Press "**MACRO**" soft key.
14. Press "**OPRT**" soft key.
15. Press "**PUNCH**" soft key.
16. Press "**EXEC**" soft key (OUTPUT will be display on the screen, all the macro variables will output to the Flash Card as **MACROVAR.DAT**).

(Backup the PITCH)

17. Press "<" to return to the previous page.
18. Press "**PITCH**" soft key on the LCD screen.
19. Press "**OPRT**" soft key.

20. Press "PUNCH" soft key.
  21. Press "EXEC" soft key. (OUTPUT will be display on the screen, all the pitch compensation will output to the Flash Card as **PITCHERR.DAT**).
- (Backup the work offset)
22. Press "<" to return to the previous page.
  23. Press "WORK" soft key.
  24. Press "OPRT" soft key.
  25. Press "PUNCH" soft key.
  26. Press "EXEC" soft key. (OUTPUT will be display on the screen, all the parameter will output to the Flash Card as **WORK-G54.DAT**).
- (Backup the Magazine)
27. Press "<" to return to the previous page.
  28. Press "+" to the next page.
  29. Press "MAG".
  30. Press "OPRT".
  31. Press "PUNCH".
  32. Press "EXEC" (OUTPUT will be display on the screen, all the parameter will output to the Flash Card as **MAGAZINE.DAT**).
- (Backup the Tool management).
33. Press "<" to return to the previous page.
  34. Press "TOOL-MNG" soft key.
  35. Press "OPRT" soft key.
  36. Press "PUNCH" soft key.
  37. Press "EXEC" soft key. (OUTPUT will be display on the screen, all the parameter will output to the Flash Card as **TOOL-MNG.DAT**).

## II Step

(Backup the PLC software)

1. Put the Flash Card into the slot on the LCD.

2. Press "EDIT" button to change to change edit mode.
3. Press "SYSTEM" on the MDI panel until parameter screen comes out.
4. Press "PMC" soft key.
5. Press "I/O".
6. Select the page as following (move the cursor, and use the soft key to select the correct data type).

**M-card  
Write  
Ladder  
PMC-SB.000**

7. Press "EXEC" soft key. (OUTPUT will be display on the screen, PLC software will output to the Flash Card as PMC-SB.000).

(Backup the PLC parameter)

8. Move the cursor to the "DATA TYPE=".
9. Press the "PARAM" soft key on the LCD.
10. Select the page as following.

**M-card  
Write  
PARAM  
PMC-SB.PRM**

11. Press "EXEC" (OUTPUT will be display on the screen, PLC software will output to the Flash Card as PMC-SB.PRM).

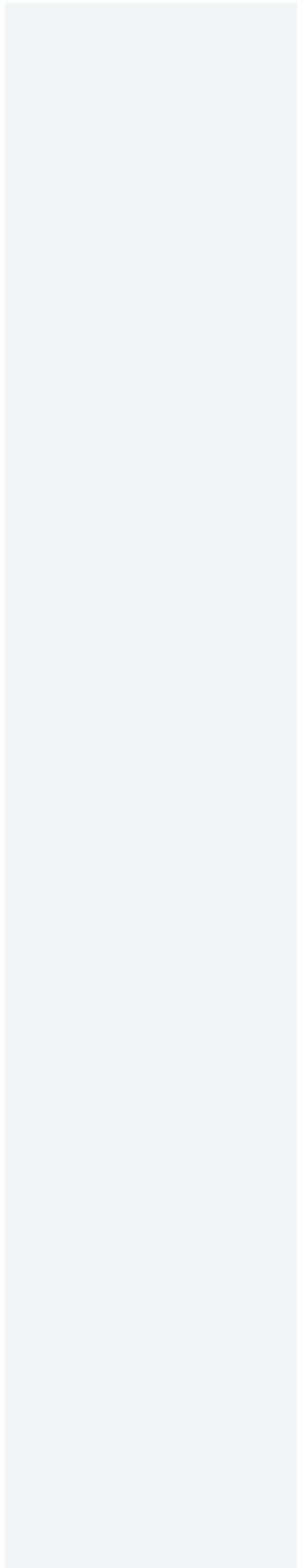
### III Step: SRAM backup procedure:(*Series 18i models B*)

1. Power OFF the machine.
2. Put the flash card into the slot.
3. Press the most right side two soft key on the LCD screen and hold.
4. Press the CNC power on push button.
5. Wait until the CNC system screen comes out.
6. Press the "DOWN" soft key to move the cursor to the item 5 (is highlighted):

***"5-System Data Backup"***



7. Press "SELECT" soft key will display two choices:  
*"1. SRAM BACKUP(CNC — Flash Card)"*  
*"2....."*
8. Make sure the cursor is on the first line.
9. Press "SELECT" soft key and you will be asked "Backup SRAM Data OK?"
10. Select "YES" to continue. (or "NO" to return to previous menu)
11. Wait couple of seconds. The screen will display complete information.
12. Press "SELECT" soft key.
13. Press the "DOWN" soft key until the cursor moves to item : END.
14. Press "SELECT".
15. Press the "DOWN" soft key until the cursor moves to item : END.
16. Press "SELECT" soft key.
17. Press "YES" soft key.
18. Finish, the CNC system will boot up.



---

---

## INDEX

---

---

### A

A AXIS DIRECTION 60  
AIR PRESSURE 2  
ALARM LIST 64  
ALARM MSG 9  
APPENDIX 61  
AUTO 9  
AUTO COOL 46  
AUTO OPERATION 36  
AUTOMATIC OPERATION 17  
AUTOMATIC OPERATION MODES 18

### B

B AXIS TILT TABLE 58  
BLOCK DELETE 24

### C

CLR MAG 38  
COOL POWER RESERVOIR 4  
COOLANT CONTROL PUSH BUTTONS 46  
COOLANT OPERATION 45

### D

DATC 32-TOOL WITH TOOL CAGE DOOR (6535 40T/50t) 42  
DNC OPERATION 20  
DOOR OVRD 8  
DRUM FWD 32  
DRUM PSN 32  
DRUM REV 32  
DRY RUN 24

### E

EDIT 9  
EDIT THE TOOL/MAGAZINE LIST 39  
EMERGENCY STOP SWITCH 8

### F

FANUC FILES BACKUP PROCEDURE 72  
FEED RATE 59  
FLOOD COOL 46

FLOOD COOLANT 3  
FUNCTION 28

## G

G90 ABSOLUTE MODE 58  
G90 ABSOLUTE MODE 60  
G91 INCREMENTAL MODE 59  
G91 INCREMENTAL MODE 60

## H

HANDLE OPERATION 15  
HANDLE 10  
HMOP OPERATION GUIDE 51  
HMOP OPERATION ON/OFF SWITCH 52  
HOME PSN 28

## I

INC JOG OPERATION 14  
INC JOG 10  
INITIALIZING ATC TOOL MAGAZINE 40

## J

JOG 10  
JOG FEED RATE 16  
JOG OPERATION 13

## K

KEY SHEET LAYOUT 52

## L

LOAD PSN 28

## M

M CODE / S CODE 49  
M CODE LIST 62  
MACHINE COLD START PROCEDURE 70  
MACHINE SETUP FUNCTION 27  
MAIN OPERATOR PANEL 8  
MANUAL HANDLE FEED PROCEDURE 53  
MANUAL JOG FEED PROCEDURE 54  
MANUAL OPERATION 11  
MANUAL OPERATION 32  
MANUAL OPERATION MODES 12  
MANUAL REFERENCE POSITION RETURN 12  
MANUAL REFERENCE RETURN 54

MANUAL REFERENCE RETURN 58  
MANUAL SPINDLE CONTROL 55  
MANUAL SPINDLE SPEED 49  
MC LOCK 24  
MDI 9  
MDI OPERATION 18  
MEMORY OPERATION (AUTO MODE) 19  
MEMORY OPERATION PROCEDURE 53  
MEMORY PROTECTION KEY 21  
MESSAGE KEY 56  
MIST COOL 46

## O

OIL RESERVOIR 2  
OPERATION MODE SELECTION 9  
OPERATOR PANEL LAYOUT 7  
OPTION STOP 24  
OVERRIDE SWITCH 52

## P

POWER OFF 5  
POWER ON 4  
POWER ON/OFF 1  
POWER ON/OFF 4  
PRE-START CHECKING STEPS 2  
PROGRAM STOP 24

## R

REF RETURN 8  
REFERENCE RETURN 59  
REFERENCE THE DRUM (ONLY FOR DATC) 36  
REMOTE 9  
ROTARY TABLE 57

## S

SAFETY FUNCTIONS 8  
SETUP MODES 28  
SINGLE BLOCK 24  
SPDL CCW 48  
SPDL CW 48  
SPDL STOP 48  
SPINDLE OVERRIDE 50  
SPINDLE OPERATION 47  
SPINDLE OPERATION PUSH BUTTONS 48  
SPINDLE ORIENTATION 50

SPINDLE SPEED RANGE 49

## T

T-COMMAND (TOOL NUMBER) 36  
TEST FUNCTION 23  
TEST FUNCTION MODES 24  
TOOL BRKN 38  
TOOL CHANGER (RECOVERY ATC) 42  
TOOL CHANGER COMMAND 36  
TOOL CHNG (DATC ONLY) 34  
TOOL DATA 38  
TOOL LOAD 34  
TOOL LOADING (ATC) 41  
TOOL MANAGEMENT 38  
TOOL MEAS 28  
TOOL OPERATION 31  
TOOL ORDER PROCEDURE (ONLY FOR ATC) (M11) 39  
TOOL REL 32  
TOOL REL 56  
TR 165 58

## V

V300 59

## W

WATER RESERVOIR 3  
WORK LIGHT 9  
WORK MEAS 28

## Z

Z HOME 29