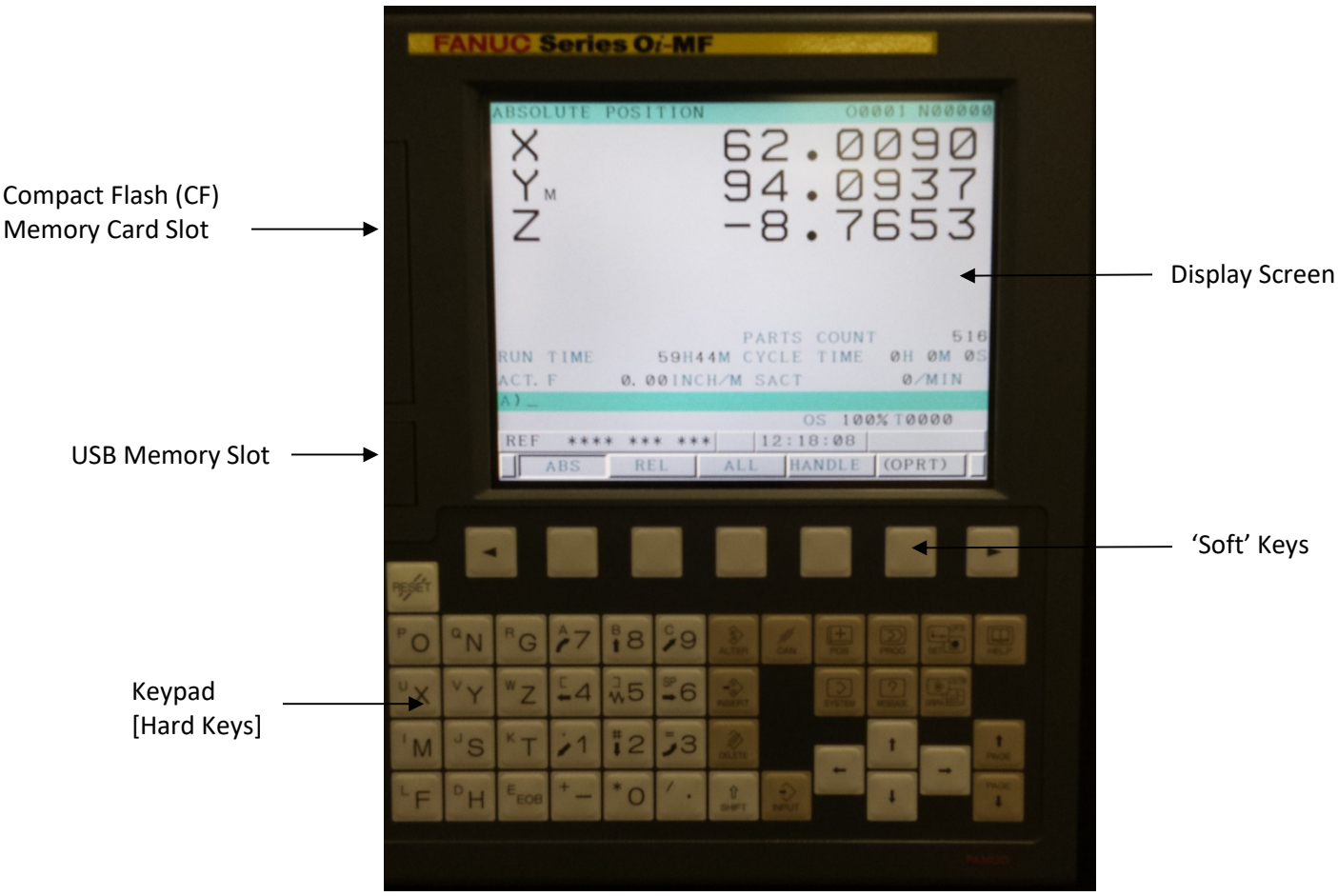
Control Cabinet Parts

CNC Controller Module Layout



Display Screen – Main screen for viewing machine operation and status.

USB Memory Slot – USB interface for transferring programs and system backup.

Compact Flash (CF) Memory Slot – Memory interface for system backup and memory expansion.

Keypad [Hard Keys] – CNC Controller functions keys with printed labels.

Keypad (Soft Keys) – CNC Controller function keys without printed labels.

Note: 'Soft' Keys are keys whose functions change depending on the operation being performed. They do not have a printed label however the text shown directly above the key is the key's currently assigned function. Throughout this manual, a soft key will be indicated using round parenthesis () around the text display above the corresponding key; For example, as displayed above (ABS)(REL)(ALL)… A Hard Key has a printed label and its function does not change. A hard key will be indicated using square brackets [ ] around the key's printed label; For example, as displayed above [POS][PROG]…

Operator Keyboard Module Layout

Emergency Stop Button – Pressing the Emergency Stop Button stops all machine motion.

Spindle Speed Override Knob– Turning the spindle speed override knob will change the spindle rotation 0% to 120% of programmed rotation rate.

Motion Speed Override Knob – Turning the motion speed override knob will change each axes travel 0% to 120% of programmed travel speed.

Function Keys – Operator Keyboard Function Keys.

Light Tower

A light tower is located at the top of the control cabinet. Its purpose is to provide color-coded machine status. More than one light may be lit at one time.

Red Light – When lit an alarm condition exists. View alarm information on the CNC Controller screen

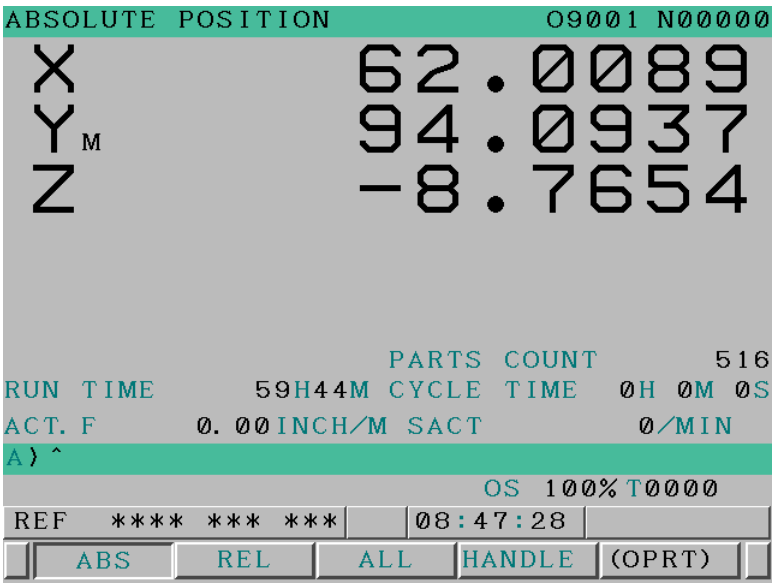
Yellow Light – When lit the CNC Controller is executing program code.

Green Light – When lit the CNC Controller is powered and functional.

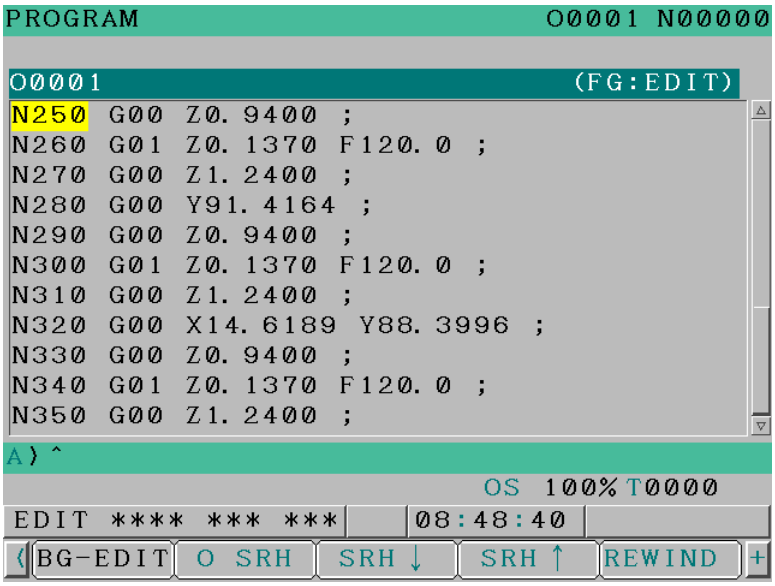
Common CNC Operator Screens

Boot Screen – The window below is displayed after CNC Controller boots.

Position Screen. – This window is used to display the current position of each axis, feedrate, and cycle time. It is displayed by pressing the [POS] key on the CNC Controller. Pressing the [POS] key multiple times will toggle through Absolute position, Relative position, and ALL positions screen. Pressing (ABS), (REL), or (ALL) soft keys will display, Absolute position, Relative position, or ALL position screens, respectively.



Program Edit Screen – The window shown below displays the contents of the currently loaded program. This window can be used to display and edit programs as well as display program lines as they are being run.



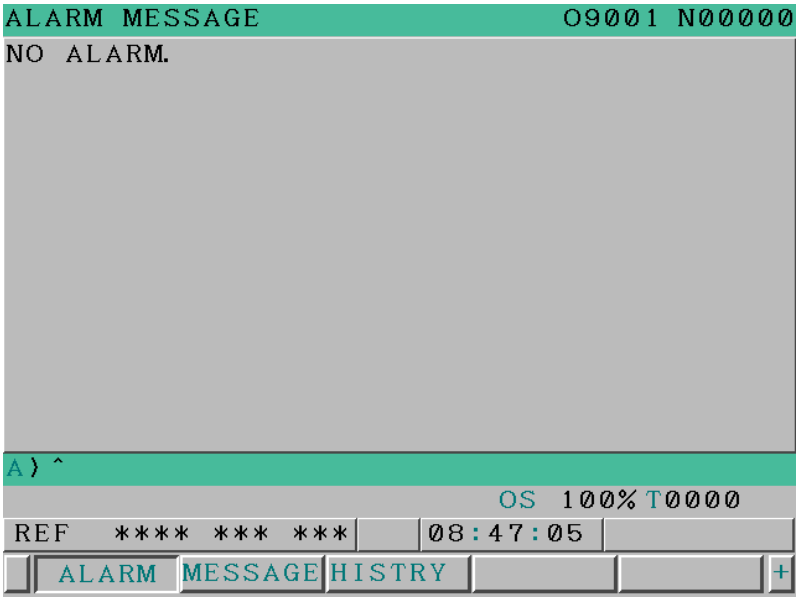
Operation Basics

Turning On the Machine

1. Make sure there are no objects or people near the machine that can be damaged or injured by machine movement.
2. Make sure the Emergency Stop button is released by rotating it clockwise.

3. Rotate the main contact switch clockwise until it clicks in place at the ON position.

4. As CNC Controller powers up, the screen will display a series of status messages. When the system has completed the startup process the screen below will be displayed. The light tower will have the green light illuminated.



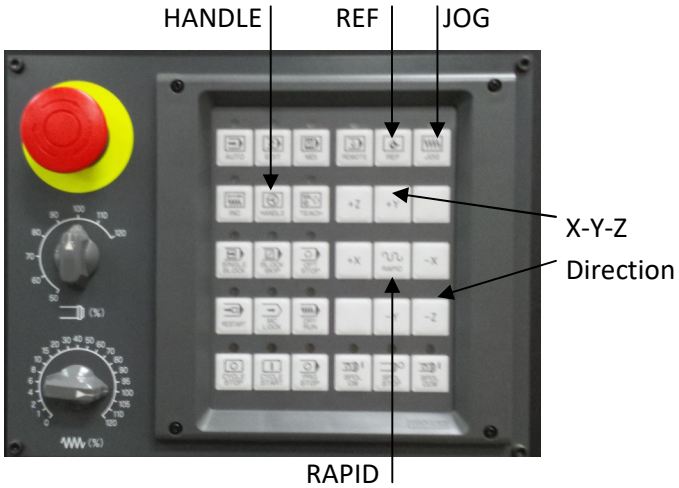
SYSTEM RUNNING

Turning Off the Machine

Turn off the main power by rotating the main contact switch counter-clockwise to the OFF position.

Rotate knob to OFF position. →

Machine Movement Operations



Jog Axes

Press [JOG] to enter jog mode. Press [-X], [+X], [-Y],[+Y],[-Z] or [+Z] to jog axis in desired direction.

Press and hold [RAPID] to jog axis at higher rate.

If an axis over-travel message is displayed, jog the machine away from its positive or negative limit, then press RESET to clear the message.

Homing

Position each axis about 12 inches from its respective home position. For X and Y that is in the negative direction, Z is in the positive direction.

Press [REF], then press [-X]. When the X-switch is tripped, the X-motion stops.
Press [REF], then press [-Y]. When the Y- switch is tripped, the Y-motion stops.
Press [REF], then press [+Z]. When the Z- switch is tripped, the Z-motion stops.
If equipped;
Press [REF], then press [B+]. When the B-switch is tripped, the B-motion stops.
Press [REF], then press [C-]. When the C-switch is tripped, the C-motion stops.
Note: In [REF] mode, axes will only travel toward their home position.

Jog Axes using the Manual Pulse Generator (MPG)

Press [HANDLE] button to enter MPG Mode.

Select axis (X – Y – Z) to be moved using the Axis Selector Knob
If equipped, additional axis selections are available.

Select motion multiplier,
- X1 for slow (precision) movement speed
- X10 for faster movement speed
- X100 for fastest movement speed

Rotate knob clockwise to move axis in positive direction
Rotate knob counter-clockwise to move axis in negative direction.

If an axis over-travel message is displayed, jog the machine away from its positive or negative limit, then press [RESET] to clear the message.

Jogging the Tool Changer Magazine

The tool changer magazine can be jogged manually by pressing the [unlabeled] key to the right of [+Y] and above [-X]. Spindle must be in the down position.

Manual Tool Release

The tool holder can be released from the spindle manually by pressing the manual release button. The machine must be put into JOG state by pressing the [JOG] button on the Operator Keyboard as shown previously.

The manual tool release button is located on the back of rotary 8-position tool changer. Jog the spindle near the rotary tool changer so it is within easy reach.

Note. When the manual release button is pressed the clamping method is released and the tool will fall out. Place your hand so that the tool holder [not the cutter] is supported.

Setting Work Coordinates and Tool Offsets

Automatic Tool Touch-Off
Z-Offsets for Tools 1 – 8

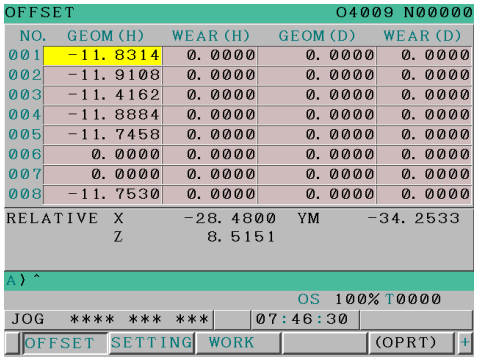
Z-offsets for Tools 1 – 8 are determined automatically by the tool touch-off procedure, M-Code M401.

Press [PROG][MDI] TxM401 [EOB][INSERT][CYCLE START], where x is tool number 1 – 8.

Note: This will create machine motion.

The CNC Controller will perform the touch-off procedure and automatically update the tool offset table for the corresponding tool. To view the offset table for Tools 1 – 8, press [SET/OFF](OFFSET)

Z-Offsets for Tools 1 - 8

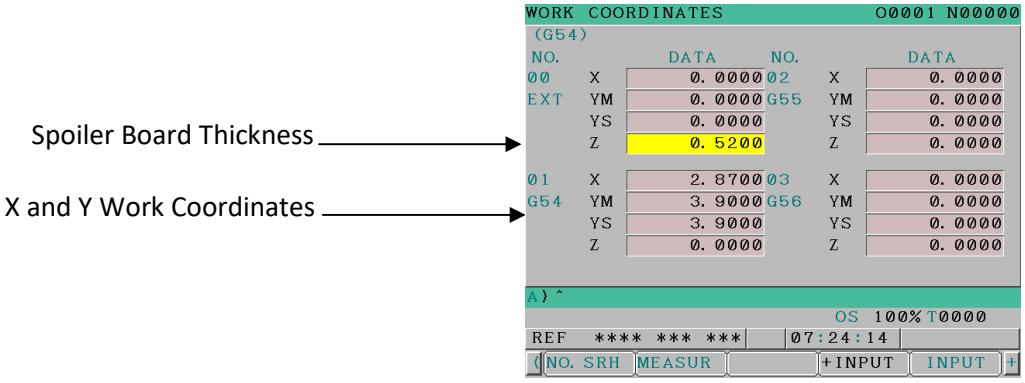


Setting Spoiler Board Thickness

Measure the thickness of the spoiler board. Enter the thickness in the [SET/OFF](WORK) screen at NO. 00 EXT block Z-Coordinate (example 0.6710).

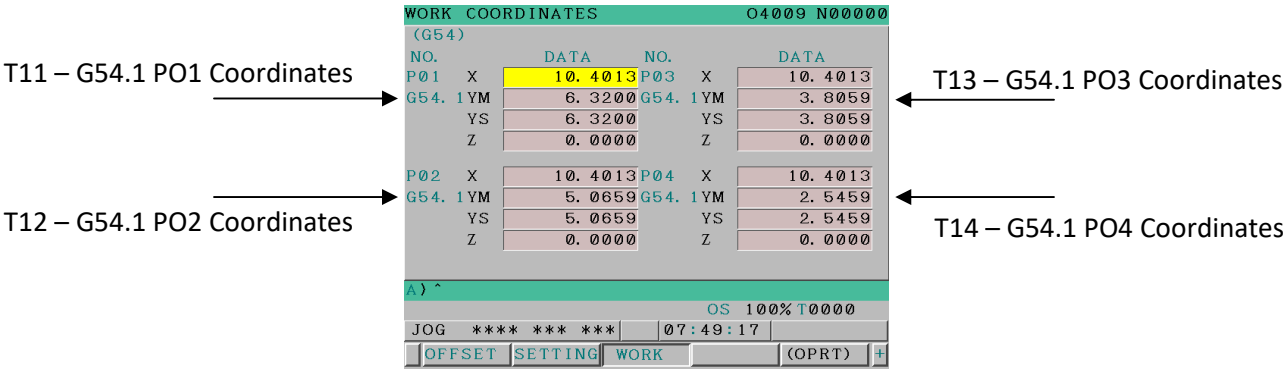
Setting Work Coordinates

JOG the spindle to the desired X – Y location (front left corner of work surface). Note the Machine coordinates from the Position Screen [POS] (example X=2.8700; Y=3.9000). Enter the X and Y values in the [SET/OFF](WORK) screen G54 No. 01 set of coordinates.



Setting Drill Block Offsets for X and Y Axes

1. Extend front most drill (T11;)
2. Jog machine so the drill is located at the front-left most corner of the work surface.
3. Lower Z until the drill just makes contact with the work surface.
4. Note machine values for X, Y, and Z axes. This is the drill block work space origin.
5. Enter X and Y coordinate values for T11 into G54.1 P01 area.
   To access the Offset Value screen enter:
   [SET/OFS](WORK)[PG DN]…[PG DN] to G54.1 P01 section for the first drill.
6. Enter the X and Y values for the drill.
7. REPEAT for each drill in the drill block.
   Hint: Drills are positioned 32mm (1.26”) apart. Add or subtract (1.26”) to previous drill’s values to set the work space offsets for each drill.



Press [PG DN] to access additional G54.1 Px coordinate groups.

Sample work coordinate values for a 5 x 5 drill block with the following orientation:

T19 T18 T17 T16 T15

T14

T13

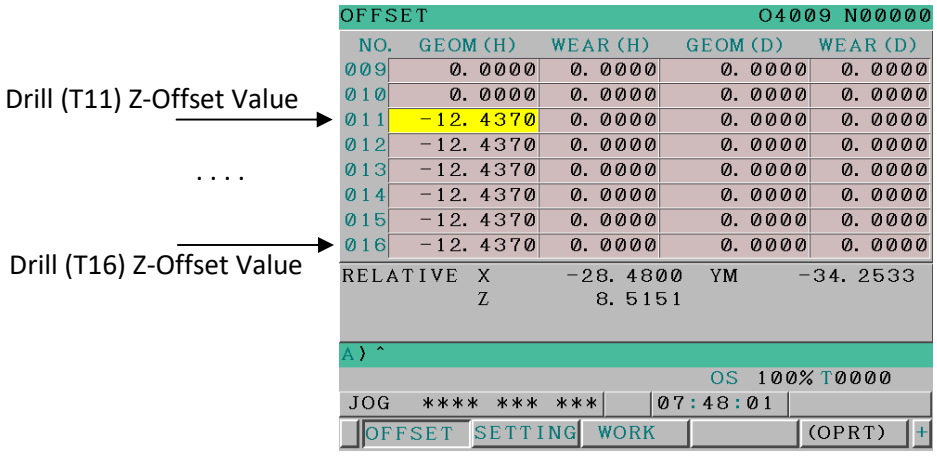
T12

T11

Drill	Group	X	Y	Z
T11	G54.1 P01	10.4013	6.3259	0.0000
T12	G54.1 P02	10.4013	5.0659	0.0000
T13	G54.1 P03	10.4013	3.8059	0.0000
T14	G54.1 P04	10.4013	2.5459	0.0000
T15	G54.1 P05	10.4013	1.2859	0.0000
T16	G54.1 P06	11.6613	1.2859	0.0000
T17	G54.1 P07	12.9213	1.2859	0.0000
T18	G54.1 P08	14.1830	1.2859	0.0000
T19	G54.1 P09	15.4413	1.2859	0.0000

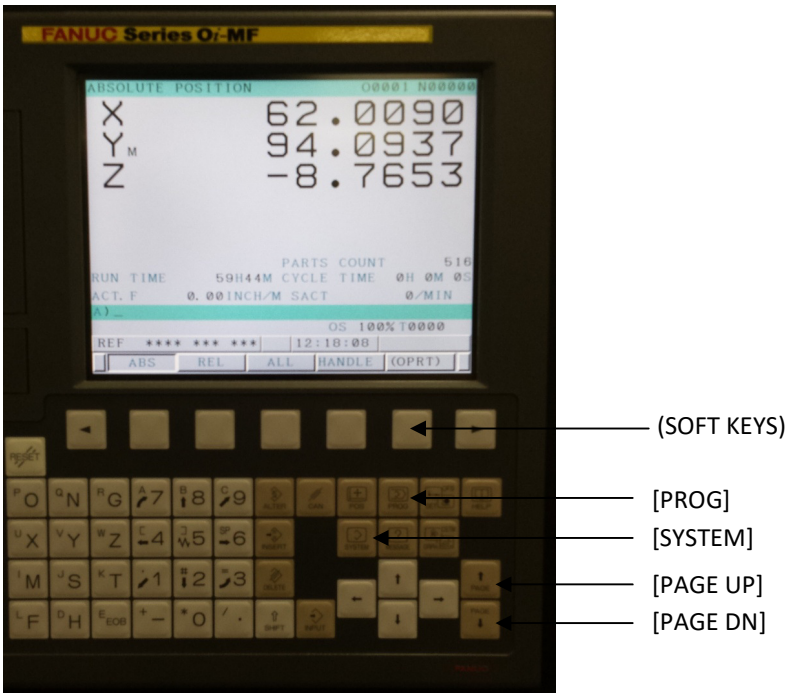
Setting Drill Block Z-Axis Offset

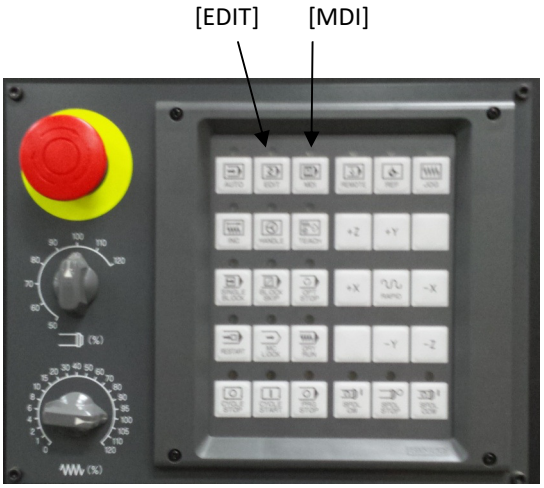
1. To enter drill Z axis offset, press [SET/OFS](OFFSET)[PG DN]..[PG DN] until 011 row is displayed.
2. Enter the offset value (example, -12.437) for Tool 11 in the GEOM (H) field, then press [INPUT].
3. Repeat for remaining drills present in the drill block. Note: Z value will not change unless a drill has a different bit length.



PROGRAM OPERATIONS

For Program Operations the [HARD KEYS] shown below will be used.





Viewing and Navigating Directories

View CNC Controller Program Directory

[PROG][EDIT](FOLDER)

(Press [Up Arrow] or [Down Arrow] buttons to scroll through list of programs)

View Directory Of Memory Devices (USB Memory OR CF Memory Card)

[PROG][EDIT](FOLDER)

(Press [Up Arrow] or [Down Arrow] buttons to scroll through list of programs)

To view the directory of a different device press [PROG][EDIT](OPRT)(DEVICE CHANGE). Select the device.

Directory Structure – The CNC has a directory structure as described below. To navigate to other directories, highlight the directory name then press [INPUT]. The directory name and path will be displayed at the top of the display. The contents of the selected directory will be displayed in the main window. Select "RETURN TO UPPER FOLDER" [INPUT] to return to previous folder.

//CNCMEM/MTB1	Location for CNC Macros
//CNCMEM/MTB2	Empty
//CNCMEM/SYSTEM	Empty
//CNCMEM/USER/LC1	Empty

//CNCMEM/USER/LC2	Empty
//CNCMEM/USER/LIBRARY	Empty
//CNCMEM/USER/PATH1	Program File Storage
//CNCMEM/USER/PATH2	Empty

Loading Programs

Load Program From CNC Controller Memory

[PROG][EDIT](highlight the program number)(OPRT)(MAIN). The selected program number will be preceded with the @ symbol. For example, @ 04001. The program number will also be displayed at the top of the CNC display.

Copying Programs

Copy Program from USB or CF Memory Card to CNC Controller Memory

(SET I/O CHANNEL = 17 for USB; I/O CHANNEL = 4 for MEMORY CARD)

[PROG][EDIT](OPRT)(DEVICE CHANGE)(USBMEM) or (MEMORY CARD).

(Press [Arrow Up] or [Arrow Down] buttons to scroll through list of programs)

Highlight the program name then press (COPY). Note: Press the (Up Arrow) or (Down Arrow) to see the selected program name is highlighted.

Press (DEVICE CHANGE)(CNC –MEM)(PASTE). The program file will be copied to the CNC Memory.

Copy Program From CNC Memory to USB or CF Memory Card

(SET I/O CHANNEL = 17 for USB; I/O CHANNEL = 4 for MEMORY CARD)

[PROG][EDIT](OPRT)(DEVICE CHANGE)(CNC-MEM).

(Press [Arrow Up] or [Arrow Down] buttons to scroll through list of programs)

Highlight the program name then press (COPY). Note: Press the (Up Arrow) or (Down Arrow) to see the selected program name is highlighted.

Press (DEVICE CHANGE)(USB –MEM) or (MEMORY CARD), then (PASTE). The program file will be copied to the selected device.

Deleting Programs

Delete Program From CNC Controller Memory

[PROG][EDIT](OPRT)(DEVICE CHANGE)(CNCMEM).

Press [Arrow Up] or [Arrow Down] buttons to scroll through list of programs)

Highlight the program name then press (DELETE). Press (EXEC) to delete or (CAN) to cancel operation.

Delete Program from USB Memory or CF Memory Card

[PROG][EDIT](OPRT)(DEVICE CHANGE)(USBMEM) or (MEMORY CARD)

Press [Arrow Up] or [Arrow Down] buttons to scroll through list of programs.

Highlight the program name then press (DELETE). Press (EXEC) to delete or (CAN) to cancel operation.

Editing Programs

Edit Program on CNC Controller

[PROG][EDIT](FOLDER)

Press [Arrow Up] or [Arrow Down] buttons to scroll through list of programs.

Highlight the program name then press (MAIN PROGRAM). Press (PROG). The program is displayed.

MODIFY existing field:

Highlight field to be edited. Enter new field data, then press [ALTER] to replace field data.

DELETE existing field:

Highlight field to be deleted, then press [Delete] to delete field.

INSERT new program line:

Highlight field ahead of the new field/line to insert

Enter new field [INSERT] or

Enter new program line [EOB][INSERT]

Note: Every command line must end with an 'end of block' [EOB] which is displayed as a semi-colon (;).

CHANGES ARE INSTANTLY SAVED.

Renaming Programs

Rename a Program on CNC Controller

[PROG][EDIT](FOLDER)

Press [Arrow Up] or [Arrow Down] buttons to scroll through list of programs.

Highlight the program name then press (MAIN PROGRAM). Press (PROG). The program is displayed.

Highlight first line of program that displays program name.

Enter new program name (e.g. O1098),then press [ALTER] to replace field data

Program O1099 has now been renamed to O1098.

Program O1099 no longer exists.

Creating New Programs

Create a Program on CNC Controller

[PROG][EDIT] Enter Filename (e.g. O1099), then press (NEWPRO).

Program O1099 now exists.

Insert program lines as desired.

Loading and Running Programs on CNC Controller

Loading a Program

[PROG][EDIT](FOLDER)(OPRT) Highlight the program name then press (MAIN PROGRAM).

The program name is now preceded by a @ and its name is displayed at the top of the CNC display.

Press (PROG). The program's contents are now displayed.

Starting a Program in AUTO Mode (after program is loaded into CNC memory)

[PROG][AUTO][CYCLE START]

Running a Program In Single-Block Mode

[PROG][AUTO][SINGLE BLOCK] … [CYCLE START] to advance program to next line.

Program will run one line at a time. Press [CYCLE START] to progress through program manually line-by-line.

Stopping Programs

Stopping a Running Program

[CYCLE STOP] to halt all motion except rotation of spindle or drill block.

Press [RESET] to stop spindle or drill block rotation and to move to beginning of program.

Resume a Stopped Program

To restart a stopped program press [CYCLE START]. The program will continue.

Note: If [RESET] has been pressed then the program will restart from the beginning.

MDI Mode

MDI Mode allows the operator to control the machine by entering specific M-Codes and G-Codes.

Entering Commands in MDI Mode

For a single line

[PROG][MDI] enter command.

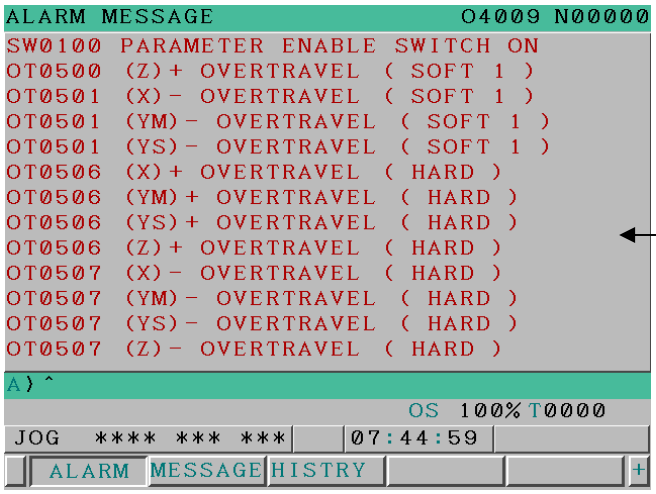
Example, T8M06[EOB] [INSERT]…[CYCLE START] to execute command.

For multiple command lines separate commands with [EOB] key.

Example, T8M06[EOB]M03S12000[EOB][INSERT]…[CYCLE START] to execute command.

Alarms

Alarm conditions are displayed on the CNC Controller. When an alarm occurs, the red light on the light tower will be lit. Below is a list of some common alarms and instructions to clear them.



Alarm Display on CNC Controller

Alarm Present When Lit

OT0500 (X,Y,Z,B,C) + OVERTRAVEL (SOFT)
OT0501 (X,Y,Z,B,C) - OVERTRAVEL (SOFT)

Clearing Soft Over-Travel Alarms

A soft over-travel alarm occurs when an axis has traveled beyond a coordinate limit preset in the CNC Controller. This will occur if the axis has traveled too far in the (+) positive or (-) negative direction. To clear the alarm, jog the axis using [JOG] mode or [HANDLE] mode and the MPG unit. Once the axis has been jogged within the soft limit coordinates, press the [RESET] button to clear the alarm.

Clearing Soft Over-Travel Alarms During Homing

If a soft over-travel alarm occurs during homing, follow the procedure below.

Set Parameter 1815 to 0000 0000 (APC, APZ = 0) for all axis.
Cycle power
Home all Axes
Set Parameter 1815.5(APC) = 1
Home all Axes, then cycle power.

OT0506 (X,Y,Z,B,C) + OVERTRAVEL (HARD)
OT0507 (X,Y,Z,B,C) - OVERTRAVEL (HARD)

Note: Any axis that creates a hard over-travel alarm will cause hard over-travel alarms for all axes.

Clearing Hard Over-Travel Alarms (Hard Over-Travel Switch Override)

If an axis trips an over-travel switch a HARD OVER-TRAVEL alarm occurs and the axis movement will be disabled. The axis must be moved off of the switch before the alarm can be cleared. To override the switch, press and hold the [unlabeled] button under the [+X] button and to the left of the [-Y] button. While pressing this button also press the [RESET] button to clear the alarm. While continuing to press the [unlabeled] button, jog the over-traveled axis off of the limit switch. Jogging the axis can be done using [JOG] mode button or [HANDLE] mode using the handheld manual pulse generator.

SW0100 – Parameter Enable Switch On

Clearing Parameter Enable Switch Alarm

A Parameter Enable Switch Alarm occurs at boot time if the ability to write/modify parameters on the CNC Controller is enabled. This alarm is cleared by pressing [RESET]. To prevent the alarm from occurring, Parameter Write ability can be disabled by accessing the SETTINGS screen by pressing the [SET/OFF] key, then (SETTING) soft key. Highlight the Parameter Write field and Enter 0, then [INSERT]. The Parameter Write ability is now disabled.

DS0001 SYNC EXCESS ERROR (POS DEV)

Clearing Sync Excess Errors: Refer to Parameter 8323.

DS0300 (X,Y,Z,B,C) APC ALARM: NEED REF RETURN

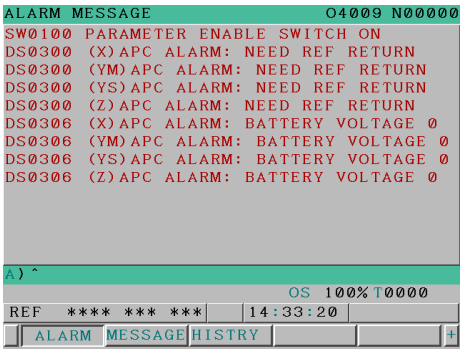
Clearing Ref Return Alarms

A Reference Return Alarm indicates an axis or axes need to be homed. To home the specific axis, press [REF] key then –X, -Y, or +Z depending on which axis or axes have alarmed. Once the axis has reached the home switch the alarm can be cleared by pressing [RESET].

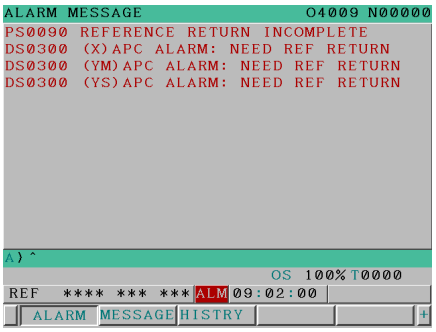
DS0306 (X,Y,Z) APC ALARM: BATTERY VOLTAGE 0
DS0307 (X,Y,Z) APC ALARM: BATTERY LOW 1
DS0308 (X,Y,Z) APC ALARM: BATTERY LOW 2

Clearing Low Battery Voltage Alarms

A four D-cell battery pack provides backup supply voltage to the servo motors. This supply voltage allows each servo motor to retain its absolute position coordinates while the system is not powered. If the CNC Controller detects low battery, a Battery Voltage Alarm will occur. To clear the alarm, change-out the batteries. Replace the batteries while the machine is powered. After the batteries have been changed, cycle the CNC Controller power to clear the alarm(s).



PS0090 REFERENCE RETURN INCOMPLETE



Clearing Reference Return Incomplete Alarms

A Return Incomplete Alarm occurs if the axis has traveled too far (or not far enough) before reaching its home switch. To clear the alarm press [RESET]. Repeat the Reference Return process for the axis. Ideally, the axis should travel about 8" towards the home switch.

PS0224 ZERO RETURN NOT FINISHED

Clearing Zero Return Not Finished Alarms

An axis whose absolute encoder position is not saved while the system is not powered requires a Reference Return before commands can be issued. Reference the axis then press RESET.

EX1006 SPINDLE OVER TEMP

Clearing Spindle Over-Temperature Alarms (DI X0025.0)

The spindle over temperature switch has been activated. Inspect the spindle cooling equipment. After spindle cools sufficiently the switch will deactivate. Note: Not all systems or spindles support this alarm.

SV0001 SYNC ALIGN ERROR

Clearing Sync Excess Errors: Refer to Parameter 8325.

SV0005 SYNC EXCESS ERROR (MCN)

Clearing Sync Excess Errors: Refer to Parameter 8314.

SV0440 PS EXCESS-REGENERATION2

Regen resistor overheat or open thermocouple wiring.

Memory Card Program – PC Tool Operation

Creating and Modifying FANUCPRG.BIN

By using the Memory Card Program (FANUCPRG.EXE), you can create and modify (FANUCPRG.BIN), a memory card program file which functions from the CF Memory card slot of the CNC Controller and provides extended memory function. The maximum size of the memory card program file is 2048 Mbytes (2 Gbytes). Instructions for the Memory Card Program are provided in a separate FANUC document, named 05_J_pc_tool_for_memory_card.pdf. After FANUCPRG.BIN has been created and copied to a CF memory card, the memory card is inserted into the CNC Controller and accessed using the following procedures.

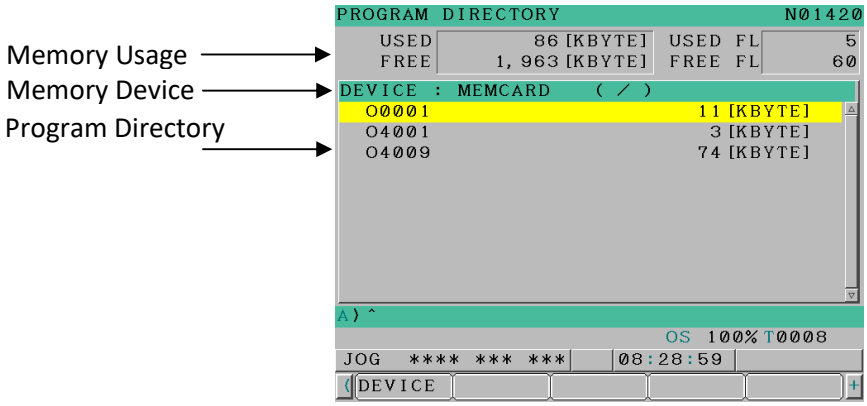
NOTE: Folders/Program Numbers selection must remain at 63.

Mounting the CF Memory Card

Press [PROG](FOLDER)(OPRT)(DEVICE)(MEMCRD)

The CF Memory Card will now be the source and destination of program functions.

You will notice the directory of the programs in FANUCPRG.BIN, those selected using FANUCPRG.EXE.

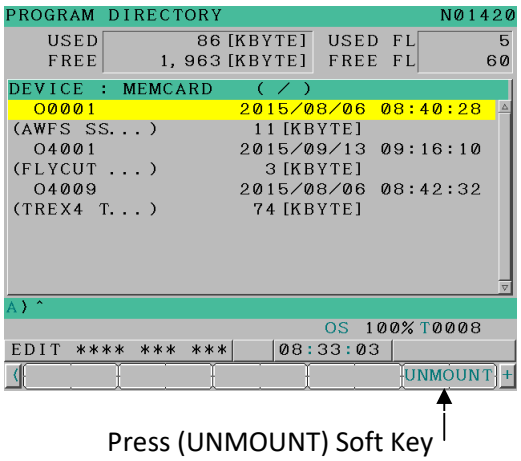


Unmounting the CF Memory Card

To use the CNC Controller built-in memory the CF Memory Card must be unmounted.

Press [PROG](FOLDER)(OPRT)(DEVICE)(UNMOUN)

The CNC Controller's built-in memory will now be the source and destination of program functions.



Advanced Configuration Information

Information for System Administrators

Setting System Time and Date

Press [MDI][SET/OFS][SETTING] , then [Page DN] until you see Date and Time fields. Highlight field and enter new data followed by [INPUT].

Screen Capture of the CNC Controller

Press and hold the [SHIFT] key until the system clock stops. Release the button and the screen capture (BMP) will be written to the set memory device (I/O Channel).

Connecting the CNC Controller to the Network

Setting Up the Ethernet Port

Note: The CNC is setup as an FTP Client. Therefore, FTP Server software needs to installed on the host PC. All file copy operations are initiated from the CNC (FTP client). File copy operations cannot be initiated from the PC (FTP server).

Press [MDI] [SET/OFS](SETTING) highlight I/O CHANNEL field, enter:

9 for Embedded Ethernet (transfer between PC and CNC Controller)

Press [INPUT]

[EDIT][SYSTEM](EMBED)(COMMON)

Highlight IP ADDRESS field, Enter IP address of CNC, (e.g. 192.168.1.2) then press [INPUT]

Highlight SUBNET MASK field, Enter subnet mask, (e.g. 255.255.255.0) then press [INPUT]

Highlight ROUTER IP ADDRESS field, Enter router IP address (if any), then press [INPUT]

Highlight DNS IP ADDRESS 1 field, Enter DNS IP address (if any), then press [INPUT]

Highlight HOST NAME field, Enter host name of CNC, then press [INPUT]

When done press (OPRT])(RESTART)(EXECUTE).

Setting Up FTP Transport **(for connection to PC)**

[EDIT][SYSTEM](EMBED)(COMMON)(FTPTRNS)

Highlight HOST NAME (IP ADDRESS) field, Enter host IP address, (e.g. 192.168.1.10), then press [INPUT]

Highlight PORT NUMBER field, Enter port number (21), then press [INPUT]

Highlight USER NAME field, Enter user name, (FANUC), then press [INPUT]

Highlight PASSWORD field, Enter password, (FANUC), then press [INPUT]

[PAGE DN] Highlight LOGIN FOLDER field, Enter folder name (CNC), then press [INPUT]

Note: Verify CNC folder is under the root directory on the target PC's (C:\CNC).

Repeat steps if additional PC connections are needed. Three PC connections are possible.

[EDIT][SYSTEM](EMBED)(FOCAS2)

Highlight PORT NUMBER (TCP), Enter 8193, then press [INPUT]

Highlight PORT NUMBER (UDP), Enter 8192, then press [INPUT]

Press (OPRT)(RESTART)(EXECUTE) to load new changes.

Testing the Network Connection (PING)

[EDIT][SYSTEM](EMBED)(PING)(OPRT)(P. FTP1) to ping first PC setup.

The PC will reply if a successful connection was made.

You may also ping the CNC from the PC. It should reply.

Note: The firewall on the PC may need to be configured or turned off.

Connecting to the PC

[EDIT][SYSTEM](EMBED)(FTPTRANS)(OPRT)(HOST)(CONECT1) to connect to the PC.

Press [EDIT][PROG](DIR) to display the CNC folder on the PC.

The PC's disk contents will be visible if a successful connection was made.

Copying Files Using FTP Protocol (via ethernet)

To copy a program to CNC memory press (OPRT)(F INPUT). Enter filename as shown on PC then press (F NAME). Enter program number to write to the CNC then press (O SET). Press (EXEC) to complete the action. Note: Program file size cannot be larger than available CNC memory. If an error occurs during the copy process, delete the incomplete program file from CNC memory, free up additional CNC memory, then retry.

To copy a program from CNC memory to the PC press (OPRT)(F OUTPUT). Enter the program number to copy to the PC then press (O SET). Enter the filename to write to the PC then press (FNAME). Press (EXEC) to complete the action.

System Backup and Restore

Full System Backup and Restore Using USB Memory Device

The CNC system (Configuration, Ladder, Programs, Parameters, SRAM) can be backed up to a USB memory device or Memory Card.

(To backup to USB drive – I/O Channel = 17)
(To backup to Memory Card – I/O Channel = 4)
Files will be written to the root folder of the target device overwriting any existing files named the same.

Perform Backup: [SYSTEM][EDIT](ALL IO)(ALL DT)(OPRT)(FOUTPUT)

> MESSAGE: THIS OPERATION NEEDS POWER OFF. SAVE FILE IS OUTPUT TO ROOT FOLDER. SAME NAME FILE IS OVERWRITTEN. EXECUTE?
>
> DEVICE NAME: will show the target device (USB MEM or MEM CARD).

(EXEC) to perform the backup. After the files have been written you will be prompted to cycle the power. When the system reboots, other SRAM memory contents will be saved to the output device.

Restoring Parameters from Backup Files

(For backup files located on USB drive – I/O Channel = 17)
(For backup files located on CF memory card – I/O Channel = 4)

[SYSTEM][EDIT](PARAM)(OPRT)(F INPUT)(EXEC) The CNC System knows to load the set of parameter files found in root directory of the set device. Message to cycle power will appear. Cycle power.

SRAM Backup (CF Memory Card Only)

A FULL SYSTEM BACKUP CAN BE PERFORMED BEFORE THE CNC CONTROLLER BOOTS BY PRESSING THE TWO TOP-RIGHT MOST BUTTONS ON THE CNC KEYBOARD WHILE THE POWER IS SWITCHED ON. Entire SRAM contents will be written to a single file and stored on the Memory Card.

At the SYSTEM MONITOR MAIN MENU highlight 7. SYSTEM DATA UTILITY (SELECT)

Highlight 1. SRAM BACKUP (CNC → MEMORY CARD) (SELECT)

ARE YOU SURE? Press (YES). Contents of CNC SRAM will be written to the memory card.

Highlight 1. END (SELECT) to exit backup utility and resume boot operation.

SRAM Restore (CF Memory Card Only)

A FULL SYSTEM BACKUP CAN BE RESTORED TO THE CNC BY PRESSING THE TWO TOP-RIGHT MOST BUTTONS ON THE CNC KEYBOARD WHILE THE POWER IS SWITCHED ON. Entire SRAM contents will be restored from a single file written during a previous backup operation.

At the SYSTEM MONITOR MAIN MENU highlight 7. SYSTEM DATA UTILITY (SELECT)

Highlight 2. SRAM RESTORE (MEMORY CARD → CNC) (SELECT)

ARE YOU SURE? Press (YES). Contents of previous backup will be restored to CNC SRAM.

Highlight 1. END (SELECT) to exit backup utility and resume boot operation.

Note: The SRAM_BAK.001 file needs to be on the CF Memory card. This file can be from a previous backup to the CF card or copied from an (ALL DATA) backup located on USB.

Setting and Understanding System Parameters and Variables

Macro Variable List

There are several macro programs that run to perform certain operations; automatic tool change, tool touch off. As these programs run they use variables (macro variables) to track machine status, direction, locations… These variables are stored in specific memory locations and are described below.

Access and Modify Macro Variables

To access memory variable locations: Press [SET/OFS] (MACRO]). Press [Page Up],[Page Dn] until desired macro variable number is displayed or enter the Variable Number, then (NO. SRH) to search to the location.

To change a memory location value, highlight the location to be changed, enter the new value (123.456), then press [INPUT]. Verify the correct location was changed. Verify the correct value is displayed.

Note: Use Variable number as a reference, since variable name is optional and not always used.

(SETVN 503 [Z-MAX];

Variable No.	Name	Data (MM)	Description
500	X_TCHGPN	64.8800	X-coordinate for tool change
501	Z_TCHGPD	-6.1733	Z-coordinate for tool change (down – to grip)
502	Z_TCHGPU	-2.7000	Z-coordinate for tool change (up – safe height)
503	(Z-MAX)	-12.0000	Maximum Z travel for TTO
504	TTO-DIFF	-1.2128	(Z-Table - TTO) Difference between top of table and touch-off switch. This is the z-distance added to the tool length when the touch-off program is executed. See Touch-Off Procedures for further explanation.
505	X-TTO	10.5000	X-coordinate for the touch-off switch.
506	Y-TTO	102.9100	Y-coordinate for the touch-off switch.
507	Z-TTO	-8.0000	Z-coordinate to start touch-off sequence (safe height).

Automatic Tool Changer – Memory and Counter Locations

Instructions to Initialize/Verify Program State and Tool Changer Position

The automatic tool changer (ATC) routine performed by the Ladder uses memory locations and Counter (C0000) to track its progress, current tool in spindle, current tool changer position. The memory locations used and their function is described below.

Note: Only D0402 and C0000-CURRENT should be manually adjusted.
Other memory and counter addresses shown are for reference only.

To access the memory locations press [SYSTEM](PMCMNT)(DATA). Highlight D0400 then press (OPRT)(ZOOM).

Memory Location	Data	Description
D0400	8	Temporary Location – Index to tool location
D0401	8	Maximum tool #
D0402	3	Tool Number in spindle***
D0403	0	Always 0
D0404	0	Always 0
D0405	8	Tool Command – new tool number

The above memory location (D0402) shows that Tool 3 is in the spindle. CONFIRM THAT IT IS.

If the tool number in D0402 is not correct, highlight the field and enter the number of the tool that is in the spindle, then press [INPUT]. If the spindle is empty, enter 0, then press [INPUT].

The Counter C0002 can be viewed by pressing (COUNTR) button. C0000 for this example is 3

C0000-PRESET	8	PRESET = Max tool positions
C0000-CURRENT	3	CURRENT = position of tool changer***

If C0000-CURRENT does not show the current position of the tool changer, enter the current tool changer position, then press [INPUT].

Tool Touch-Off – Setting Coordinates and Determining Z-Offset

1. Place a bit into the spindle using the ATC (T1M06, Tool 1 for example).
2. Jog the spindle until it is directly over the top-center of the touch-off switch. Note the MACHINE X and Y coordinates on the [POS]screen and enter them into Custom Macro Variable Locations, 505 and 506, respectively.
   (See instructions how to access and modify MACRO VARIABLE values.)
3. Set MACRO VARIABLE #504 (Z-offset) to 0.0000.
4. Execute T1M401, tool touch-off routine for Tool 1.
5. When complete, note value placed into Offset Table for Tool 1 [SET/OFS] H1 GEOM location. Example: -10.0849; This is the Z-coordinate for the top of the TTO switch

6. Using the [JOG] or [HANDLE], place tool over the machine table. Slowly lower the bit towards the table until it just touches the surface.
7. Note the MACHINE Z coordinate on the [POS] Screen.
   Example: -11.2977; This is the Z-Table coordinate for the machine table.
8. Subtract Z-TTO coordinate from Z-table coordinate
   Example: (-11.2977)-(-10.0849) = -1.2128
9. Enter this value into MACRO VARIABLE LOCATION #504. This is the Z-offset value
10. Perform new Touch-Offs for each tool present. (TxM401; where x is tool number 1 – 8)

To access memory variable locations: Press [SET/OFS] (MACRO). Press [Page Up][Page Dn] until desired macro variable number is displayed or enter the Variable Number, then [NO. SRH] to search to the location.

To change a memory location value, highlight the location to be changed, enter the new value (123.456), then press [INPUT]. Verify the correct location was changed. Verify the correct value is displayed.

All About Macros

Macros are programs used the CNC system to perform specific operations and are named using the CNC programming notation; O9001. Programs O8000 and O9000 series are reserved for macro usage and are not readily accessible by the operator by setting Parameter #3202. See information in Parameter section for further information.

O8012 – Used to conveniently set variable names used in macros – helps troubleshooting.
O8113 – Program used in tool touch-off macro. This program is called by O9002.
O9001 – Program used for tool change macro (M06).
O9002 – Program used for tool touch-off macro (M401). This program calls O8113.
O9003 – Attach 4th Axis Hardware
O9004 – Detach 4th Axis Hardware
O9020 – Program used to set motor tuning parameters (M25).

Below lists the contents of each program but should not be considered the latest revision.

Contents of O8012 – Set Variable Names Macro

```
O8012(SET VARIABLE NAMES)
SETVN500[X_TCHGPN]
SETVN501[Z_TCHGPD]
SETVN502[Z_TCHGPU]
SETVN503[Z-MAX]
SETVN504[TTO-DIFF]
SETVN505[X-TTO]
SETVN506[Y-TTO]
SETVN507[Z-TTO]
M30
```

Contents of O8113 – Probe Length Offset Macro

```
O8113(PROBE LENGTH OFFSET)
G90
IF[#20EQ#0]GOTO99(ALARM ISSUED IF OFFSET IS NOT ASSIGNED)
G40G80G49(STARTUP BLOCK)
G10L52(TURN OFF SAR CHECK)
N3708R00000000
G11
IF[#26NE#0]GOTO98
#26=0
N98#3004=0(DIABLE FEEDRATE OVERRIDE=2, =0 ENABLE)
#11=#4001(STORE CURRENT G-CODE OF GROUP 01)
#13=#4003(STORE CURRENT G-CODE OF GROUP 03)
#16=#4006(STORE CURRENT G-CODE OF GROUP 06)
IF[#16EQ20.0]GOTO20(CHECK IF MAIN PROGRAM IS IN INCHES)
```

```
IF[#16EQ21.0]GOTO21(CHECK IF MAIN PROGRAM IS IN METRIC)

N20#32=#503(SET EXTRA TRAVEL IN INCHES)
#9=2.0(SET PROBING FEEDRATE IPM)
GOTO100

N21#32=[#503*25.4](SET EXTRA TRAVEL IN MM)
#9=50.0(SET PROBING FEEDRATE MM/MINUTE)

N100(PROBING STARTS HERE)
#33=#26+#32(CALCULATE THE FINAL Z PSN = -6.0MM)
IF[#1006NE1]GOTO110
G53Z#507
GOTO115
N110G53Z[#507*25.4]
N115G90G31Z#33F[#9*4](MAKE THE INITIAL PROBE TOUCH AT A FASTER
FEEDRATE)
IF[#1007NE1]GOTO99

IF[#1006NE1]GOTO120
G91G0Z0.5(INCH)
GOTO125
N120G91G0Z12.7(METRIC)
N125
(RETRACT TWICE THE AMOUNT OF STORED EXTRA TRAVEL = 6.0MM)
G90G31Z#33F[#9*2](MAKE THE FINAL PROBE TOUCH AT AT SLOWER FEEDRATE)

IF[#1006NE1]GOTO130
G91G0Z0.25(INCH)
GOTO135
N130G91G0Z12.7(METRIC)
N135G90G31Z#33F#9
IF[#1006NE1]GOTO140
#100=[#5024+#504](REGISTER Z PSN AT SKIP SIGNAL)
GOTO145
N140#100=[#5064+#504*25.4](REGISTER Z PSN AT SKIP SIGNAL)
N145

#[2200+#20]=#100(TRANSFER THE NEW VALUE TO THE SELECTED OFFSET)



GOTO1000(BYPASS ALARM MSG IF PROCESSING NORMAL)

N99#3000=99(TOOL TOUCHOFF NOT COMPLETED)

N1000G91G00G28Z0(RETURN Z TO MACHINE ZERO)

#3004=0(ENABLE FEEDRATE OVERRIDE)
G#11G#13G#16(RESTORE PREVIOUS G-CODES OF GROUPS 01,03,06)
G10L52
N3708R00000001
```

```
G11
M99
```

Contents of O9001 – Tool Change Macro

```
O9001(TOOL CHANGE MACRO)
(VERIFY MACRO VARIABLE WHEN SWITCH FROM INCH TO METRIC-SHIFT DECIMAL
PLACE ONLY)
G90
M16(START ATC STEP1)
(IF TCMD=TOOL# IN SPDL, NO TOOL CHANGE)
IF[#1000EQ1]GOTO200(TC=TSP, NO TOOL CHANGE)
M05(STOP SPINDLE)
M48(DUSTHOOD UP)
IF[#1001EQ1]GOTO100(NO TOOL IN SPDL)
M19(INDEX MAGAZINE TO T# IN SPDL)

IF[#1001EQ1]GOTO100(NO TOOL IN SPDL)
IF[#1006NE1]GOTO10(IF METRIC GOTO N10)
G53Z#501(TOOL CHANGE PSN-INCH)
GOTO102
N10G53Z[#501*25.4](TOOL CHANGE PSN-METRIC)
GOTO102

N100IF[#1006NE1]GOTO101
G53Z#502(CLEAR FOR MAGAZINE TO INDEX-INCH)
GOTO102
N101G53Z[#502*25.4](CLEAR FOR MAGAZINE TO INDEX-METRIC)

N102G10L52
N1420P4R1000000
G11

IF[#1006NE1]GOTO106
G53X#500(MOVE Z TO TOOL CHG PSN-INCH)
GOTO107
N106G53X[#500*25.4](MOVE X TO TOOL CHG PSN-METIC)
N107M27(EXTEND TOOL MAG TO SPINDLE)
G04X2
M21(TOOL RELEASE)

WHILE[#1003NE1]DO1(CHECK TOOL RELEASE BEFORE Z PULL UP)
G04X1
END1
G04X1.
IF[#1006NE1]GOTO120
G53Z#502(PULL Z UP-INCH)
GOTO121
N120G53Z[#502*25.4](PULL Z UP-METRIC)
N121G04X1.
IF[#1002EQ1]GOTO150(TCMD=0, PUT TOOL AWAY ONLY)
```

```
M17(INDEX MAG TO TCMD-NEW TOOL#)
(CHECK TCMD=NEW POCKET#)
WHILE[#1005NE1]DO2
G04X1
END2
(MOVE Z DOWN)
IF[#1006NE1]GOTO130
G10L52
N1420P4R600000
G11

G53Z#501(MOVE Z DOWN-INCH)
GOTO150
N130G53Z[#501*25.4](MOVE Z DOWN-METRIC)

N150G04X1
(LOCK THE TOOL)
M20(TOOL GRIPPED)
M18(UPDATE T# IN SPDL D402)
G04X2
(STEP8 RETRACT MAG)
M28(RETRACT MAGAZINE)
G04X1
#199=#4113
IF[#1100EQ1]GOTO190
M49(DUSTHOOD DOWN)
N190G10L52
N1420P4R5000000
G11

N200M99
```

Contents of O9002 – Tool Touch-Off Macro

```
O9002(TOOL TOUCH OFF)
N20G17G40G80G90(STARTUP BLOCK)
#1100=1
G65P9001
M48
#1100=0
IF[#1006NE1]GOTO30(METRIC,JUMP TO N30)
G90G00G53Z#507(Z IN INCH)
G90G00G53X#505Y#506T#4120(XY PSN FOR PROBING-INCH)
GOTO50
N30G90G00G53Z[#507*25.4](Z IN METRIC)
G90G00G53X[#505*25.4]Y[#506*25.4]T#4120(XY PSN FOR PROBING-METRIC)
N50G65P8113Z0.0T#4120(CALL MACRO FOR Z0 AND TOOL LENGTH OFFSET)
N60M99
```

Contents of O9003 – Attach 4th Axis Hardware

```
%
O9003(M35 Attach 4th Axis)
G10L52
N3115P5Q0R0
N1023P5R5
N1815P5Q5R1
G11
M99
%
```

Contents of O9004 – Detach 4th Axis Hardware

```
%
O9004(M36 Detach 4th Axis)
G10L52
N3115P5Q0R1
N1023P5R-128
N1815P5Q5R0
G11
M99
%
```

Contents of O9020 – AIAPC Parameter Setting Macro

```
O9020(AIAPC PARAMETER SETTING)
#3003=3(DISABLE SINGLE BLOCK)
#1=#1006
IF[#18EQ#0]GOTO80
IF[#18EQ1]GOTO50
IF[#18EQ2]GOTO60
IF[#18EQ3]GOTO70
GOTO80
N50
(R1 – SPEED)
G21
G10L52
N1769P1R128
N1769P2R128
N1769P3R128
N1769P4R128
N1660P1R2000000
N1660P2R2000000
N1660P3R2000000
N1660P4R2000000
N1735P1R1500000
N1735P2R1500000
N1735P3R1500000
N1735P4R1500000
N1737P1R1500000
N1737P2R1500000
N1737P3R1500000
N1737P4R1500000
N1783P1R500000
N1783P2R500000
N1783P3R500000
N1783P4R500000
G11
GOTO90
N60
(R2 – MID POINT)
G21
G10L52
N1769P1R128
N1769P2R128
N1769P3R128
N1769P4R128
N1660P1R1700000
N1660P2R1700000
N1660P3R1700000
N1660P4R1700000
N1735P1R1000000
N1735P2R1000000
N1735P3R1000000
N1735P4R1000000
```

```
N1737P1R1000000
N1737P2R1000000
N1737P3R1000000
N1737P4R1000000
N1783P1R475000
N1783P2R475000
N1783P3R475000
N1783P4R475000
G11
GOTO90
N70
(R3 - PRECISION)
G21
G10L52
N1769P1R64
N1769P2R64
N1769P3R64
N1769P4R64
N1660P1R700000
N1660P2R700000
N1660P3R700000
N1660P4R700000
N1735P1R500000
N1735P2R500000
N1735P3R500000
N1735P4R500000
N1737P1R500000
N1737P2R500000
N1737P3R500000
N1737P4R500000
N1783P1R400000
N1783P2R400000
N1783P3R400000
N1783P4R400000
G11
GOTO90
N80
#3000=1(R VALUE WAS NOT 1 THRU 3)
N90
IF[#1EQ1]GOTO92
G21
GOTO94
N92
G20
N94
#3003=0(ENABLE SINGLE BLOCK)
M99
```

About 9020 Macro Variable Values.

During motor tuning, Fanuc Servo Motor Tuning Application Software determined the following range of parameter values. The values represent a system tuned for speed(R1.), speed-precision(R2.), precision(R3.).

The values below are for X,Y,Y1,Z axes. B & C values for two axis spindle should not be changed.

Parameter	Speed(R1.)	Speed-Precision(R2.)	Precision(R3.) or	Ultra (R3.)
1769	128	128	64	64
1660	2000	1700	700	200
1783	500	475	400	100
1735	1500	1000	500	150
1737	1500	1000	500	150

Note: R1 values work well for cabinet nests or programs primarily made up of straight cuts.
R2 produced a nice performance balance between cabinet nests and fine geometry parts.
R3 is for high precision cutting – takes more time to complete program.
It is recommended to stay within the boundary limits of R1 and R3.

Digital Inputs and Outputs

View the State of a Digital Input.

[SYSTEM](PMCMNT)(STATUS)

Scroll through the digital inputs by pressing [PAGE UP] or [PAGE DN] buttons, or enter the I/O address then (SEARCH) to search to a specific digital input.

Example, X4 (SEARCH). The state of the digital input will be shown.

View the State of a Digital Output.

[SYSTEM](PMCMNT)(STATUS)

Scroll through the digital outputs by pressing [PAGE UP] or [PAGE DN] buttons, or enter the I/O address then (SEARCH) to search to a specific digital output.

Example, Y3 (SEARCH). The state of the digital output will be shown.

Change the State of a Digital Output.

To change the state of a digital output press (FORCE), highlight the specific bit you want to change. Enter a 0 or 1, then [INPUT]. The state of the output will change.

NOTES: Memory addresses proceeded with the letter;

- A - Alarms.
- X - Inputs
- Y - Outputs
- F – Addresses that are written by the CNC
- G – Addresses that are read by the CNC
- K – Addresses of Keep Relays
- D – Data Table Locations
- T – Timers
- C – Counters

This section is used to set tool length. Please read the section “Setting Tool Length” for more detail.

The Ladder

Ladder – Backup From CNC Controller

Press [SYSTEM](PMCMNT)(I/O)

Highlight USB MEMORY, Highlight WRITE, Highlight SEQUENCE PROGRAM

Enter a filename, for example Y01 (INPUT)(EXEC)

Status line will show: WRITING SEQUENCE PROGRAM

EXECUTING then COMPLETED

Note 1: This backup file is the compiled version of the ladder. Load into FANUC LADDER III software to decompile. You can then view and modify the ladder program.

Note 2: The Ladder backup file can also be from a previous [ALL DATA] backup. It is named PMC1.000 by the backup operation. See FULL SYSTEM BACKUP TO USB.

Ladder III Software – Compile and Export for Loading into CNC Controller

Using Fanuc Ladder III software you can modify the Ladder.

Once modification is complete select Tools/Compile.

When the compile operation has successfully completed it needs to be exported to a USB memory device to transport to the CNC Controller.

Select FILE/EXPORT then browse to the root folder of the USB device.

HINT: Give the export file a simple name using mostly numbers….

Ladder – Load New Ladder into CNC Controller

[SYSTEM](PMCMNT)(I/O)

Highlight USB MEMORY, Highlight READ

Enter the filename (numbers are easy to enter!)(Y01)(INPUT)(EXEC)

ARE YOU SURE YOU WANT TO READ THIS FILE?

(EXEC) to read

(CAN) to cancel operation

STATUS : READING SEQUENCE PROGRAM (PMC1)

EXECUTING then COMPLETED

Highlight FLASH ROM, Highlight WRITE

[EXEC]

STATUS : WRITING SEQUENCE PROGRAM

EXECUTING then COMPLETED

CYCLE POWER TO LOAD NEW LADDER

SYSTEM CONFIGURATION PARAMETERS

For Reference Only – Do Not Modify Unless Instructed To Do So.

PARAMETER #20 I/O Channel

4 = PCMCIA Memory Card
5 = Data Server Memory Interface
9 = Embedded Memory Ethernet
17 = USB Memory

PARAMETER #1001 (Bit 0) INM – Machine input values/parameters units are millimeters.

Set Bit 0 = 0; millimeters.

PARAMETER #1005 (Bit 0) ZRN – HOME REFERENCE REQUIRED AFTER POWER-UP BEFORE AUTOMATIC MOVE COMMAND.

Set Bit 0 = 0 for all axes.

Note: When an absolute position detector is NOT used, Home Reference is required after every Power-up. When an absolute position detector is used, Home Reference is required only under certain circumstances, such as after encoder cable disconnect, battery backup fail…

PARAMETER #1006 – SET ROTARY AXIS TYPE

(Bit 0) ROT = 1 – SET AS B-TYPE ROTARY AXIS
(Bit 1) ROS = 0 – SET AS B-TYPE ROTARY AXIS with Coordinate Roll-Over

PARAMETER #1006 (Bit 5) ZMI – DIRECTION TO TRAVEL FOR MANUAL REFERENCE (HOME) POSITION

X, Ym, Ys = 1 to set home travel in the negative direction
Z, C = 0 to set home travel in the positive direction

PARAMETER #1008 – SET ROTARY AXIS ROLL-OVER AND AXIS TRAVEL COMMAND VALUE MODE

(Bit 0) ROA = 1 – ROTARY AXIS ROLL-OVER IS ACTIVE.
(Bit 1) RAB = 1 – AXIS ROTATES IN (+/-) DIRECTION OF COMMAND VALUE
(Bit 2) RRL = 1 – SET TO 1 FOR RELATIVE COORDINATE ROLL-OVER
(Bit 4) SFD = 0 – ALLOW GRID SHIFT; SFD = 1 ALLOW REFERENCE POINT SHIFT

Note: For 3+2/31iB5
(Bit 1) RAB = 0 – Direction in which path to target is shortest

PARAMETER #1023 – INSTALLED/ATTACHED AXIS IGNORE

-128 for Axis Ignore; 5 for Axis Display Position (4th Axis connect/disconnect).

PARAMETER #1240 – COORDINATE VALUE OF THE REFERENCE POSITION IN THE MACHINE COORDINATE SYSTEM.

After the homing the axis, the position will be set the value entered.
For example; C-Axis = -89.414; B-Axis = 113.866

PARAMETER #1260 – MAXIMUM COORDINATE VALUE BEFORE ROLL-OVER.

360; For 4th Axis

PARAMETER #1320, #1321

STROKE LIMIT

Stroke limits set minimum/maximum axis coordinates which programs must operate within.
[MDI][SYSTEM][PARAM] enter 1320 [NO. SRH]

01320 LIMIT 1+ sets the limit for each axis positive travel. Units are in millimeters. Highlight the field and enter desired number. Include a decimal point. Press [INPUT].
Example:

X	1676.000
Ym	2620.000
Ys	2620.000
Z	1.000

01321 LIMIT 1- sets the limit for each axis negative travel. Units are in millimeters. Highlight the field and enter desired number. Include a decimal point. Press [INPUT].
Example:

X	-1.000
Ym	-1.000
Ys	-1.000
Z	-310.000

PARAMETER #1401#4 – RAPID TRAVERSE OVERRIDE BY KNOB ENABLE

SET BIT#4(RF0) = 1

PARAMETER #1402#1 – JOG SPEED OVERRIDE BY KNOB ENABLE

SET BIT#1(JOV) = 0

PARAMETER #1408#0 – COORDINATE FEED RATE BETWEEN AXIS

(X, Y1,Y2,Z=0; C = 1 (Do not coordinate C-axis)

PARAMETER #1420 – MAXIMUM RAPID FEED RATE

(X=63500. Ym=63500.Ys=63500. Z=12700.)

PARAMETER #1423 – MAXIMUM JOG FEED RATE

(X=2540. Ym=2540.Ys=2540. Z=2540.)

PARAMETER #1424 – MANUAL RAPID FEED RATE

(X=10160. Ym=10160. Ys=10160. Z=5080.)

PARAMETER #1425 – REFERENCE RETURN FEED RATE

(X=500. Ym=500.Ys=500. Z=500.)

PARAMETER #1428 – REFERENCE FEED RATE

(X=2540. Ym=2540.Ys=2540. Z=2540.)

PARAMETER #1430 – MAXIMUM CUT FEED RATE

(X=15240. Ym=15240.Ys=15240. Z=15240.)

PARAMETER #1432 – MAXIMUM CUT FEED RATE (BIPL)

(X=40640. Ym=40640.Ys=40640. Z=40640.)

PARAMETER #1465 – IMAGINARY AXIS DIAMETER

(1465=0 FOR ALL AXES)

PARAMETER #1815 bit 5 – APC; SET TO 0 if NON-APC

Note1: Set to 0 to force homing of axis.
Note 2: Clearing erroneous axis offset after power cycle.
Set 1815 bit 5(APC) and 1815 bit 4(APZ) to 0.
Cycle power, set 1815 bit 5(APC) to 1, cycle power and reference the axis.
Offset should be removed.

PARAMETER #1821 – REFERENCE COUNTER SIZE (Standard X,Y = 23333, Z = 2500)

PARAMETER #1825 – LOOP GAIN (Not to exceed 3000 for any axis. Value must be the same for all axes.)

PARAMETER #1850 - GRID SHIFT (SLAVE MOTOR). (Example, 300 = .3mm)

PARAMETER #2005 BIT 6 (BRKC) = 1 – ENABLE BRAKE CONTROL FUNCTION FOR X,Y,Z
(SEE PARAMETER 2083)
BIT 6 (BRKC) = 0 - DISABLE BRAKE CONTROL FUNCTIONFOR B and C AXES.

PARAMETER #2083 = 500 – BRAKE CONTROL TIMER. VALID ACTIVATION DELAY VALUE (0-1600mS)
Delay motor power disconnect by 'VALUE' after E-STOP is engaged.
Allow Z-BRAKE to fully engage before power down of motors.

PARAMETER #2084 – FLEXIBLE FEED GEAR NUMERATOR (N = 7)
Standard X, Y = 7, Z = 1

PARAMETER #2085 – FLEXIBLE FEED GEAR DENOMINATOR (M = 150)
Standard X, Y = 150, Z = 200
(Example X, Y; Gearbox 3:1, Rack Pitch = .200", Gear = 28 teeth/revolution)
(Example Z; Gearbox - none, Rack Pitch = .200", Gear - none)

PARAMETER #2178 – TRAVERSE LOOP GAIN (Standard X, Z = 2500, Y = 1666).
Smoothes slower traverse rates.

PARAMETER #3004.5 – Over-Travel Bypass – ONLY FOR DIAGNOSTICS
Set To 0 for Normal Over-travel switch functionality
Set To 1 To Bypass Over-travel switch functionality

PARAMETER #3104 – Machine Position Display.

Bit 4 (DRL) = 0 Relative position displayed includes tool length offset.
Bit 4 (DRL) = 1 Relative position displayed does not include tool length offset.
Bit 6 (DAL) = 0 Actual position displayed includes tool length offset.
Bit 6 (DAL) = 1 Programmed position does not include tool length offset.

PARAMETER #3116 – Parameter Write Disable Alarm Clear Option
Bit 0 (PWR) = 0 [CAN][RESET] clears SW0100 Parameter Enable Switch Alarm
Bit 0 (PWR) = 1 [RESET] clears SW0100 Parameter Enable Switch Alarm

PARAMETER #3123 – Screen Saver Enable
Set value = #Minutes to blank screen. 0 disables screen blanking.
Cycle power after changing value.

PARAMETER #3202 – WRITE PROTECT O8000 AND O9000 SERIES PROGRAMS

Bit 4 (NE9) set to 1 will not allow access (view/edit/backup) to O9000 series programs files. These files are allocated for macros or called from other macros.

Bit 8 (NE8) set to 1 will not allow access (view/edit/backup) to O8000 series program files. These files are allocated for macros or called from other macros.

Set bit(s) to 0 to allow access/editing/backup.

If NE8 and NE9 are set to 1 O8000 and O9000 series macros can only be backed up through SRAM backup.

PARAMETER #3204 – Program executed in MDI mode automatically erased upon completion.

Bit 6 (MKP) = 1 Program not erased automatically upon (M30)(%) completion .

PARAMETER #3208 – DISABLE [SYSTEM] BUTTON

Bit 0 (SKY) = 1

PARAMETER #3301 – ENABLE CNC SCREEN CAPTURE

Bit 7 (HDC) = 1

To activate press and hold the [SHIFT] button until the system clock pauses.

PARAMETER #3401 – FORCE DECIMAL POINT ENTERING NUMBER VALUES

Bit 0 (DPI) = 1 Decimal point automatically entered at end of number.

PARAMETER #3454 – Macro write parameter 3115 enable

Bit 4 (G1B) = 1 Macro write parameter 3115 enable

PARAMETER #3457 – SETTING SEARCH FOLDERS FOR PROGRAMS AND MACROS

Bit 0 (LIB) = 0 //CNC_MEM/USER/LIBRARY is set as a search directory
Bit 1 (MC2) = 1 //CNC_MEM/MTB2 is not set as a search directory
Bit 2 (MC1) = 0 //CNC_MEM/MTB1 is set as a search directory
Bit 3 (SYS) = 1 //CNC_MEM/SYS is not set as a search directory
Bit 6 (SCC) = 1 Main program directory is added to search path
Bit 7 (SCF) = 1 Search folder is added for macros

PARAMETER #3708.0 (BIT 0) – SPINDLE ARRIVE at RPM (SAR)

Bit 0 = 1 for normal operation – utilize SAR signal.
Bit 0 = 0 for motor tuning operations – or when SAR signal not needed.

PARAMETER #3730 - SPINDLE SPEED GAIN ADJUSTMENT VALUE

Set spindle to maximum RPM. Measure output voltage – max. voltage is 10 V.
Use the following equation to determine Gain Value.
Value = ((10V)/Measured (V)) x 1000.
Valid Value Range (700-1250)

PARAMETER #3731 - SPINDLE SPEED OFFSET ADJUSTMENT VALUE

Set spindle to 0 RPM. Measure output voltage – max. voltage is 10 V. Use the following equation to determine the offset Value.
Value = (-8196 x Offset Voltage) /12.5
Valid value range (+/-1024)

PARAMETER #3741 - MAXIMUM SPINDLE SPEED

Enter spindle's maximum RPM into field. (Example: 20000)(204EH)
[SYSTEM](PMCMNT)(STATUS)
Note: D4 location is LSB of Hex value of Parameter 3741 (20)
D5 location is MSB of Hex value of Parameter 3741 (4E)

PARAMETER #5005 -TOOL OFFSET OPERATION (31iB5)

Bit 6 (TOS) = 1 Tool length compensation or tool offset operation is performed by shifting the coordinate system. MAXIMUM SPINDLE SPEED

M-CODE MACRO ASSIGNMENT

These parameters specify what macro number to run when a specific M-code is executed.

Parameter #6071 - Macro 09001 - Tool Change Macro

6 (Macro O9001 is run when M06 is executed)
Example; T5 M06 – Change to Tool 5. T0 puts current tool away.

Parameter #6072 – Macro O9002 **-** Tool Touch-Off Macro

401 (Macro O9002 is run when M401 is executed)
Example; T5 M401 – Touch-off Tool 5

Parameter #6073 – Macro O9003 **–** Detach 4th Axis Macro

35 (Macro O9003 is run when M35 is executed)
Example; M35 – Detach 4th Axis Hardware

Parameter #6074 – Macro O9004 **– Attach** 4th Axis Macro

36 (Macro O9004 is run when M36 is executed)
Example; M36 – Attach 4th Axis Hardware

Parameter #6080 – Macro 09020 – Change Servo Motor Tuning Parameter Set

25 (Macro O9020 is run when M25 is executed)
Specify motor tuning parameters based on geometry being cut.
M25 R1. for speed cutting.
M25 R2. for balance of speed and precision.
M25 R3. for precision cutting.

SKIP SIGNAL CONFIGURATION – Parameter 6200

Parameter #6200 – SKIP signal polarity

SKIP signal is valid when signal = 1; Set bit #1(SK0) = 0
SKIP signal is valid when signal = 0; Set bit #1(SK0) = 1

Parameter #6200 – SKIP function feedrate G31F* override by knob enabled.

Set bit#7(SKF) = 1

Parameter #8303 (Bit 7 - SOFx) – Master-Slave position match (For slave motor only.)
Enables position matching function for synchronized axis.

Initialization Procedure: Power machine, home coordinated axis, press E-Stop, Cycle power.

Parameter #8314 – Maximum allowable error in synchronization error check based on machine coordinates.

Y-Slave value = 0; No synchronization error check is made

SV0005 “SYNC EXCESS ERROR (MCN)” occurs if value other than 0 is set and exceeded.

Parameter #8323 – Limit in positional deviation check in axis synchronous control.

Y-Slave value = 0; No position deviation difference check is made

DS0001 “SYNC EXCESS ERROR (POS DEV)” occurs if value other than 0 is set and exceeded.

Parameter #8325 – Maximum compensation value in synchronization establishment based on machine coordinates.

Y-Slave value = 0; Adjust value within specifications.

SV0001 “SYNC ALIGNMENT ERROR” occurs if value is exceeded.

Parameter #11502 – Writing a ‘power-off’ parameter enable using G10L5.

Set (WPP) bit 2 = 1

Parameter #13115 – Softkey input of special characters <>Y%$!:”’ enabled.

Set bit 4 (SI1) = 1
Select (KEYINP) during edit

Parameter #13115#5 – Softkey input of special characters ()?*@_ enabled.

Set bit 5 (SI2) = 1
Select (KEYINP) during edit

Parameter#24303#3 – Enable Cutter Compensation [GEOM (D) WEAR (D)]
Set bit 3 (HD8) = 1 Cycle Power

Parameter #24309 – DataServer – Allows copy of programs and data between USB. Otherwise data only – no programs. WRONG DATA TYPE ERROR OCCURS

Set bit 1 (DUC) = 1

LADDER – STARTING/STOPPING EXECUTION
STOPPING = [SYSTEM](PMCCNF)(PMCST.)(OPRT)(STOP)(YES)
STARTING = [SYSTEM](PMCCNF)(PMCST.)(OPRT)(RUN)(YES)

Servo Tuning Parameters – FOR REFERENCE ONLY – DO NOT CHANGE

These parameters are determined and set using Fanuc Servo Tuning Software by field engineer.

PARAMETER #1620 - Rapid Time Constant for Acceleration/Deceleration

(X=200. Ym=200.Ys=200. Z=200.)

PARAMETER #1621 – RAPID T2 (BELL)

(X=96. Ym=96.Ys=96. Z=32.)

PARAMETER #1622 – CUT TIME CONSTANT

(X=120. Ym=120.Ys=120. Z=50.)

PARAMETER #1623 – CUT FL

(X=50. Ym=50.Ys=50. Z=50.)

PARAMETER #1624 – JOG TIME CONSTANT

(X=100. Ym=100.Ys=100. Z=200.)

PARAMETER #1660 – Related to Accel/Decel

(X=2000. Ym=2000.Ys=2000. Z=2000.)

PARAMETER #1732 – Description not available

(100.)

PARAMETER #1735 – MAX ACC. (CIRCLE)

(X=1500. Ym=1500.Ys=1500. Z=1500.)

PARAMETER #1737 – MAX ACC. (AICC MODE)

(X=1500. Ym=1500.Ys=1500. Z=1500.)

PARAMETER #1738 – Description not available

(100.)

PARAMETER #1769 – Description not available

(X=128. Ym=128.Ys=128 Z=128.)

PARAMETER #1783 – CRITICAL VELOCITY

(X=500. Ym=500.Ys=500. Z=500.)

ADJUSTING VELOCITY GAIN IF MOTOR SQUEAL ON DUAL MOTOR AXIS.

[MDI][SYSTEM][PRMSET][SERVO TUNING][OPRT][SELECT]

Press [PG DN] until desired axis is shown (Ym and Ys). Lower the value in the VELOC GAIN field. Press [INPUT] to save parameter. Enter same value for second motor.

Laguna Fanuc System values; X = 250, Ym = 225, Ys = 225, Z = 250.

Appendix

M-CODE LIST

M03 – Spindle On - Clockwise

M04 – Spindle On – Counter-Clockwise

M05 – Spindle Off

M06 – Tool change. Example: “M06T2” changes to tool 2. (T0 puts tool away) (Spindle must be down.)

M13 – Drilling block rotation on. (Spindle must be up.) (T11;T12;T13;T14;T15;T16;T17;T18;T19;)

M15 – Drilling block rotation off (T0; retract all tools on drilling block)

M25 - Specify servo motor tuning parameters (O9020) based on geometry being cut.
M25 R1. for speed cutting.
M25 R2. for balance of speed and precision.
M25 R3. for precision cutting.

M27 – Extend tool changer cylinder

M28 – Retract tool changer cylinder

M30 – End program

M35 – Attach 4th Axis – For systems with attachable/detachable turner hardware

M36 – Detach 4th Axis – For systems with attachable/detachable turner hardware

M37 – Mister Control – On (if equipped).

M38 – Mister Control – Off.

M48 – Dusthood up – For systems equipped with dusthood hardware

M49 – Dusthood down – For systems equipped with dusthood hardware

M52 – Spindle piston up

M53 – Spindle piston down

M401 – Automatic touch-off. Example, “M401T2” command will automatically touch off tool 2. The specified tool will automatically be loaded into spindle.

M-Codes used by Macro Programs

M16 – Start ATC
M17 – Index magazine to new tool number.
M18 – Update Tool# in spindle. Copies variable #405 to #402.
M19 – Index magazine to match tool# in spindle.

M20 – Main spindle tool grip
M21 – Main spindle tool release

Sample G-Code

The characters after semicolon on each line are comments and are not read by the machine.

```
G00                 ; rapid mode
G17                 ; Circular interpolation and cutter diameter compensation in the XY plane
G27                 ; Continuous sequence operation with automatic speed reduction on corners
G40                 ; Disables cutter diameter compensation
G70                 ; Programming in inches
G80                 ; Disables fixed cycles
G90                 ; Absolute programming
G79  Z0             ; Programming referred to machine zero, move z axis to zero.
T0                  ; retract tools on drill block
(UAO,1)             ; use origin offset 1
M63 T11             ; extend tool 11 on drill block
M13                 ; turn on drill block rotation
G00 X2 Y2           ; rapid move to X2 Y2
G00 Z3
G01 Z-0.25 F300     ; Feed move to Z-0.25
G01 Z3
M15                 ; turn off drill block rotation
T0                  ; retract tool on drill block
M6T2                ; Tool change to tool 2 on main spindle
M3 S12000           ; turn on the spindle to 12000RPM
G00 X2 Y2
G00 Z3
G01 Z-0.25
G01 Z3
M05                 ; turn off spindle
M30                 ; end of G-code program
```

The Fanuc PC Front-End software consists of two modules, NCBoot32.exe and CNCScrn.exe. NCBoot32 is low level software that establishes communication with the CNC. CNCScrn is the graphical user interface.

NCBoot32 loads automatically when the PC boots. Its icon is displayed in the program tray and is shown below. Once it establishes communication with the CNC hardware it launches CNCScrn.exe. The system is now ready to be operated.

CNCScrn.exe can be started manually by selecting CNC Screen Display Function from the Windows program list or by double-clicking on its desktop icon as shown below.

The following describes functional differences between ‘standard’ Fanuc Oi-MF systems and those equipped with a PC.

Maintenance

Performing regular maintenance will ensure optimal performance of the machine. Please follow these maintenance procedures.

Failure to follow maintenance procedures will void the warranty.

Table is based on 30 hours of use a week						
	Daily	**Weekly**	**Monthly**	**Every Three Months**	**Every Six Months**	**Yearly**
Check the tool blades for chips and dullness	X					
Remove all tooling from the spindle at the end of the day. Do not leave any tooling in the spindle overnight. This includes tool cones, collets, router bits, etc. Leaving any tooling in the spindle overnight can cause the tooling to get dirty, stuck, rust, and cause damage to the spindle.	X					
Clean the collets and spindle holes. Uncleaned spindle holes may affect cut quality and pose a safety hazard if significantly dirty. Uncleaned spindle holes may affect cut quality and may pose a safety hazard if significantly dirty.	X					
Clean the router bits	X					
Clean surface dust	X					
Clean the x and y-axis rack rails and the z-axis screw guides		X				
Check the dust extraction for blockages, as large pieces could cause blockages		X				
Inspect the overall machine for damage and loose or worn parts.	X	X	X	X		
Check lubrication level and fill with 10wt oil lubricant if necessary		X				
Disconnect power at the wall and clean the dust from the cabinet and fan covers			X			