

GE FANUC 0i MC OPERATOR'S 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

The content of this manual has been reviewed for accuracy.

Differences may exist and we cannot guarantee that they are completely covered in this document.

The information in this document is reviewed regularly and any necessary changes will be incorporated in the next revision. We welcome any suggestions for improvement.

Material is subject to change without notice.

This manual is for the exclusive use of Fadal Customers and Distributors. The reproduction, transmission or use of this document or its contents is not permitted without the expressed written permission of Fadal Machining Centers. All rights, including rights created by patent grant or registration of a utility, model or design, are reserved.

© Copyright 2002 Fadal Machining Centers.

TABLE OF CONTENTS

TABLE OF CONTENTS	i
1.0 POWER ON/OFF	1
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	3
1.2 POWER ON/OFF	4
1.2.1 POWER ON	4
1.2.2 POWER OFF	5
1.2.3 ELECTRICAL CABINET DOOR	5
2.0 CNC CONTROL	7
2.1 CONTROL LAYOUT	8
2.1.1 GE FANUC LCD	9
2.1.2 SOFTKEYS	9
2.1.3 GE FANUC MANUAL DATA INPUT	9
2.1.4 OPERATOR PANEL A	10
2.1.5 OPERATOR PANEL B	21
2.1.6 MANUAL PULSE GENERATOR	23
2.1.7 PCMCIA CARD PORT	23
2.1.8 RS 232	23
2.1.9 BECON	23
3.0 CONTROL FUNCTION	25
3.1 OPERATION MODE SELECTION	26
3.1.1 AUTO MODE	26
3.1.2 EDIT MODE	28
3.1.3 MDI MODE	29
3.1.4 REMOTE MODE	31
3.1.5 REF RETURN MODE	35
3.1.6 JOG MODE	37
3.1.7 HANDLE MODE	38
3.2 SAFETY FUNCTIONS	39
3.2.1 EMERGENCY STOP	39
3.2.2 DOOR OVERRIDE	39
3.2.3 WORKLIGHT	39
3.2.4 ALARM MESSAGE	39
3.3 TEST FUNCTION MODES	40
3.3.1 SINGLE BLOCK	40
3.3.2 BLOCK DELETE	40
3.3.3 OPTION STOP	40

3.3.4 PROGRAM RESTART	40
3.3.5 MC LOCK	41
3.3.6 DRY RUN	41
3.4 SETUP MODES	42
3.4.1 FUNCTION KEY	42
3.5 TOOL OPERATION	43
3.5.1 DRUM FWD	43
3.5.2 DRUM REV	43
3.5.3 TOOL REL	43
3.5.4 T-COMMAND (TOOL NUMBER)	44
3.5.5 TOOL CHANGER COMMAND	45
3.6 AXIS DIRECTION SELECTION	56
3.6.1 X+ AXIS	56
3.6.2 X- AXIS	56
3.6.3 Y+ AXIS	56
3.6.4 Y- AXIS	57
3.6.5 Z+ AXIS	57
3.6.6 Z- AXIS	57
3.6.7 A+ AXIS	57
3.6.8 A- AXIS	57
3.6.9 RAPID OVERRIDE	57
3.7 SPINDLE OPERATION	58
3.7.1 SPINDLE STOP	58
3.7.2 SPINDLE CLOCKWISE	58
3.7.3 SPINDLE COUNTER-CLOCKWISE	58
3.7.4 MANUAL SPINDLE SPEED	58
3.7.5 M CODE / S CODE	58
3.7.6 SPINDLE SPEED RANGE	59
3.7.7 SPINDLE OVERRIDE	60
3.7.8 SPINDLE ORIENTATION	60
3.7.9 SPINDLE SPEED OVERRIDE	60
3.7.10 SPINDLE CALIBRATION	60
3.8 OPERATION CONTROL	61
3.8.1 FEED/RAPID/JOG OVERRIDE	61
3.8.2 MANUAL PULSE GENERATOR	62
3.8.3 HANDLE	62
3.9 EXECUTION FUNCTION	62
3.9.1 CYCLE START	62
3.9.2 FEED HOLD	63
4.0 M/C SET UP	65
4.1 USER PREFERENCE	67
4.1.1 PROGRAM COORDINATE UNITS	68
4.1.2 T/H/D ALIGNMENT CHECK	69
4.1.3 TOOL CHANGE ADDS D/H OFFSETS	70
4.2 AXES SET UP	71
4.2.1 4TH (A) AXIS AMP FITTED	71
4.2.2 4TH (A) AXIS FITTED	71
4.3 AUTO TOOL CHANGER	72
4.3.1 TOOL CHANGE TYPE	72

4.4 TOOL LOAD MONITOR	73
4.4.1 POWER SET	74
4.4.2 TOOL TYPE	76
4.4.3 DATUM	77
4.4.4 LIMIT	78
4.4.5 ACTION	80
5.0 LENGTH SETTING	81
5.1 SETTING TOOL ORIGIN POINT	82
5.2 SETTING TOOL LENGTH	84
5.3 TOOL DIAMETER/RADIUS SETTING	85
5.4 TOOL LENGTH DIAMETER/RADIUS ADJUSTMENT	86
6.0 DATUM SETTING	87
6.1 DATUM EDGE LOCATION	88
6.1.1 X - AXIS LOCATION	88
6.1.2 Y - AXIS LOCATION	90
6.1.3 Z - AXIS LOCATION	91
6.2 COPYING OFFSETS	92
6.3 GLOBAL DATUM SHIFT	93
6.4 G92 POSITION SET	94
6.5 G92 POSITION SET CANCEL	97
7.0 CALCULATOR	99
7.1 TRIANGLE SOLVER	100
7.2 CIRCLE SOLVER	101
7.3 SPEED AND FEED CALCULATOR	102
8.0 QUICK CUT	107
8.1 SPINDLE CONTROL	109
8.2 FEED CONTROL	109
9.0 AICC FUNCTION	111
9.1 PROGRAMMING CONSIDERATIONS	112
9.2 AICC PRECISION LEVEL DATA SCREEN	114
10.0 TOOL MANAGEMENT	117
10.1 TOOL LIFE MANAGEMENT	118
10.2 TOOL LIFE MANAGEMENT ACCESS	121
10.3 ADD A TOOL	121
10.4 ADD MORE TOOLS	122
10.5 TOOL LIFE LIMIT SETTING	122

10.6 DELETE A TOOL	123
10.7 DELETE ALL TOOLS	124
10.8 RESET "TOOL LIFE EXPIRED" STATUS FOR THE TOOL	124
10.9 RESET "TOOL LIFE EXPIRED" STATUS FOR ALL TOOLS	125
10.10 TOOL SKIP	126
11.0 MANUAL GUIDE I	127
11.1 MANUAL GUIDE I FUNCTION	128
12.0 DATA I/O	133
12.1 DATA I/O FUNCTION	134
13.0 COOLANT CONTROL	137
13.1 AUTO COOL	138
13.2 FLOOD COOLANT	138
13.3 MIST COOL	138
13.4 COOLANT THROUGH SPINDLE (OPTIONAL)	138
14.0 DATA SERVER.....	139
14.1	139
15.0 APPENDIX	141
15.1 M CODE LIST	142
15.2 PLC ALARM LIST	144
15.3 PLC MESSAGES	147
15.4 G CODES	150
15.5 FINAL BACK UP PROCEDURE	150
15.6 COLD START POSITION SETUP PROCEDURE	154
15.7 PROB	157
INDEX	159
RENISHAW	165

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 (optional)

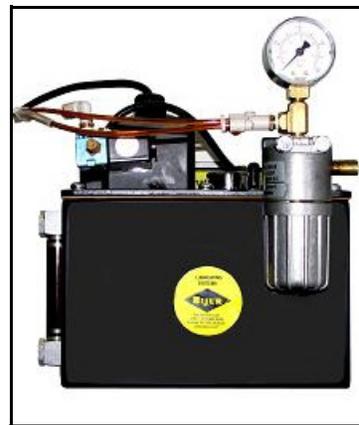


Figure 1-2: Way Lube Reservoir

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

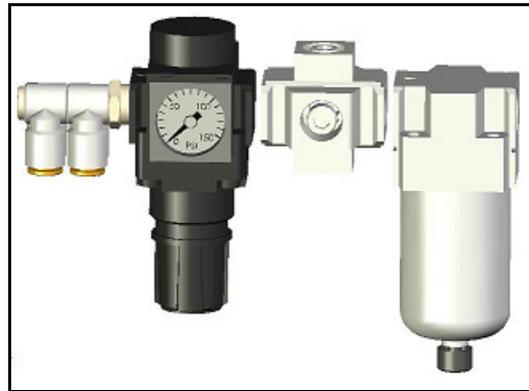


Figure 1-3: Air Pressure

1.1.3 WATER RESERVOIR

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 (Back side of the machine)

1.1.5 COOL POWER RESERVOIR (OPT)

Examine the cool power reservoir once a month.

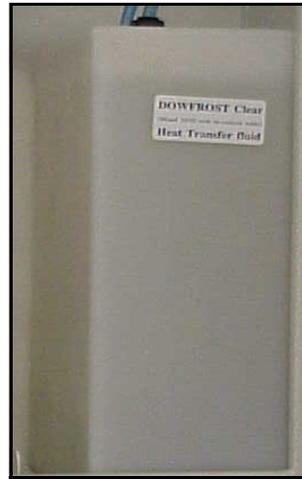


Figure 1-5: Cool Power Reservoir

1.2 POWER ON/OFF

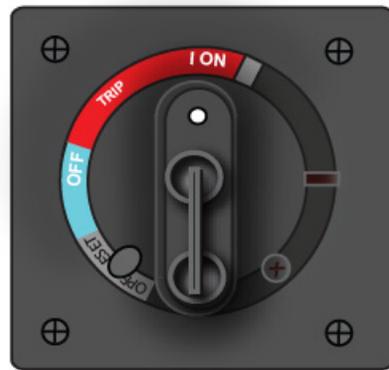


Figure 1-6: Power Switch

1.2.1 POWER ON

To turn the VMC ON:

1. Turn the power switch CW onto ON position.
2. Press the CNC control ON, [I] button on the operator panel.

When the machine is powered on, it will enter the Fadal custom screen (Figure 1-7:).



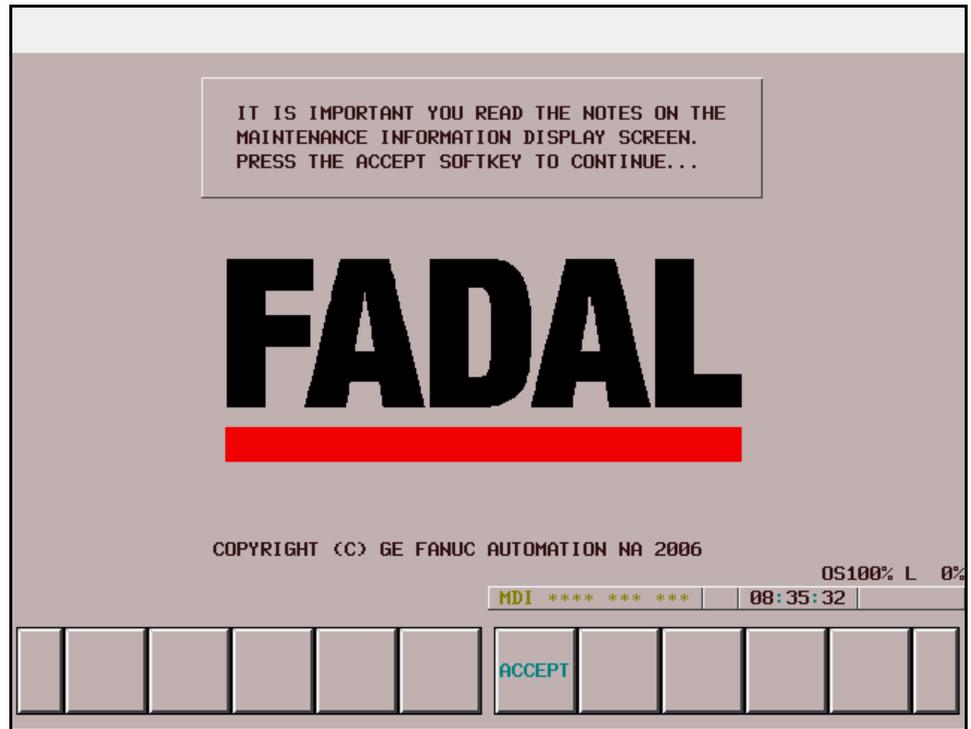


Figure 1-7: Fadal Custom Screen.

It is important you read the notes on the maintenance information display screen.

3. Press [Accept] softkey.

CNC will enter the System Configuration screen automatically.

4. Press POS pushbutton on the MDI panel to enter position display screen if needed.

1.2.2 POWER OFF

To turn the VMC OFF:

1. Press Emergency Stop button.
2. Press the CNC control OFF, [O] button on the operator panel.
3. Turn the power switch CCW onto OFF position.



1.2.3 ELECTRICAL CABINET DOOR

To lock the electrical cabinet door, push the power switch in and hold it hard until the click sound indicating that the door is locked.

To unlock the electrical cabinet door, turn the power switch CCW onto OPEN/RESET position.

2.0

CNC CONTROL

2.1.1 GE FANUC LCD



Figure 2-2: GE Fanuc LCD

LCD displays comprising screen indicating the operation and setup status of the machine.

2.1.2 SOFTKEYS



Figure 2-3: LCD softkeys

The softkeys are software controlled and screen-sensitive. Their functions change each time screen is displayed. Detailed description of the use of the softkeys is given in the GE Fanuc Operator Programming Manual supplied on CD-Rom with the machine.

2.1.3 GE FANUC
MANUAL DATA
INPUT

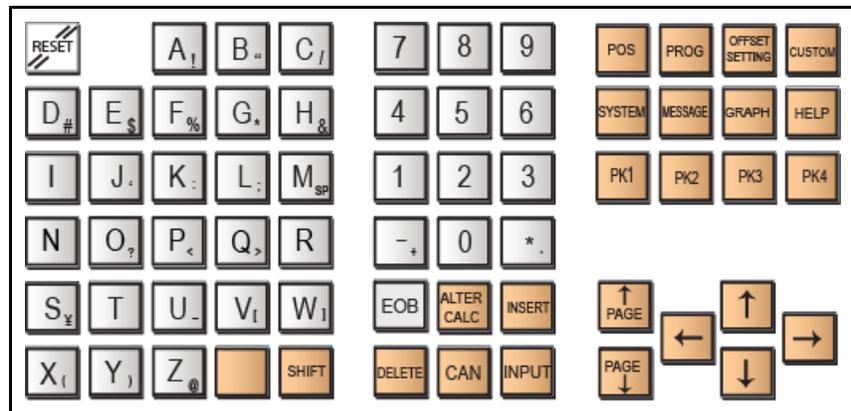


Figure 2-4: Manual Data Input Panel (US)

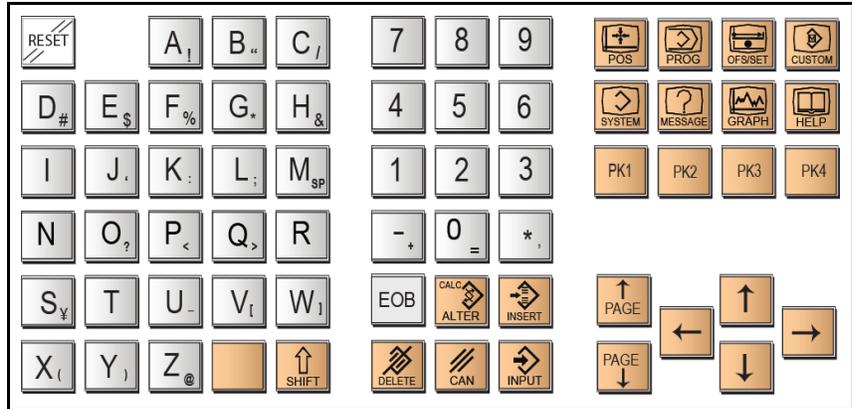


Figure 2-5: Manual Data Input Panel (CE)

Manual Data Input (MDI) panel is used for simple test operations. Detailed description of the use of the MDI panel is given in the GE Fanuc Operator Programming Manual supplied on CD-Rom with the machine.

2.1.4 OPERATOR PANEL A

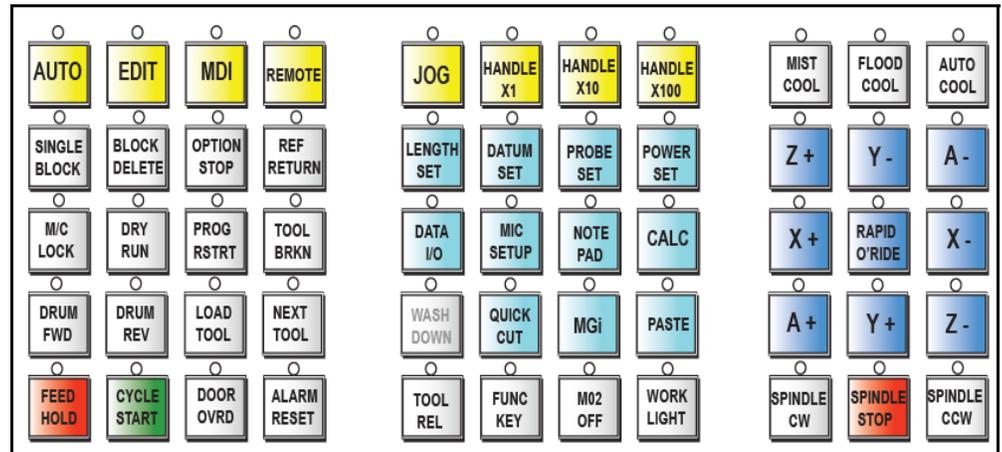


Figure 2-6: Operator Panel A (US)

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		REMOTE (DEEP FEED)	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. For more detailed information see section 3.1.4
		SINGLE BLOCK	Allows to execute part program block by block. Single Block mode is implemented to toggle ON/OFF with press of pushbutton. Pressing Single Block switch starts the single block mode. When Cycle Start pushbutton 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.
		BLOCK DELETE	Skip execution of program block (/). Multi level Block Skip is not supported. Block Delete is implemented to toggle the ON/OFF with press of pushbutton.
		OPTION STOP	Execution of the program will stop at M01 when Option Stop pushbutton is ON. Operator needs to press Cycle Start pushbutton to restart the program. It does not effect the program when the Option Stop pushbutton is OFF.
		REFERENCE RETURN MODE	Opportunity to return all the axes to the machine zero position. For more detailed information see section 3.1.5
		MACHINE LOCK	Machine Lock enables execution of part program without axis motion, but M/S/T command still is able to execute. This pushbutton is for test purpose. Machine Lock is implemented to toggle ON/OFF with press of pushbutton.

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		DRY RUN	Dry Run feed rate forces program federate to fixed "dry run" rate to speed non-cutting testing of part programs. DRY RUN is implemented to toggle On/Off with press of pushbutton except AUTO and REMOTE mode. For more detailed information see section 3.3.4
		PROGRAM RESTART	Program Restart provides the facility for restarting a program at a chosen sequence block number following a program interruption For more detailed information see section 3.3.4
		TOOL BROKEN	Tool Broken function enables operator to mark tool as damaged without editing Tool Management Data. FUNC pushbutton must be pressed simultaneously with TOOL BRKN to execute the program. Pressing TOOL BRKN and FUNC buttons at the same time is a shortcut to mark active tool as broken/damaged in the tool management. Tool management will not select a damaged tool for use from the tool group.
		DRUM FORWARD	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 pushbutton for indexing of multiple tool pots, or press momentarily to index one tool.
		DRUM REVERSE	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 .

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		LOAD TOOL	<p>ATC: Load Tool pushbutton is used to load a tool from the tool drum into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.</p> <ul style="list-style-type: none"> • Press JOG button. • Press DRUM FWD or DRUM REV button to move the carousel to the pocket has the required tool. • Press FUNC + LOAD TOOL will load the tool aligned with the spindle into the spindle. It will return the tool of spindle to the carousel first, movr the carousel to the tool which pocket was aligned with the spindle, then load that tool into the spindle. • If the tool number at the current position is in the spindle, it will do nothing. • Operator can see the tool number changing from tool management screen. <p>DATC: Load Tool pushbotton is used to load a tool from spindle into an empty spot of magazine. It is only active when the operator door is closet and CNC is in the Jog mode.</p> <ul style="list-style-type: none"> • Press Jog button. • Pressing FUNC + LOAD TOOL will load a tool from spindle into an empty spot of magazine. • If the tool number at the current position is in the spindle, it will do nothing. • Operator can see the tool number changing from tool management screen.

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		NEXT TOOL	<p>ATC: Next Tool pushbutton is used to load a tool which is next to the current aligned position from the tool drum into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.</p> <ul style="list-style-type: none"> • Press JOG button. • Pressing FUNC + NEXT TOOL will load the next pocket tool into the spindle. It will return the tool in the spindle to the tool drum first, then increment the tool drum 1 pocket and load the tool into the spindle. • Operator can see the tool number changing from tool management screen. <p>DATC: Next Tool pushbutton is used to load current pocket tool into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.</p> <ul style="list-style-type: none"> • Press Jog button. • Pressing FUNC + NEXT TOOL will swap the spindle tool with current pocket tool. • Operator can see the tool number changing from tool management screen.
		FEED HOLD	Feed Hold pushbutton stops all feed motors and suspends the CNC cycle.
		CYCLE START	Cycle Start pushbutton causes CNC program to start.
		DOOR OVERRIDE	DOOR OVRD pushbutton 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 pushbutton is momentary. Momentary function requires operator to hold pushbutton while door override is required.

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		DOOR UNLOCK	DOOR UNLOCK pushbutton is use for unlock the CE front door wten the following safety condition is satisfied: all axes motion is stop, spindle is stop, tool change cycle is finished and hardware monitor board agrees to be able open the front door.
		ALARM RESET	Alarm Reset pushbutton resets the CNC and clears the alarm/message.
		JOG	In the JOG mode, pressing the direction switch on the operator's panel moves the tool along with the selected axes in the selected direction. For more detailed information see section 3.1.6
		HANDLE X1	<p>In the HANDLE mode, operator can move the axes using the Manual Pulse Generator (MPG) on the operator's panel.</p> <p>If machine is in metric mode the increments are:</p> <p>Handle x 1= 0.001mm Handle x 10= 0.010mm Handle x 100= 0.100mm</p> <p>If machine is in inch mode the increments are:</p> <p>Handle x 1= 0.0001 inch Handle x 10 = 0.001 inch Handle x 100 = 0.01 inch</p> <p>For more detailed information see section 3.1.7</p>
		HANDLE X10	
		HANDLE X100	
		LENGTH SET	Length Set pushbutton enables tool length setting screen mode. For more detailed information see section 5.0
		DATUM SET	DATUM SET pushbutton enables datum setting mode. For more detailed information see section 6.0
		PROBE SET	Probe Set pushbutton enables machine to enter the Renishaw probe setup screen.

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		POWER SET	The POWER SET pushbutton enables machine to display the Tool Load monitoring screen. It is used to display and monitor axes drive and spindle load information.
		DATA I/O	Data I/O pushbutton opens the Data I/O screen. User can transfer the program between CNC memory and flash card. For more detailed information see section 12.0
		M/C SETUP	Machine Configuration Setup pushbutton enables the machine configuration displays; this allows the user to configure the machine tool for user preferences. For more detailed information see section 4.0
		NOTE PAD	NOTE PAD pushbutton provides the operator with the facility to write and save text messages.
		CALCULATOR	CALC pushbutton enables the calculator function. Three modes are available: Triangle solver. Circle solver. Speed and feed calculator For more detailed information see section 7.0
		QUICK CUT	QUICK CUT pushbutton enables the quick cut feature which is used to quickly input speeds and feeds for manual machine operation. For more detailed information see section 8.0
		MGi	MGi pushbutton enables CNC to enter Manual Guide i mode. For more detailed information see section 4.0
		PASTE	PASTE pushbutton is used in conjunction with the Calculator. The LED flashes when data are available to paste into a data entry field.

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		TOOL RELEASE	<p>Tool Release (Draw Bar Open)</p> <p>This button is for manually loading/unloading a tool holder. Operator must be prepared to catch tool from spindle cartridge immediately upon pressing this pushbutton with FUNC pushbutton at the same time. Spindle air is turned on along with release of tool. Draw bar will open after pressing the pushbutton more than 1 second and remain open while push button is still held.</p> <p>For more detailed information see section 3.5.3</p>
		FUNCTION KEY	<p>To prevent accidental activation of selected operator panel operations, the FUNC pushbutton is required to be pressed in combination with desired operator panel function.</p> <p>Operator Panel functions currently requiring FUNC pushbutton are:</p> <ul style="list-style-type: none"> SPINDLE CW SPINDLE CCW TOOL BRKN LOAD TOOL NEXT TOOL TOOL REL DRY RUN
		M02 OFF	<p>M02 OFF pushbutton is used to disable the axis and spindle drives at the end of a program, if program ends with M02 code, and can be turned ON and OFF at will. When this feature is ON the LED is ON.</p> <p>M02 OFF pushbutton is also used to automatically remove power to the axis and spindle drives when the machine cycle is completed after the end of a shift.</p>
		WORK LIGHT	<p>Operator control of machine's work light(s). Pressing WORK LIGHT pushbutton toggles work light On/ Off.</p> <p>Machine work light is set to automatically turn on with power-up of machine.</p>

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		MIST COOL	MIST COOL turns on/off the mist coolant pump and coolant through spindle solenoid when the AUTO COOL is off. 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.
		FLOOD COOL	FLOOD COOL turns on/off the flood coolant pump when the AUTO COOL is OFF. FLOOD COOL and MIST COOL may be turned on simultaneously.
		AUTO COOL	AUTO COOL pushbutton selects automatic control of coolant by programmed M-Code. AUTO COOL pushbutton toggles on/off, as indicated by LED. While AUTO COOL is active, manual coolant control pushbuttons 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
		Z POSITIVE	Z+ AXIS pushbutton is used for continuous axis jogging in the Z+ direction. Select Z+ axis in the Handle mode.
		Y NEGATIVE	Y- AXIS pushbutton is used for continuous axis jogging in the Y- direction. Select Y- axis in the Handle mode.
		A NEGATIVE	A- AXIS pushbutton is used for continuous axis jogging in the A- direction. Select A- axis in the Handle mode.

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		X POSITIVE	X+ AXIS pushbutton s used for continuous axis jogging in the X+ direction. Select X+ axis in the Handle mode.
		RAPID OVERRIDE	RAPID O'RIDE can be operated simultaneously with any power feed push-button to produce rapid axis jogging, when the operator door is closed
		X NEGATIVE	X- AXIS pushbutton is used for continuous axis jogging in the X- direction. Select X- axis in the Handle mode.
		A POSITIVE	A+AXIS pushbutton is used for continuous axis jogging in the A+ direction. Select A+ axis in the Handle mode.
		Y POSITIVE	Y+ AXIS pushbutton is used for continuous axis jogging in the Y+ direction. Select Y+ axis in the Handle mode.
		Z NEGATIVE	Z-AXIS pushbutton is used for continuous axis jogging in the Z- direction. Select Z- axis in the Handle mode.
		SPINDLE CLOCKWISE	Manual Spindle Start Clock-Wise & Spindle Jog CW FUNC pushbutton must be pressed simultaneously with SPDL CW to execute program. For more detailed information see section 3.7
		SPINDLE STOP	SPDL STOP applies stop to spindle running in either CW or CCW direction. SPDL STOP pushbutton 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. For more detailed information see section 3.7

Operator Panel A pushbuttons

BUTTON (US)	BUTTON (CE)	DESCRIPTION	FUNCTION
		SPINDLE COUNTER CLOCKWISE	Manual Spindle Start Counter Clock-Wise & Spindle Jog CCW FUNC pushbutton must be pressed simultaneously with SPDL CCW to execute program. For more detailed information see section 3.7

2.1.5 OPERATOR PANEL B

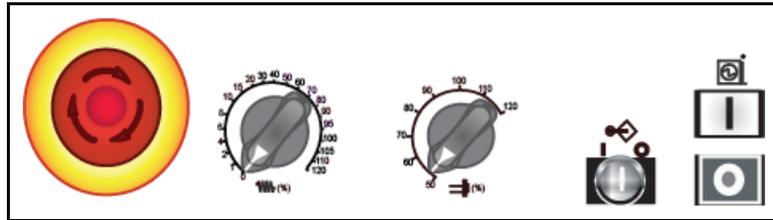
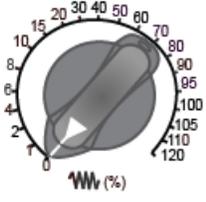
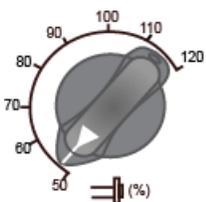


Figure 2-8: Operator Panel B

Operator panel B allows the operator to control machine operation, and is used in conjunction with the display screen and with the other panels on the control station.

Operator Panel B

BUTTON	DESCRIPTION	FUNCTION
	<p>EMERGENCY STOP</p>	<p> CAUTION Operating this button with the machine in a cutting cycle may damage cutting tools and workpieces. Failure to heed this Caution may result in damage to equipment. Pressing this button causes an immediate slide and spindle stop, followed by the removal of drive power. Any mechanisms in operation will stop immediately. The button remains latched-OFF when operated, and must be un-latched by turning the head of the button. The button is instantly active whenever it is pressed</p>
	<p>FEED/RAPID/JOG OVERRIDE</p>	<p>Feed/Rapid/Jog Override selector switch is used to override programmed feed rate, rapid traverse, and jog rate.</p>
	<p>SPINDLE SPEED OVERRIDE</p>	<p>Spindle Speed Override selector switch is used to override the programmed speed within the range 50 to 120% of the programmed spindle speed. The amount of override selected will affect all machining processes until it is changed by the operator.</p>
	<p>MEMORY PROTECTION KEYSWITCH</p>	<p>This key switch is used to prevent part programmes, offset values, parameters, and setting data from being accidentally edited, registered, modified, or deleted. The switch can be enabled in all operating modes, and the named functions are LOCKED when the MEMORY PROTECTION KEYSWITCH is in the (I) PROTECTION ON position, when the key may be removed for security. When the key is in the (O) position the key is retained and is unable to be removed.</p>

Operator Panel B

BUTTON	DESCRIPTION	FUNCTION
	POWER OFF	Power OFF button powers Off the control.
	POWER ON	Power On button powers On the control.

2.1.6 MANUAL PULSE GENERATOR

When Handle mode is selected, clockwise (CW) rotation of the handwheel gives positive axis motion, counterclockwise (CCW) rotation will give negative axis motion. When in Handle mode, both axis selection and incremental distance moved, are made using the selection and increment pushbuttons on the Operator panel A.



Figure 2-9: Manual Pulse Generator

2.1.7 PCMCIA CARD PORT

Detailed description of the use of the PCMCIA card port is given in the GE Fanuc Operator Programming Manual supplied on CD-Rom with the machine.

2.1.8 RS 232

The machines are supplied with data ports beneath a hinged protective cover. The twenty five (25) pin port fitted to the side of the control panel is used to interface any RS 232 serial data input/output device with the machine.

2.1.9 BECON

Beacon light has two modes: OFF or ON (Blinking).

It is ON and flashing when machine is in Auto, MDI, Waiting or Feed Hold mode. Beacon light is used to remind the operator to reactivate the machine. Other than that it is OFF.

3.0

CONTROL FUNCTION

3.1 OPERATION MODE SELECTION

3.1.1 AUTO MODE



AUTO (MEM) mode runs a part program registered in the active program memory. For detailed information, see GE Fanuc Operators Programming Manual supplied with the machine.

The screen below is shown in Auto mode.

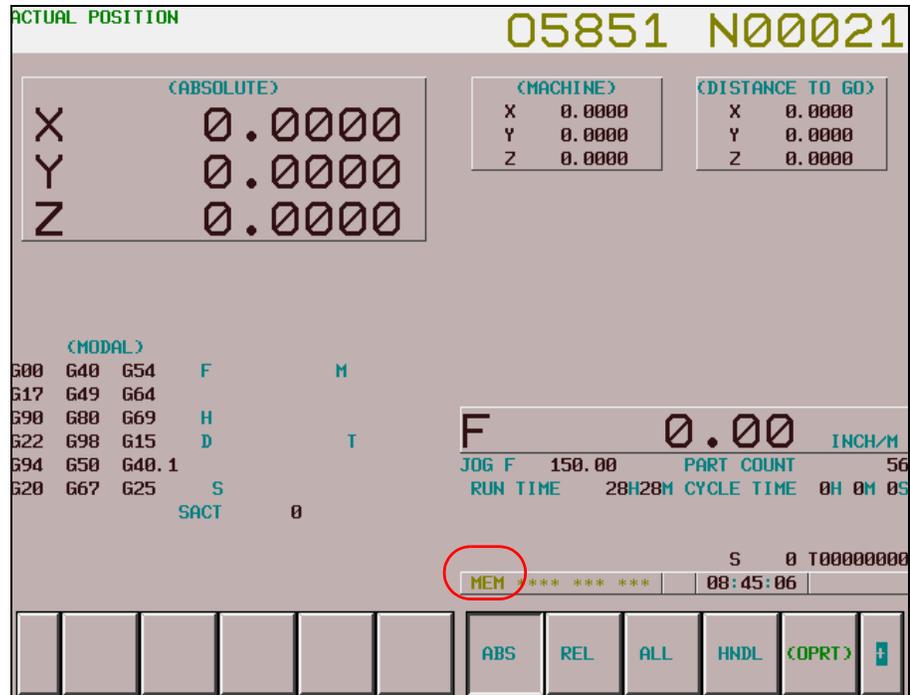


Figure 3-1: Auto (Memory Mode)

Circled MEM on the Figure 3-1: indicates that operation is in Auto mode.

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

When the FEED HOLD pushbutton on the machine operator's panel is pressed during automatic operation, this operation is stopped temporarily. When the CYCLE START pushbutton is pressed again, automatic operation is restarted.

When the RESET pushbutton 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 AUTO (MEMORY) mode selection pushbutton.
2. Select the program from the registered programs doing the following steps:
3. Press PROG pushbutton on the MDI panel to display the program screen.
4. Press [+] softkey.
5. Press [DIR] softkey and the program library will be displayed.
6. Enter a program number using the numeric keys.
7. Press the [O SRH] softkey.
8. Press the CYCLE START pushbutton on the machine operator's panel A.

Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.

To stop or cancel memory operation midway through, follow the steps below:

Stopping memory operation.

1. Press the CYCLE STOP pushbutton on the machine operator's panel A. The FEED HOLD LED goes on and the CYCLE START LED goes off. The machine responds as follows:
 - 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.

1. 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.

3.1.2 EDIT MODE



EDIT mode edits its part programs registered in the active program memory. For detailed information, see GE Fanuc Operators Programming Manual supplied with the machine.

The screen below is shown in Edit mode.

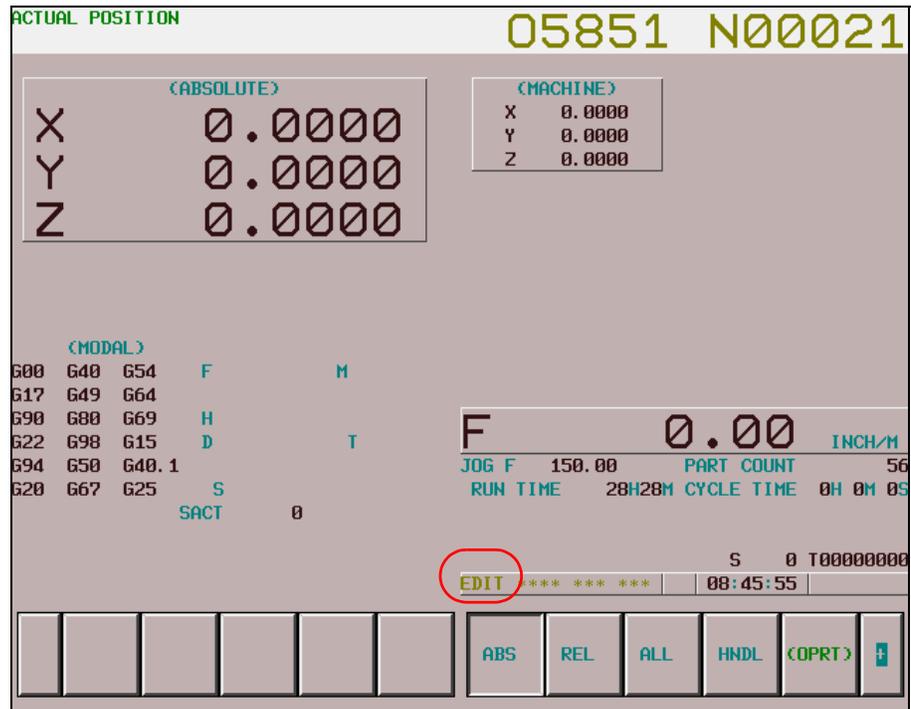


Figure 3-2: Edit Mode

Circled EDIT on the Figure 3-2: indicates that operation is in Edit mode.

3.1.3 MDI MODE



MDI mode enables a program of up to 10 blocks to be created and processed from the MDI buffer memory. For detailed information, see GE Fanuc Operators Programming Manual supplied with the machine.

The screen below is shown in MDI mode.

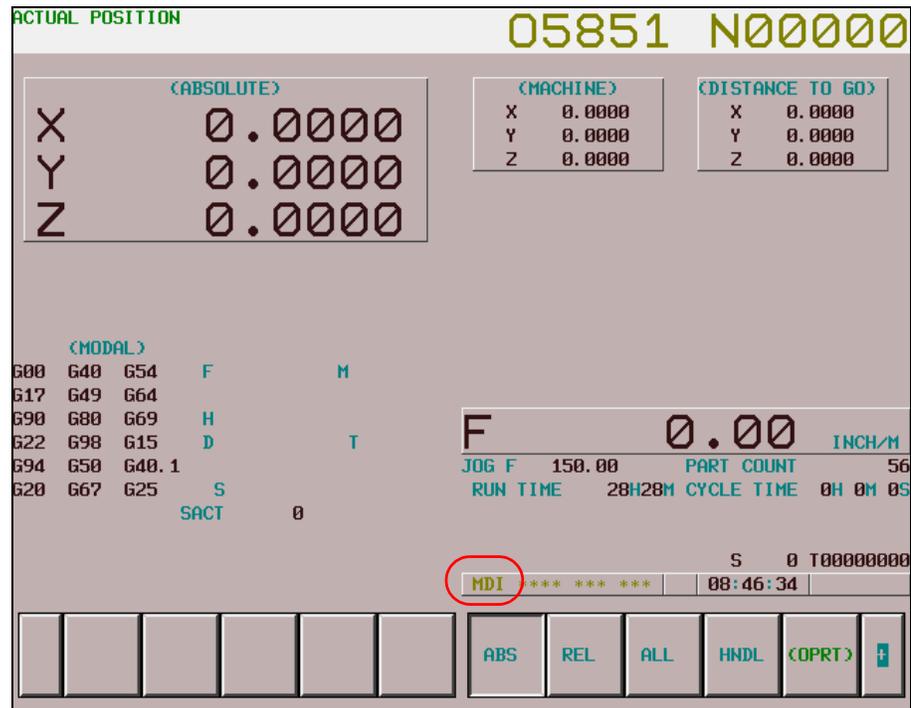


Figure 3-3: MDI Mode

Circled MDI on the Figure 3-3: indicates that operation is in MDI mode.

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 pushbutton.
2. Press the PROG pushbutton 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 pushbutton on the MDI panel.
 - Alternatively, press the RESET pushbutton.
5. To execute a program, set the cursor on the head of the program. Push CYCLE START pushbutton 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 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 FEED HOLD pushbutton on the machine operator's panel. The FEED HOLD LED goes on and the CYCLE START LED goes off.
 - Terminate MDI operation.
7. Press the RESET pushbutton 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.

3.1.4 REMOTE MODE



REMOTE mode is also called DNC mode. In the REMOTE 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.

The screen below is shown in REMOTE mode.

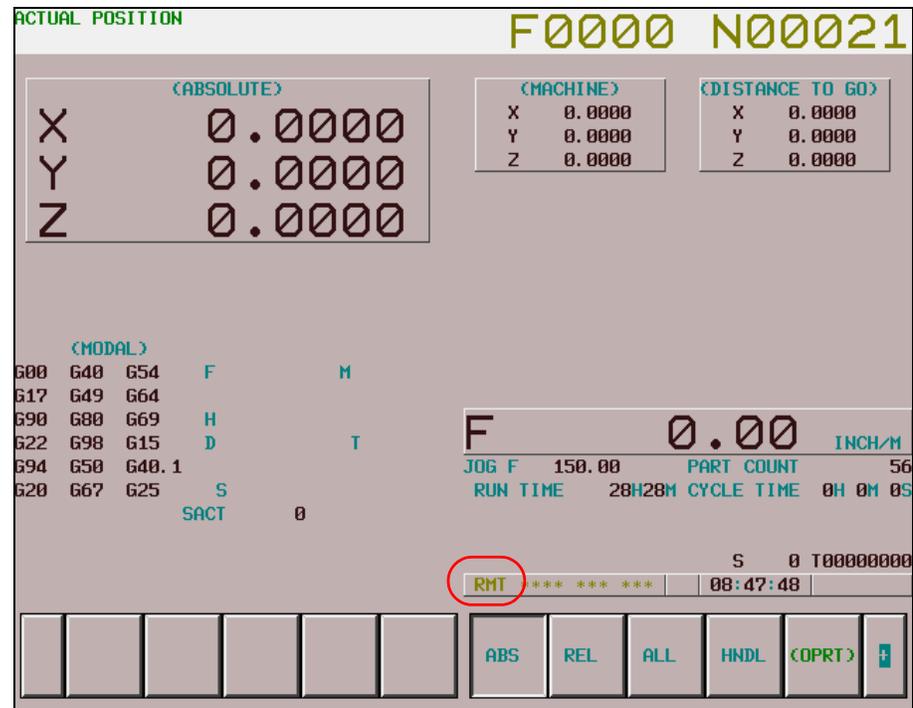


Figure 3-4: Remote Mode

Circled RMT on the Figure 3-4: indicates that operation is in Remote mode.

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 pushbutton on the machine operator's panel to set REMOTE mode, then press CYCLE START pushbutton. 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 pushbutton on the MDI panel.
4. Push [+] softkey twice.
5. When [DNC-CD] softkey is pressed, the following screen is displayed.

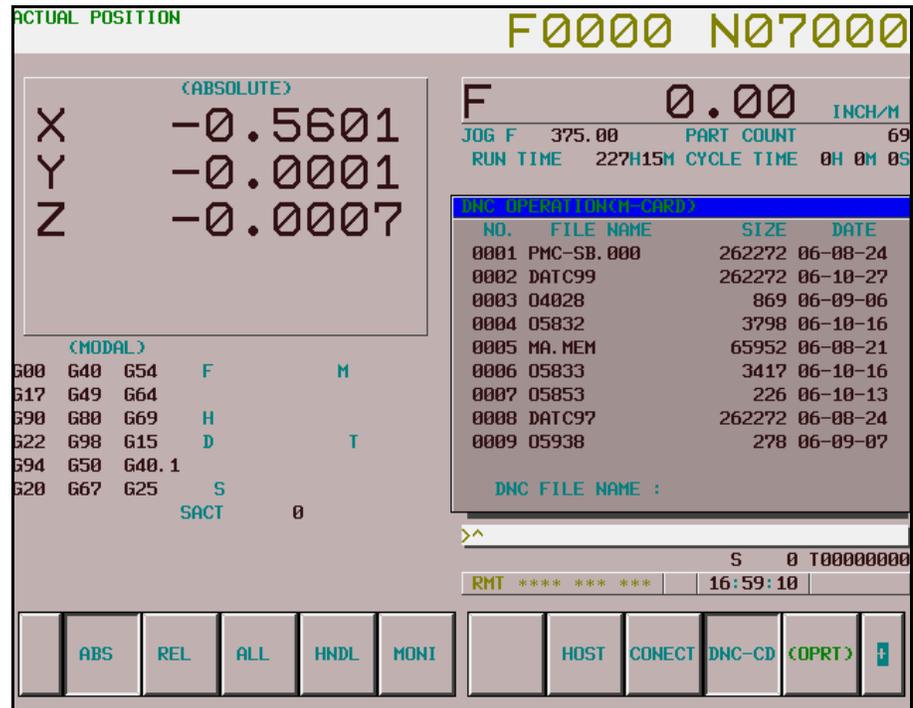


Figure 3-5: DNC-CD Softkey

The screen can be scrolled by page key. An arbitrary file number is input, and [F SRH] softkey is pressed (Figure 3-6:). Then the arbitrary file name is displayed at the top of DNC operation (memory card) screen.

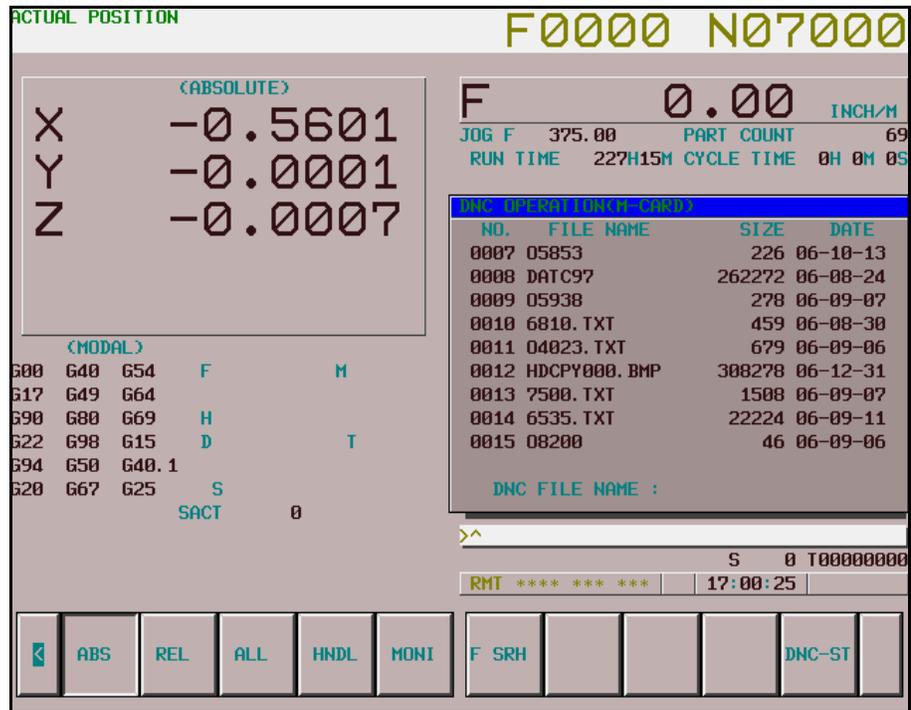


Figure 3-6: F-SRH Softkey

- Input the file number (for example No. 7) which is going to be executed.

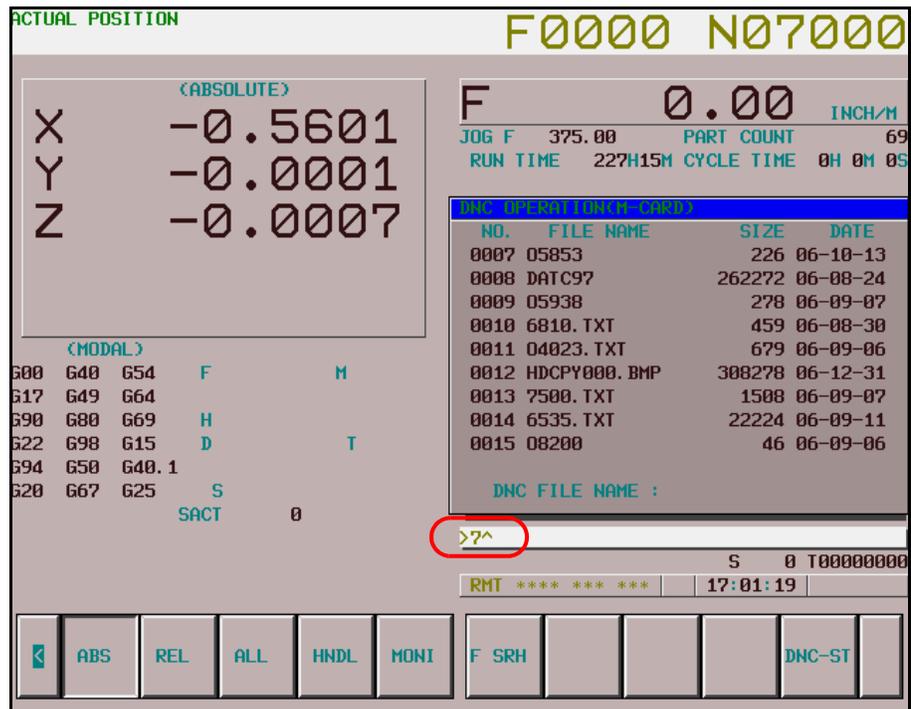


Figure 3-7: File No. to be executed

- Press the [DNC-ST] softkey (Figure 3-7:), the file name will be displayed in the right side of DNC FILE NAME:

For example; *DNC FILE NAME:O0853*

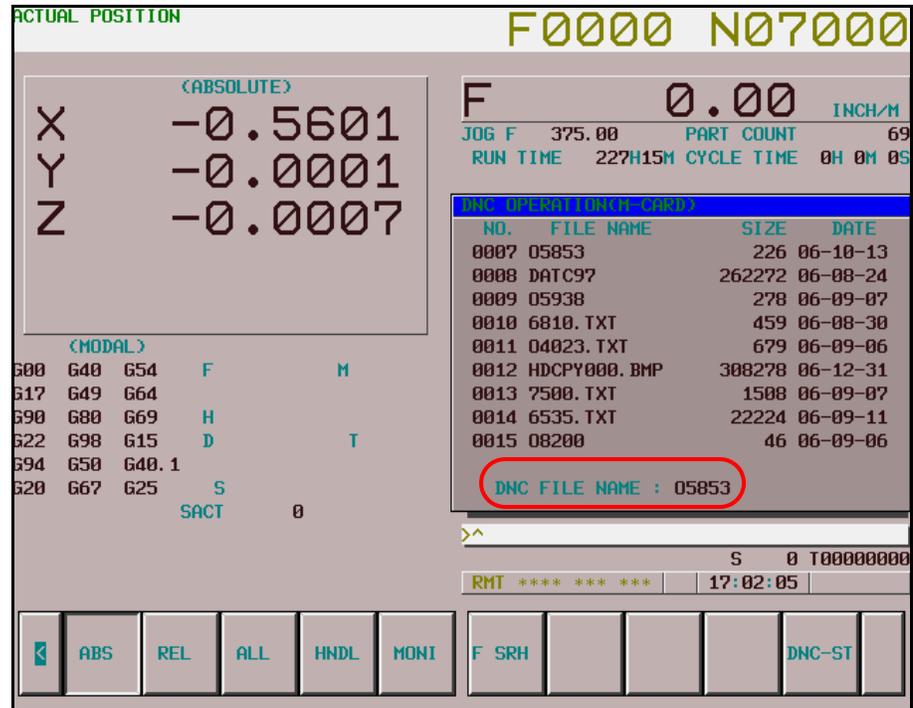


Figure 3-8: Executed File

- Press the CYCLE START pushbutton to execute the program selected.

3.1.5 REF RETURN MODE



REFERENCE RETURN mode gives an opportunity to return all the axes to the machine zero position manually. If Ref. Return and axes are selected machine references all the selected axis to its Cold Start position (Machine Home position).

It is recommended to reference all the axis every time machine is powered ON.

The screen below is shown in Ref. Return mode.

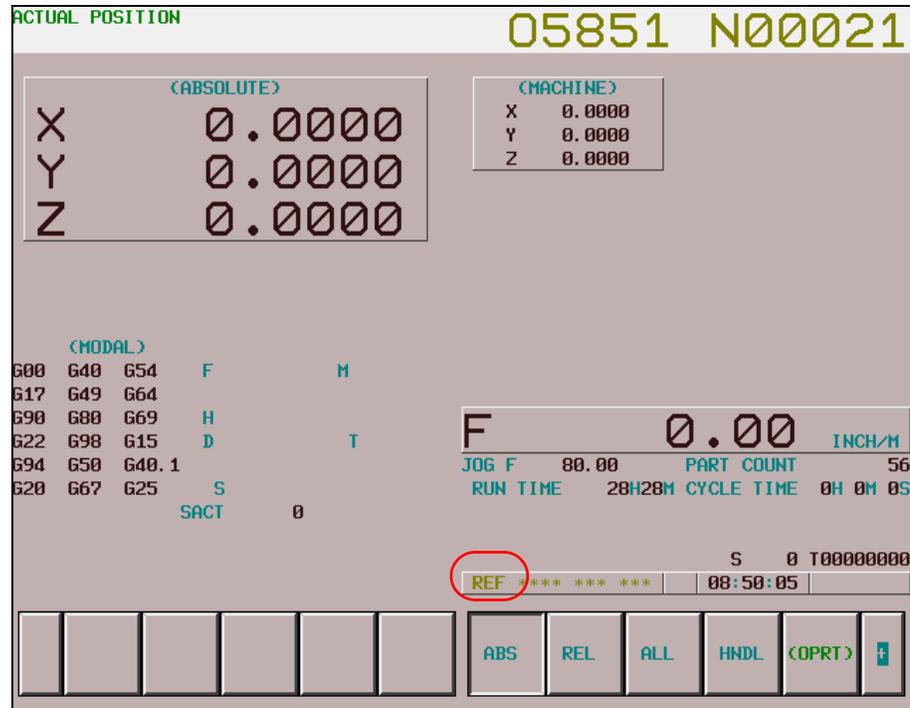


Figure 3-9: Ref. Return Mode

Circled REF on the Figure 3-9: indicates that operation is in Reference Return mode.

Reference Zero:

Manual reference position return is to move the tool to the reference position using switches and pushbuttons 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±, A± pushbutton.

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, HANDLE or AUTO, MDI, and EDIT mode keys to exit the Reference mode.

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

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 pushbutton.
- Press X±/Y±/Z±/ A± pushbutton on the operator panel A and wait until the reference moving is finished.
- Press JOG pushbutton.
- Press the REF RETURN pushbutton.
- Press X±/Y±/Z±/ A± pushbutton on the operator panel A and X±/Y±/Z±/ A± axes will move to the machine reference position.

3.1.6 JOG MODE



In the JOG mode, pressing the direction pushbutton on the operator's panel moves the tool along with the selected axes in the selected direction.

The screen below is shown in Jog mode.

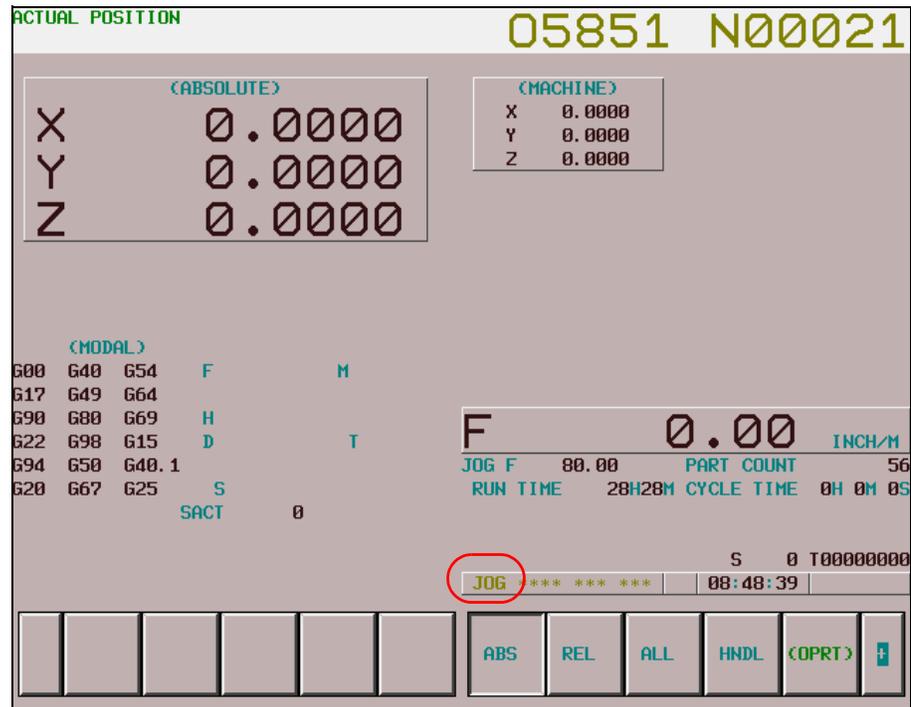


Figure 3-10: Jog Mode

Circled JOG on the Figure 3-10: indicates that operation is in Jog mode.

1. Select JOG mode.
2. Select one axis to be manually jogged by X±, Y±, Z±, or A± axes pushbuttons. Axis selection pushbutton does not have to be held down. LED will light indicating selected axis. Pressing any axis pushbutton will automatically de-select previously selected axis. It is implemented to prevent jogging of more than one axis simultaneously.

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

3. Pressing RAPID O'RIDE pushbutton during continuous jog move will increase axis jog rate to traverse speed (MAX 1000IPM for XYZ axis). Feed rate switch is also used to override traverse rates.

NOTE

If the feedrate is at 0%, no motion will occur when using the axis pushbutton.

3.1.7 HANDLE MODE



In the HANDLE mode, operator can move the axes using the Manual Pulse Generator (MPG) on the operator's panel.

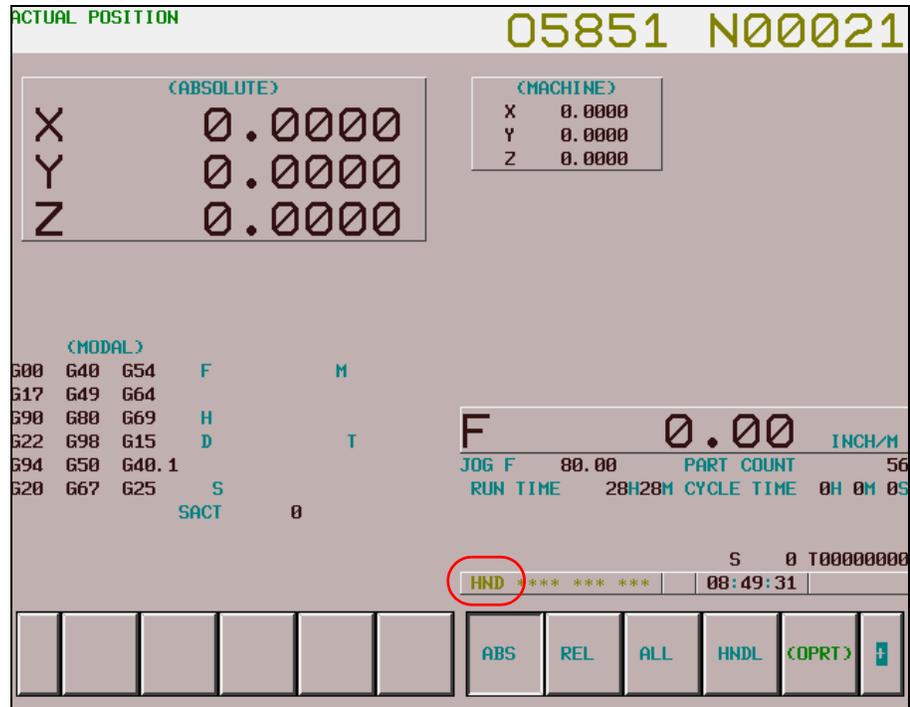


Figure 3-11: Handle Mode

Circled HND on the Figure 3-11: indicates that operation is in Handle mode.

By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

1. Select HANDLE x1, HANDLE x10 or HANDLE x100 mode.
2. Select one axis by X±, Y±, Z±, A± axes pushbuttons, as in JOG mode.
3. 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

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.

3.2 SAFETY FUNCTIONS

3.2.1 EMERGENCY STOP



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 X/Y/Z/A axes are stopped, and the other actions are interrupted. CNC will display EMG STOP message on the screen.

3.2.2 DOOR OVERRIDE



DOOR OVRD pushbutton 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 pushbutton is momentary. Momentary function requires operator to hold pushbutton while door override is required.

3.2.3 WORKLIGHT



Operator control of machine's work light(s). Pressing WORK LIGHT pushbutton toggles work light On/ Off.

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

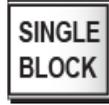
3.2.4 ALARM MESSAGE



ALARM RESET pushbutton resets the CNC and clears the PMC message.

3.3 TEST FUNCTION MODES

3.3.1 SINGLE BLOCK



SINGLE BLOCK allows an active part program program to be processed one block at a time. When this feature is active each block must be started using the CYCLE START pushbutton.

The single block pushbutton is enabled in MEM, REMOTE and MDI mode.

NOTE

It is permitted to use this pushbutton to interrupt an active cycle. For the affect of single block programming on canned cycles refer to GE Fanuc Operators Programming Manual supplied with the machine.

3.3.2 BLOCK DELETE



BLOCK DELETE when active all program blocks preceded by the BLOCK DELETE character (/) are skipped during a CNC cycle. This pushbutton is enabled only in MEM, REMOTE, and MDI mode.

3.3.3 OPTION STOP



OPTIONAL STOP feature is enabled by M01. When optional stop is active the CNC cycle will stop on reading an M01 code and can only be re-started by pressing the CYCLE START pushbutton. All existing modal program information is unaffected by this function. This pushbutton is enabled in MEM, REMOTE, and MDI mode only.

3.3.4 PROGRAM RESTART



This feature provides the facility for restarting a program at a chosen sequence block number following a program interruption. The application of Program Restart is particularly useful when there are a considerable number of data blocks between tool changes normally adopted as restart blocks. The feature may be used to restart the program at a specific point in a milling operation or at a chosen position in any drilling or boring canned cycle.

The Program Restart facility is enabled via a pushbutton located on the Operator Panel A.

The operating procedures which follow replace the text describing Program Restart in the Fanuc Operating Manual, publication No. B-63534 -02

1. AUTO CYCLE in progress
2. Press CYCLE STOP pushbutton
3. Press SPINDLE STOP / COOLANT OFF pushbutton
4. Press RESET pushbutton

5. Choose one of these procedures:

Table 3-1:

SPINDLE TOOL CORRECT FOR RESTART	TOOL TO BE CHANGED PRIOR TO RESTART
<p>Press EDIT pushbutton</p> <ul style="list-style-type: none"> - Edit the program (if necessary) - Press RESET pushbutton (Cursor returns to the top of the program. The screen displays the opening data blocks of the Program) <p>Press MDI pushbutton.</p> <ul style="list-style-type: none"> - Key-in the following restart data:- Sxxxx Spindle Speed * Mxx Spindle Start & Coolant On * Press EOB & INSERT pushbuttons Press CYCLE START pushbutton. - Spindle rotates (Coolant ON - if requested ON) <p>* Mandatory Input - see Caution below.</p>	<p>Press EDIT pushbutton</p> <ul style="list-style-type: none"> - Edit the program (if necessary) - Press RESET pushbutton (Cursor returns to the top of the program. The screen displays the opening data blocks of the Program) <p>-Select Spindle Tool for restart, i.e, Key-in Txx M6 Press EOB & INSERT pushbuttons Press CYCLE START pushbutton.</p> <ul style="list-style-type: none"> -Key-in the following restart data:- Sxxxx Spindle Speed* Mxx Spindle Start & Coolant On* Press EOB & INSERT pushbuttons Press CYCLE START pushbutton. -Spindle rotates (Coolant ON - if requested ON) <p>* Mandatory Input - see Caution below.</p>

CAUTION

Starting the spindle (and coolant, if required) is a mandatory input via MDI. Failure to heed this Caution will cause the program to restart without spindle rotation resulting in the possibility of damage to the machine, cutting tool and workpiece.

3.3.5 MC LOCK



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

3.3.6 DRY RUN



DRY RUN when active, inhibits coolant flow and the feedrate specified by the CNC program is ignored. The following notes are relevant:

- The FEED/RAPID/JOG OVERRIDE selector switch is used to modify the DRY RUN feedrate from 0.1m/min. to 9.55 m/min.
- Rapid movements are conducted as normal, and can also be overridden by the FEED/RAPID/JOG OVERRIDE selector switch.
- This pushbutton can be enabled in both MEM and MDI modes by pressing FUNC+DRY RUN pushbuttons.
- The LED is ON when DRY RUN is active.
- Dry run can only be selected when the machine is not in cycle.

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 pushbutton 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 pushbuttons and turn it Off by pressing DRY RUN pushbutton. 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 1000 IPM when the feed rate is overridden by 100%. The rapid traverse switch can also be used for changing the feed rate.

3.4 SETUP MODES

3.4.1 FUNCTION KEY



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

Operator Panel functions currently requiring FUNC pushbutton are:



3.5 TOOL OPERATION

3.5.1 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 clock-wise (ATC) when looking from the top of the carousel. Rotation is clock-wise (DATC) when looking from the left side of the machine.

Jog Mode is required. Operator may hold down pushbutton for indexing of multiple tool pots, or press momentarily to index one tool.

3.5.2 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 counter clock-wise (ATC) when looking from the top of the carousel. Rotation is counter clock-wise (DATC) when looking from the left side of the machine.

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

3.5.3 TOOL REL



Tool Release (Draw Bar Open)

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

FUNC pushbutton must be pressed simultaneously with TOOL REL pushbutton.

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.

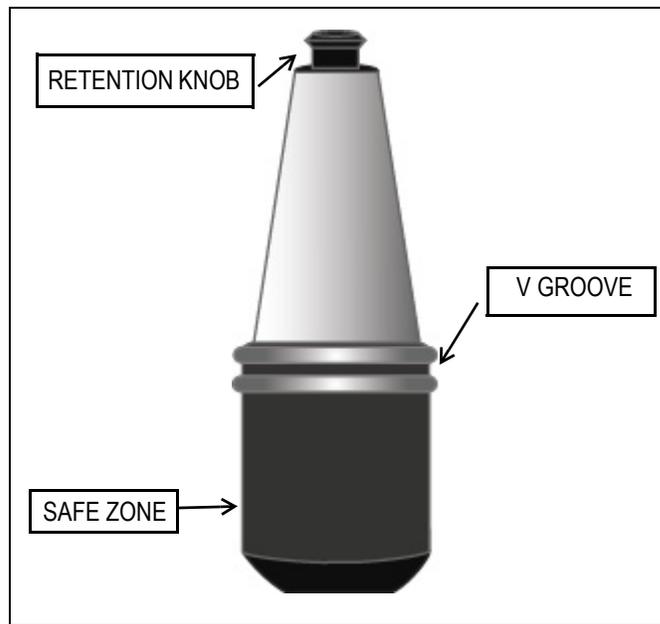


Figure 3-12: Tool Holder

2. Press the FUNC + TOOL REL pushbuttons. Keep the FUNC + TOOL REL pushbuttons 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 FUNC + TOOL REL pushbuttons, not before. The keys on the nose of the spindle must fit into the key-ways on the tool holder flange.

Release the TOOL REL pushbutton 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.

3.5.4 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) of magazine for next tool change.

Rotation of carousel will not inhibit continued execution of part program 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-Word after M06 will command positioning of carousel, followed by immediate 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.

3.5.5 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.

Legal Tool command:

M06 T10: Proper Command Syntax

M06: Proper Command Syntax(will change tool to the previous T number).

T05 M06: Proper Command Syntax

NEXT TOOL:

Next Tool pushbutton is used to load a tool which is next to the current aligned position from the tool drum into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.

Press "JOG" button.

Pressing "FUNC" + "NEXT TOOL" will load the next pocket tool into the spindle. It will return the tool in the spindle to the tool drum first, then increment the tool drum 1 pocket and load the tool into the spindle.

Operator can see the tool number changing from tool management screen.

LOAD TOOL

Load Tool pushbutton is used to load a tool from the tool drum into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.

Press "JOG" button.

Press "DRUM FWD" or "DRUM REV" button to move the carousel to the pocket has the required tool.

Pressing "FUNC" + "LOAD TOOL" will load the tool aligned with the spindle into the spindle. It will return the tool of spindle to the carousel first, move the carousel to the tool which pocket was aligned with the spindle, then load that tool into the spindle.

If the tool number at the current position is in the spindle, it will do nothing.

Operator can see the tool number changing from tool management screen.

ATC Maintain:

If FX machine DATC arm stuck some place or spindle position, One should be able to use command to recover the ATC to home position.

Procedure:

1. Move the carousel back to home position:

Press "RESET" and " RESET ALARM" P.B, button to clear all alarm.

Press "MDI" key on the operator panel.

Press "PROG" key on the MDI panel.

Press "MDI" soft key.

Type "M186"

Press "EOB" key on the MDI panel.

Press "INSERT" key on the MDI panel.

Close the front door.

Press " CYCLE START" on the operator panel.

Done.

Caution: one can set the K6.1=1 to override cycle start reference Inhibit protection. One should change the K6.1=0 as soon as one recovered the ATC.

2. Move carousel to spindle position. Move the Z axis to the cold start position.

Press "RESET" and " RESET ALARM" P.B, button to clear all alarm.

Press "MDI" key on the operator panel.

Press "PROG" key on the MDI panel.

Press "MDI" soft key.

Type "M190"

Press "EOB" key on the MDI panel.

Press "INSERT" key on the MDI panel.

Close the front door.

Press " CYCLE START" on the operator panel.

Type "M185"

Press "EOB" key on the MDI panel.

Press "INSERT" key on the MDI panel.

Close the front door.

Press " CYCLE START" on the operator panel.

Done.

3. Release the draw bar manually.

Move the draw bar manually. Press "FUNC" + "TOOL REL" soft and hold one second will open the draw bar. The ball bar will close as soon as one releases these two buttons.

4. ATC signal diagnosis screen:

One can go to ATC diagnosis screen to see all the I/O signal status. That is a good tool to check the electrical problem of ATC.

4.1. Press "JOG" key on the operator panel.

4.2. Press " M/C SETUP" key on the operator panel.

MACHINE SET-UP

MACHINE SETTING ENABLED

(1) USER PREFERENCE	(2) AXES SET UP	(3) AUTO TOOL CHANGER	(4) TOOL LOAD MONITOR
(5)	(6)	(7)	(8)

05100% L 0%

HND **** * * * * * 09:09:25

	(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)			
--	-----	-----	-----	-----	-----	-----	-----	-----	--	--	--

4.3. Press "3" soft key and the following screen come out.

MACHINE SET-UP

MACHINE SETTING ENABLED
AUTO TOOL CHANGER

TOOL CHANGE TYPE ATC

MAGAZINE POSITION 21

COUNT SW 1

LEFT SW 1

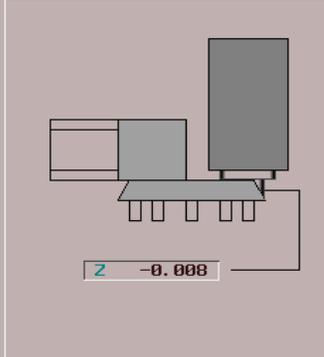
RIGHT SW 0



DRAWB SOL 0

UCLAMP SW 0

ORIENT IN 0



Z -0.008

05100% L 0%

MDI **** * * * * * ALM 17:34:23

										RETURN	
--	--	--	--	--	--	--	--	--	--	--------	--

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.

Legal Tool command:

M06 T10: Proper Command Syntax

M06: Proper Command Syntax(will change tool to the previous T number).

T05 M06: Proper Command Syntax

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

M06 T4 Legal. Tool Drum will position during tool change.

T05M06: Legal, tool Drum will position during tool change.

.

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.

1.1 Operation

1.1.1 Initialize the tool magazine and tool management.

- a. Execute M6T1 in the MDI mode.
- b. Move to the tool pocket to No.1.

"Press "REF RETURN" button on the operator panel.

"Press "FUNC" + "DRUM FWD"

"Turret will move the pocket to No.1 position automatically.

c. Enter the Tool management screen.

Press "EDIT" key on the operator panel.

Press "TOOL LENGTH" key on the operator panel.

Press "TOOL MAG "soft key.

The tool management screen will comes out like the following screen.

"POT" column displays the pocket number of magazine.

"TOOL#" column displays the tool number of magazine.

d. Edit the tool magazine management table as following display.

d.1 Press the "EDIT" soft key in the tool management screen.

d.2 One can input the tool number for each pocket to match the physical tool

POT	TOOL#
0	
2	2
3	3
4	4
5	5
6	6
7	7
8	8
9	9
10	10
11	11
12	12
13	13
14	14
15	15
16	16
17	17
18	18
19	19
20	20

SPINDLE TOOL
1
NEXT TOOL
0
MAGAZINE POSITION
1

e. Cycle the CNC power.

1.1.2 Load the physical tool.

"Initialize the tool magazine.

"Execute "M6T1" in the MDI mode.

"Open the front door.

"Press "JOG" key on the operator panel.

"Press "FUNC" + "TOOL REL" button and hold and load the tool No.1.

"Close the door.

"Execute "M6T2" in the MDI mode.

"Repeat above steps to load all tools into magazine.

"Cycle the CNC power.

1.1.3 Call each tool in the MDI mode to make sure that physical tool matches the programming tool number.

1.1.4 Jog the magazine.

Press "JOG" key on the operator panel.

Press "DRUM FWD" or "DRUM REV" to jog the magazine manually.

The magazine will stop at the next pocket as soon one release the button.

1.1.5 "NEXT TOOL" function for DATC.

Press "JOG" key on the operator panel.

Press "FUNC" + "NEXT TOOL" same time.

CNC will call the macro program and swap the spindle tool with current pocket tool.

1.1.6 "LOAD TOOL" function for DATC.

Press "JOG" key on the operator panel.

Press "FUNC" + "LOAD TOOL" same time.

CNC will call the macro program and load the spindle tool to the empty spot of the pocket.

1.1.7 Return spindle tool to magazine.

Execute the M6T0 command in the MDI or Auto mode will return the spindle tool to magazine.

1.1.8 DATC tool changer time.

It takes 5.1s to change the tool in the following condition.

Condition:

- a. Z axis is at home position.
- b. Requesting tool is at current pocket.

FOR DATC: THE "NEXT TOOL" AND "LOAD TOOL" ARE DIFFERENCE.

1.1.5 "NEXT TOOL" function for DATC.

Press "JOG" key on the operator panel.

Press "FUNC" + "NEXT TOOL" same time.

CNC will call the macro program and swap the spindle tool with current pocket tool.

1.1.6 "LOAD TOOL" function for DATC.

Press "JOG" key on the operator panel.

Press "FUNC" + "LOAD TOOL" same time.

CNC will call the macro program and load the spindle tool to the empty spot of the pocket.

NEXT TOOL:

ATC:

Next Tool pushbutton is used to load a tool which is next to the current aligned position from the tool drum into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.

Press "JOG" button.

Pressing "FUNC" + "NEXT TOOL" will load the next pocket tool into the spindle. It will return the tool in the spindle to the tool drum first, then increment the tool drum 1 pocket and load the tool into the spindle.

Operator can see the tool number changing from tool management screen.

DATC:

Next Tool pushbutton is used to load current pocket tool into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.

Press "JOG" button.

Pressing "FUNC" + "NEXT TOOL" pushbuttons will swap the spindle tool with current pocket tool

Operator can see the tool number changing from tool management screen.

LOAD TOOL

ATC:

Load Tool pushbutton is used to load a tool from the tool drum into the spindle. It is only active when the operator door is closed and CNC is in the Jog mode.

Press "JOG" button.

Press "DRUM FWD" or "DRUM REV" button to move the carousel to the pocket has the required tool.

Pressing "FUNC" + "LOAD TOOL" will load the tool aligned with the spindle into the spindle. It will return the tool of spindle to the carousel first, move the carousel to the tool which pocket was aligned with the spindle, then load that tool into the spindle.

If the tool number at the current position is in the spindle, it will do nothing.

Operator can see the tool number changing from tool management screen.

DATC:

Load Tool pushbutton is used to load a tool from spindle into an empty spot of magazine. It is only active when the operator door is closed and CNC is in the Jog mode.

Press "JOG" button.

Pressing "FUNC" + "LOAD TOOL" will load a tool from spindle into an empty spot of magazine.

If the tool number at the current position is in the spindle, it will do nothing.

Operator can see the tool number changing from tool management screen.

DOOR OVRD is for standard machine.

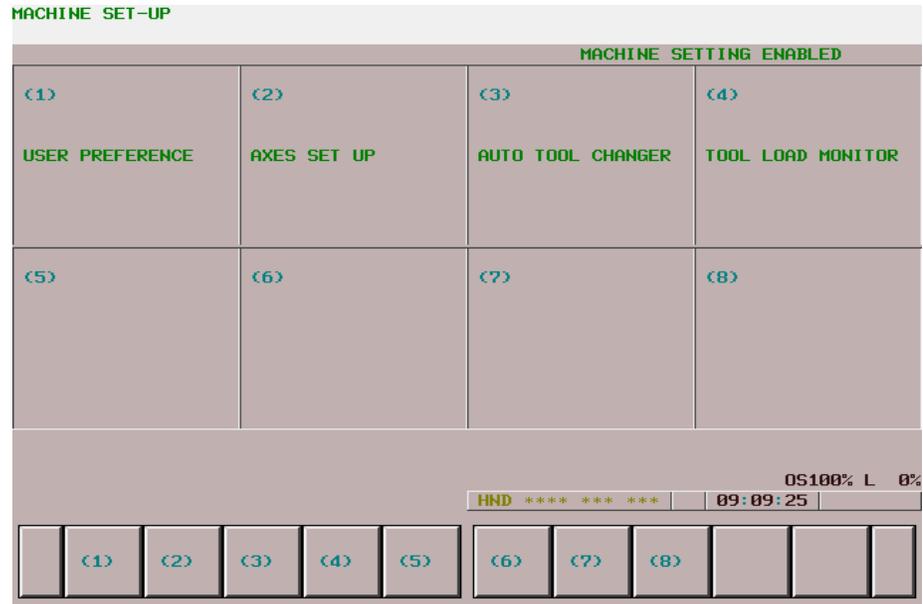
DOOR UNLOCK is for CE machine.

DATC Maintain:

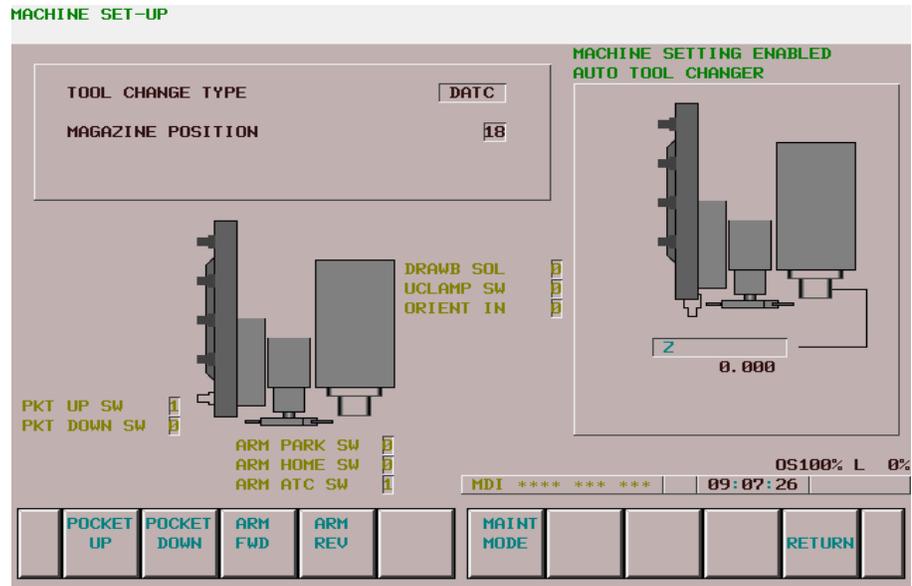
If FX machine DATC arm stuck some place or spindle position, One should be able to use the DATC maintain screen to recover the DATC and move arm and pocket back to the home position.

Procedure:

1. Press "JOG" key on the operator panel.
2. Press " M/C SETUP" key on the operator panel.

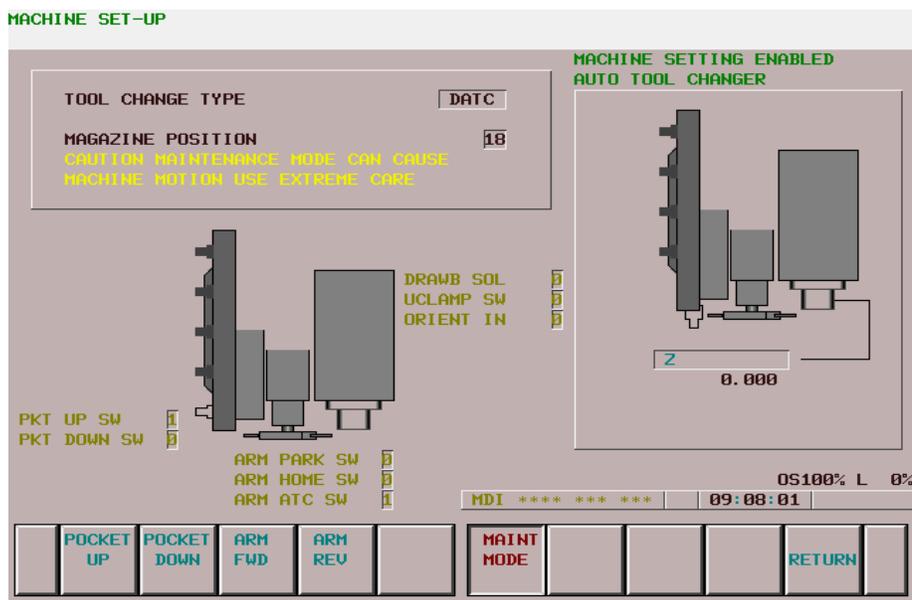


3. Press "3" soft key and the following screen comes out.



4. Press " MAINT MODE" soft key to enter the DATC maintain mode and the "MAINT MODE" soft key becomes red and the warning message comes out on the screen.

Press " MAINT MODE" soft key again will release this mode.



5. Move arm/pocket/ draw bar manually.

5.1 Move the ARM manually

Press "ARM FWD" soft key to move the arm at forward direction.

Press "ARM REV" soft key to move the arm at reverse direction.

Caution: Pressing "ARM FWD" or "ARM REV" soft key do not release the draw bar automatically. Please be carefully to move the arm in the right direction to avoid to force the tool.

5.2 Move the Pocket manually

Press "POCKET UP" soft key to move the pocket up.

Press "POCKET DOWN" soft key to move the pocket down.

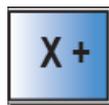
Caution: Before pressing "POCKET UP" or "POCKET DOWN" soft key, please make sure that arm have already moved to the safe position. Otherwise the collision might happen when one presses these two soft key.

5.3 Move the draw bar manually

Press "FUNC" + "TOOL REL" soft and hold one second will open the draw bar. The ball bar will close as soon as one releases these two buttons.

3.6 AXIS DIRECTION SELECTION

3.6.1 X+ AXIS



In the Jog mode X+ axis pushbutton is used for continuous axis jogging in the X+ direction.

It is also used in the Handle mode for HANDWHEEL X AXIS selection, in which case both the X+ and the X- LEDs are ON.

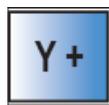
3.6.2 X- AXIS



In the Jog mode X- axis pushbutton is used for continuous axis jogging in the X- direction.

It is also used in the Handle mode for HANDWHEEL X AXIS selection, in which case both the X+ and the X- LEDs are ON.

3.6.3 Y+ AXIS



In the Jog mode Y+ axis pushbutton is used for continuous axis jogging in the Y+ direction.

It is also used in the Handle mode for HANDWHEEL Y AXIS selection, in which case both the Y+ and the Y- LED's are ON.

3.6.4 Y- AXIS



In the Jog mode Y- axis pushbutton is used for continuous axis jogging in the Y- direction.

It is also used in the Handle mode for HANDWHEEL Y AXIS selection, in which case both the Y+ and the Y- LEDs are ON.

3.6.5 Z+ AXIS



In the Jog mode Z+ axis pushbutton is used for continuous axis jogging in the Z+ direction.

It is also used in the Handle mode for HANDWHEEL Z AXIS selection in which case both the Z+ and the Z- LEDs are ON.

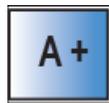
3.6.6 Z- AXIS



In the Jog mode Z- axis pushbutton is used for continuous axis jogging in the Z- direction.

It is also used in the Handle mode for HANDWHEEL Z AXIS selection in which case both the Z+ and the Z- LEDs are ON.

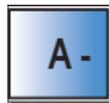
3.6.7 A+ AXIS



In the Jog mode A+ axis pushbutton is used for continuous axis jogging of the A axis in the A+ direction.

It is also used in the Handle mode for HANDWHEEL A AXIS selection in which case both the A+ and the A- LEDs are ON.

3.6.8 A- AXIS



In the Jog mode A- axis pushbutton is used for continuous axis jogging of the A axis in the A- direction.

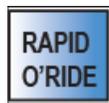
It is also used in the Handle mode for HANDWHEEL A AXIS selection where upon both A+ and A- pushbuttons are ON.

NOTE

The X-, Y-, Z+, and A+ LEDs are also ON in MEM or MDI mode when the corresponding machine axes are sent to their alignment position using G28 and also during the machine axes alignment sequence.

The Y+ (not Y-) LED will be ON, on machines aligning with the table at the front of the machine.

3.6.9 RAPID OVERRIDE



RAPID O'RIDE can be operated simultaneously with any power feed pushbutton to produce rapid axis jogging, when the operator door is closed.

If pressed in MEM or MDI modes (LED on) RAPID O'RIDE will reduce rapid rates by 50% during program execution. The rapid rate may be further reduced if it is used in conjunction with the FEED SELECTOR switch.

3.7 SPINDLE OPERATION

3.7.1 SPINDLE STOP



SPDL STOP causes an immediate spindle stop and is active at all times.

If the machine is in cycle, a feed hold is also generated.

NOTE

The LED in this pushbutton is ON if the spindle is stopped during an AUTO CYCLE interrupt condition, to act as a reminder to restart the spindle before the cycle can be resumed.

3.7.2 SPINDLE CLOCKWISE



Manual Spindle Start Clock-Wise & Spindle Jog CW

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

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

3.7.3 SPINDLE COUNTER-CLOCKWISE



Manual Spindle Start Counter Clock-Wise & Spindle Jog CCW

FUNC pushbutton 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

3.7.4 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 pushbuttons are held. Spindle will stop when CW/CCW pushbutton is released.

3.7.5 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: *S7500; spindle rotation speed is 7500RPM.*

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 pushbuttons for spindle start. Care should also be used in CW/CCW rotation.

3.7.6 SPINDLE SPEED RANGE

The spindle speed of the Fadal VMC has different configurations.

For machines with spindle speed 8K, the spindle has a single range 30-7500 RPM.

For 15K spindle (optional):

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

3.7.7 SPINDLE OVERRIDE

Spindle override 50%-120%

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

3.7.8 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.

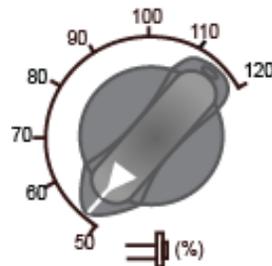
M19: Spindle special orientation command for canned cycles.

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 Back Boring canned cycles only. Using M19 for other purposes may cause machine crash.

3.7.9 SPINDLE SPEED OVERRIDE



Spindle Speed Override Selector switch is used to override the programmed speed within the range 50 to 120% of the programmed spindle speed.

The amount of override selected will affect all machining processes until it is changed by the operator.

3.7.10 SPINDLE CALIBRATION

M84: Spindle ratio calibration command:

M84 is only using for calibrating spindle ratio when the machine equipped with 10K HI/LO spindle or 15K HI/LO spindle. Machine comes with 8K 1:1 spindle does not require to calibrate The ratio.

Fadal have calibrated the spindle ratio on the floor during machine production. Use need to execute M84 to calibrate the spindle ratio if the orientation position is not accurate or spindle system parts was replaced or fixed (Example: spindle belt was replaced or spindle HI/LO parts were replaced or spindle motor was replace).

Spindle ratio calibration procedure:

Move the machine to cold start position.

Press "RESET" and " RESET ALARM" P.B, button to clear all alarm.

Press "MDI" key on the operator panel.

Press "PROG" key on the MDI panel.

Press "MDI" soft key.

Type "M84"

Press "EOB" key on the MDI panel.

Press "INSERT" key on the MDI panel.

Close the front door.

Press " CYCLE START" on the operator panel.

Wait 11 minetues until spindle stop without alarms.

Cycle the CNC power.

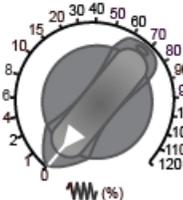
Test the spindle orientation using M190.

Test the spindle using M3Sxxxx code in the MDI panel.

Done.

3.8 OPERATION CONTROL

3.8.1 FEED/RAPID/JOG OVERRIDE



Feed/Rapid/Jog Override Selector Switch is used to override:

- Programmed Feedrate (scale 0-120)% in MEM and MDI modes. If an override could result in a feedrate greater than the maximum programmable feedrate, that feedrate is clamped to that maximum value.
- Rapid Traverse (scale 0-100)% in MEM/MDI/JOG modes.
- Jog rate (Scale 0-120)% in JOG mode only.

3.8.2 MANUAL PULSE GENERATOR

NOTE

Selecting 0% will cause a feedhold except during a Tapping Cycle.
Feed override is disabled whilst M48, G63, or Tapping Cycles are active.



When Handle mode is selected, clockwise (CW) rotation of the handwheel gives positive axis motion, counterclockwise (CCW) rotation will give negative axis motion. When in Handle mode, both axis selection and incremental distance moved, are made using the selection and increment pushbuttons on the Operator Panel A. The increments are:

PUSH-BUTTON	METRIC	INCH
HANDLE x 1	0.001mm	0.0001inch
HANDLE x 10	0.010mm	0.001inch
HANDLE x 100	0.100mm	0.010inch

3.8.3 HANDLE



HANDLEx1 selects Handle mode with 0.001mm (0.0001 inch) distance moved per handwheel graduation (increment) .



HANDLEx10 selects Handle mode with 0.010mm (0.001 inch) distance moved per handwheel graduation (increment).



HANDLEx100 selects Handle mode with 0.100mm (0.01 inch) distance moved per handwheel graduation (increment).

3.9 EXECUTION FUNCTION

3.9.1 CYCLE START



CYCLE START pushbutton causes the active CNC program to start, providing the following conditions are met:

- All axes, and the tool changer are aligned.
- No CNC or machine faults are active.
- MEM or MDI mode has been selected.

3.9.2 FEED HOLD

- The operator door is closed



FEED HOLD stops all feed motors and suspends the CNC cycle. Feed Hold is cancelled when re-starting the CNC cycle using the CYCLE START pushbutton.

NOTE

Any active program dwell will also be stopped by this pushbutton. If Feed Hold is pressed during a Tapping Cycle the CNC cycle will continue until the spindle has reversed and the Z axis has retracted

4.0

M/C SET UP

NOTE

The screens shown in this section are typical, and may vary slightly from machine to machine, depending on the specification.

To start machine setup procedure press M/C SETUP pushbutton on the Operator Panel A.

Screen displays the following setup windows:

- User Reference
- Axes Set Up
- Auto Tool Changer
- Tool Load Monitor

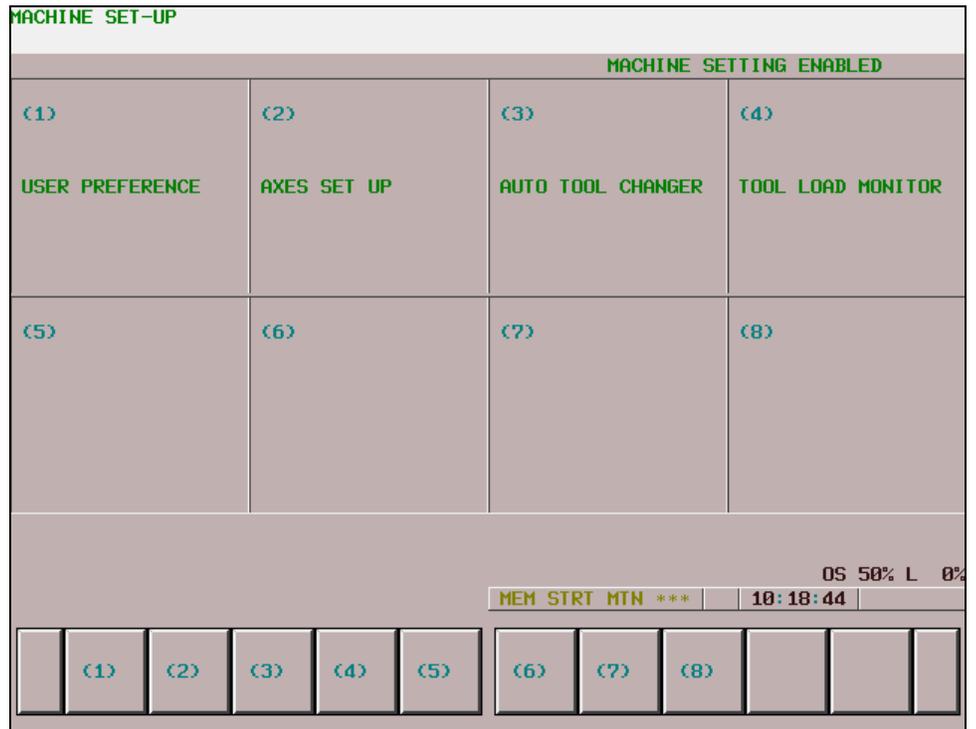


Figure 4-1: Machine Set-Up

4.1 USER PREFERENCE

Press (1) softkey (Figure 4-1:) to display User Reference screen.

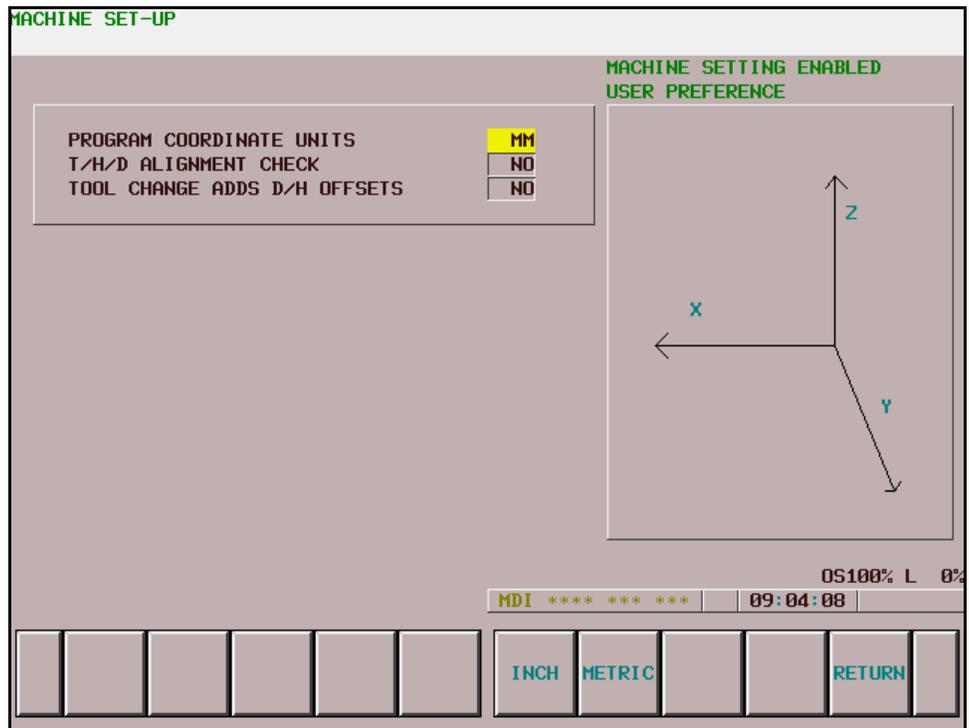


Figure 4-2: User Reference Screen

Machine Setup User Reference Screen displays the following settings:

- Program coordinate units
- T/H/D alignment check
- Tool change adds D/H offsets

RETURN soft key displays the main User Reference Screen.

4.1.1 PROGRAM COORDINATE UNITS

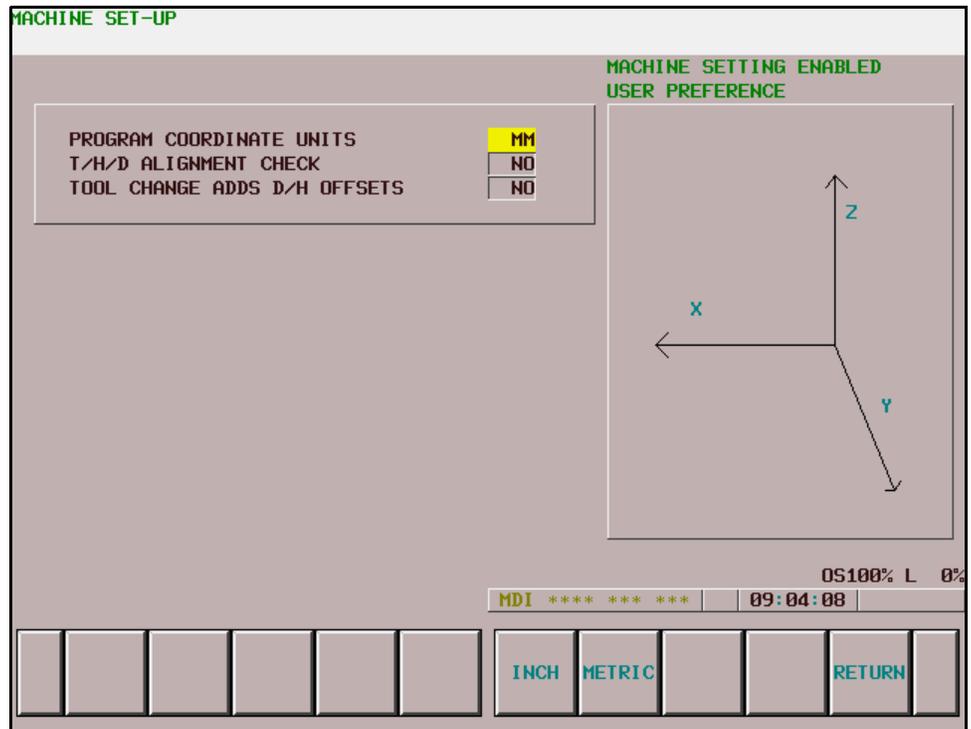


Figure 4-3: Program coordinate units

The default measurement units can be selected by pressing either the INCH or METRIC soft keys. Changing co-ordinate system automatically recalculates all tool and work offsets and re-establishes the absolute position in the new co-ordinate system.

NOTE

This may introduce a rounding error of 1 micron in the tool, work offset data. This is caused by the conversion in measurement system resolution (inch = 0.0001 and mm = 0.001).

4.1.2 T/H/D ALIGNMENT CHECK

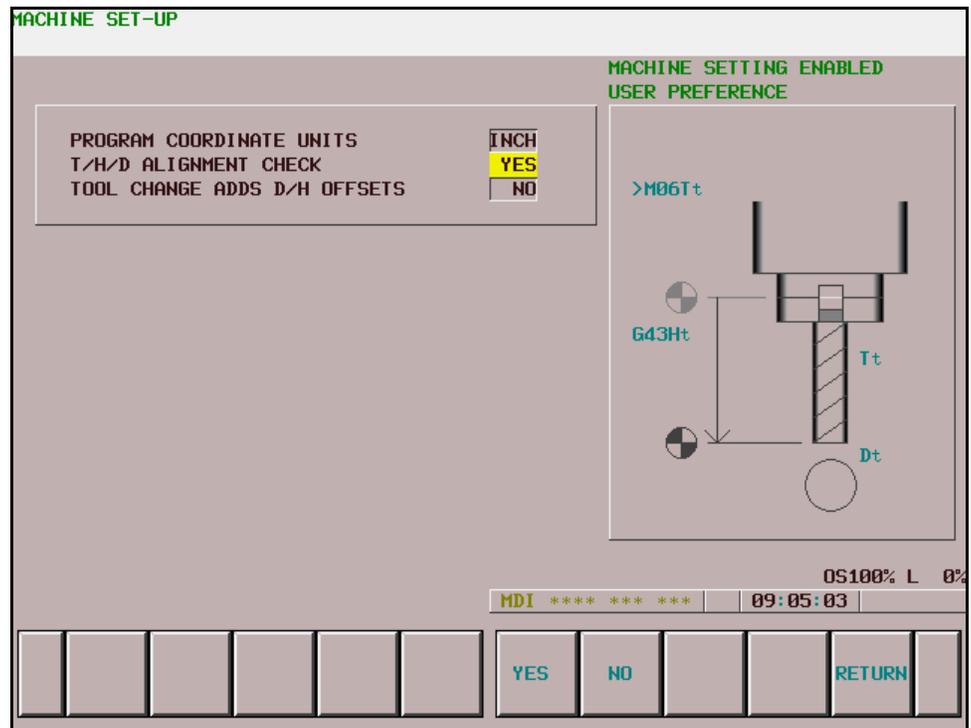


Figure 4-4: T/H/D Alignment Check

This function is used to check that a tool uses the corresponding tool length and diameter offsets.

Tool 1 can only have a length offset of H1 and a radius/diameter offset of D1.

If any other H or D offset is programmed with tool 1 in the spindle, a system error is generated and the cycle is stopped.

Tool 1 H2 D2 will generate a system error.

WARNING!

There is a possibility that Z-axis movement will occur if there is a large discrepancy between the active tool length and new programmed tool length. This movement can be as much as 13mm.

With this feature disabled, no T/H/D checking is performed, any tool can have any length (H) and any radius/diameter (D) offset assigned.

To enable or disable this feature use the cursor keys to move to the T/H/D Alignment Check and press either the YES or NO soft keys (Figure 4-4:).

4.1.3 TOOL CHANGE ADDS D/H OFFSETS

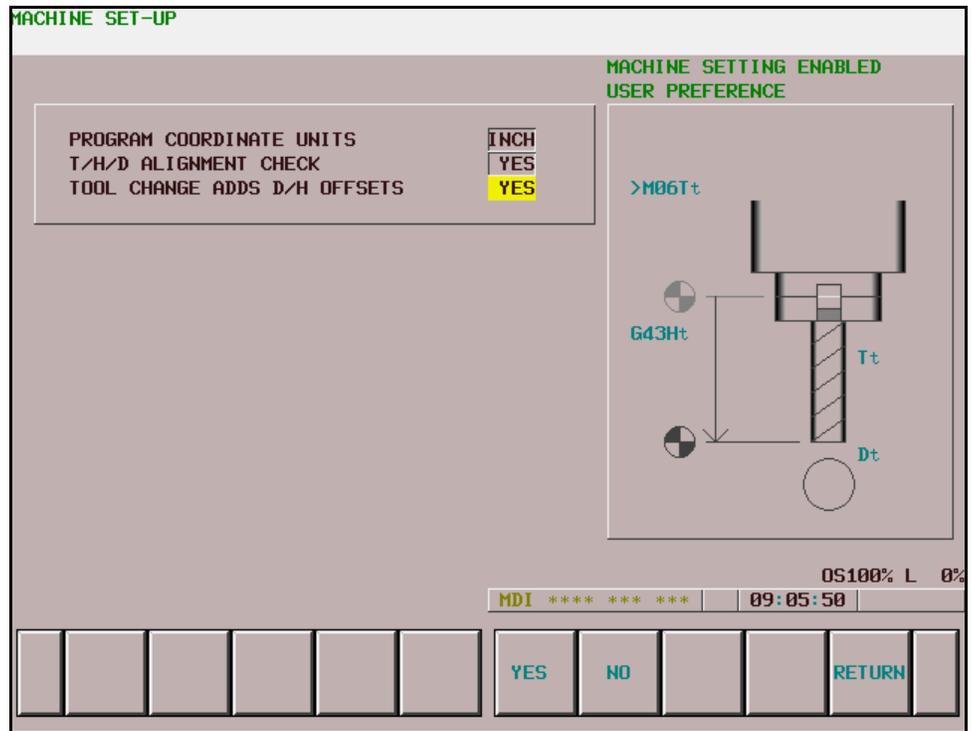


Figure 4-5: Tool Change Adds D/H Offsets

With this feature enabled; programming T1 M6 will load tool 1 into the spindle, the length offset H1 will be automatically applied without any axis movement and the Radius/Diameter offset D1 will be pre-set for use with cutter compensation.

The programmer can also program a different length offset H word at any time in the program using the normal G43 command, provided the feature "T/H/D Alignment Check" is disabled. This arrangement enables standard GE Fanuc programs to function as normal.

In addition, enabling the feature "Tool Change adds D and H Offset" automatically highlights the the 'Hxx' Length Geometry register associated numerically with the tool pocket number; i.e., if the tool from pocket 5 is loaded into the spindle it will be the Length Geometry register '005' which is automatically in highlight.

To enable or disable "Tool Change adds D and H Offsets" use the cursor keys to move to the "Tool Change adds D and H Offsets" feature in the menu and press either the YES or NO soft keys (Figure 4-5:).

4.2 AXES SET UP

Press (2) softkey (Figure 4-1:) to display Axes Set Up screen.

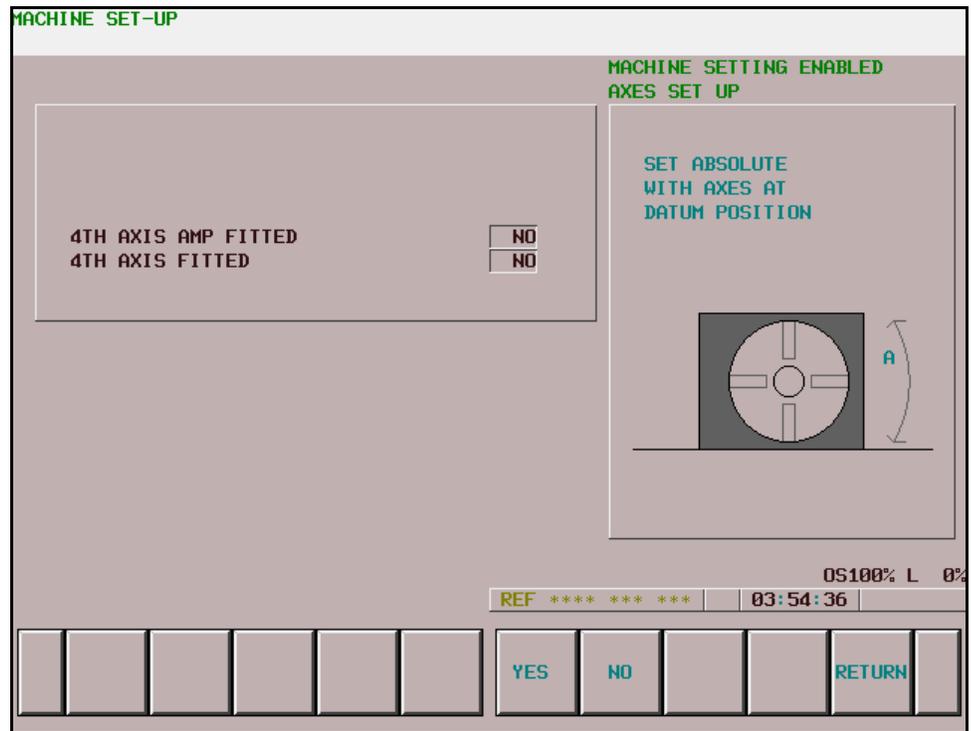


Figure 4-6: Axis Setup Screen

This screen allows the user to easily attach or detach a 4th and the commissioning / maintenance engineer to configure both the main machine axes and a 4th axis option.

NOTE

5th (B) axis is not applicable for GE Fanuc 0i control.

4.2.1 4TH (A) AXIS AMP FITTED

Enables the 4th axis hardware option.

To enable or disable this feature use the cursor keys to move to the 4th axis Amp Fitted and press either the YES or NO soft keys (Figure 4-6:)

4.2.2 4TH (A) AXIS FITTED

For this function to be enabled the 4th axis amp option must be installed.

If the user does not require the control to retain the 4th axis position after the machine has been shut down due to the nature of the work, the 4th axis is ready for future immediate use once the setting is changed to YES.

Setting of the 4th axis datum is accomplished using the Datum Set push button.

To enable or disable this feature use the cursor keys to move to the 4th AXIS FITTED and press either the YES or NO soft keys.

4.3 AUTO TOOL CHANGER

Press (3) softkey (Figure 4-1:) to display Auto Tool Changer screen.

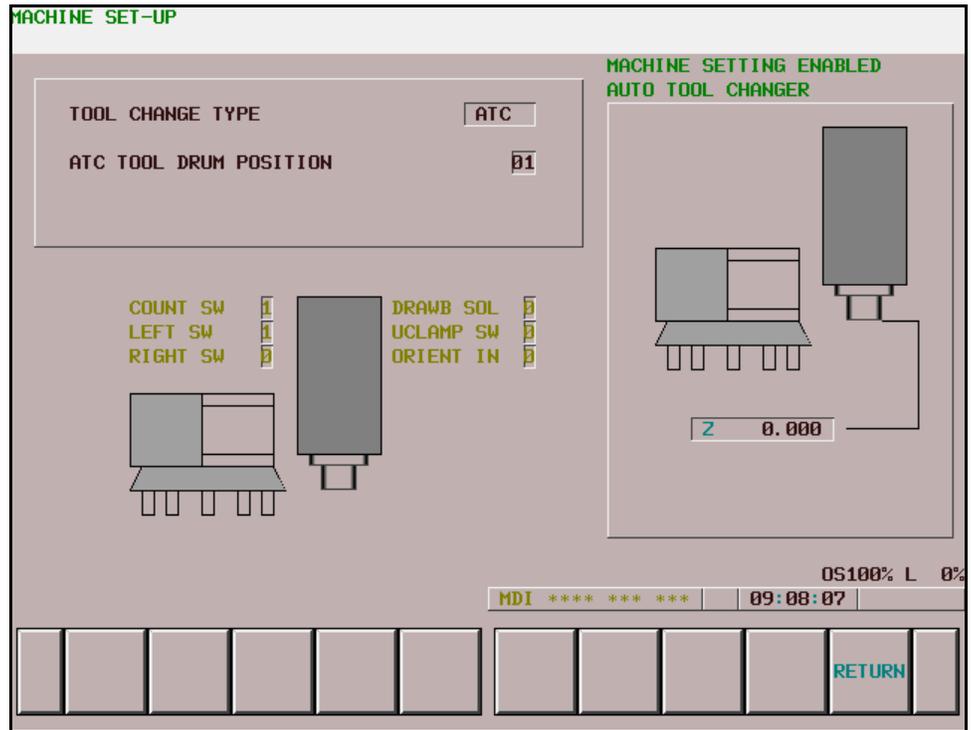


Figure 4-7: Auto Tool Changer Screen

4.3.1 TOOL CHANGE TYPE

The ATC is the 21 tool linear type (Figure 4-7:).

WARNING!

Re-configuring the spindle orient position, automatic tool change height and manual tool change height must be conducted by a Fadal engineer or authorized person. Erroneous configuration data may result in damage to the machine and/or personal injury.

4.4 TOOL LOAD MONITOR

Press (4) softkey (Figure 4-1:) to display Tool Load Monitor screen.

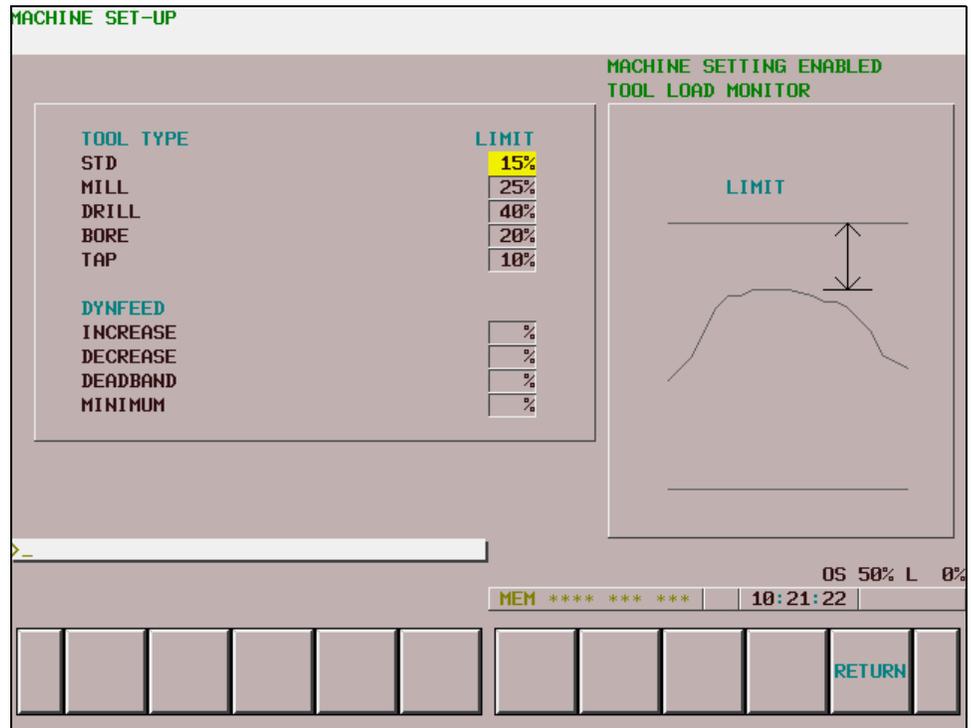


Figure 4-8: Tool Load Monitor Screen

The POWER SET push-button is used to display axis drive and spindle load information. Pressing this push button opens the Tool Load Monitor screen.

This function enables spindle tool protection by monitoring the spindle load. It is easy to set and use using the four rows of data displayed on the tool load monitor display. The control can be set to learn the values, using the teach facility, while running a program, negating any operator input or calculations. Once the values for each tool used in the program have been learned, the operator can run the program either with or without tool protection.

The Power Set facility enables dynamic feed rates and also spindle overload warning and machine stop.

This screen (Figure 4-8:) shows the percentage increase in spindle load before action from the control is implemented.

Five tool types are available:

- STD Default Value 15% Range 0 to 99 percent
- MILL Default Value 25% Range 0 to 99 percent

- DRILL Default Value 40% Range 0 to 99 percent
- BORE Default Value 20% Range 0 to 99 percent
- TAP Default Value 10% Range 0 to 99 percent

To modify the default values for each tool type use the following procedure:

1. Use the cursor key to highlight the value to be modified.
2. Using the number key pad type the value to be entered.
3. Press the INPUT push button.

4.4.1 POWER SET



The Dynamic Feedrate facility enables the operator to maximise the machines cutting power, effectively the feedrate is taught the optimum speed and feed for each cutting tool, for a specific job and material. Each cutting tool is set up and taught in turn, as follows:

- Press the POWER SET push-button.
- Use the cursor keys to navigate to the function column for the selected tool.
- Highlight MONITOR.

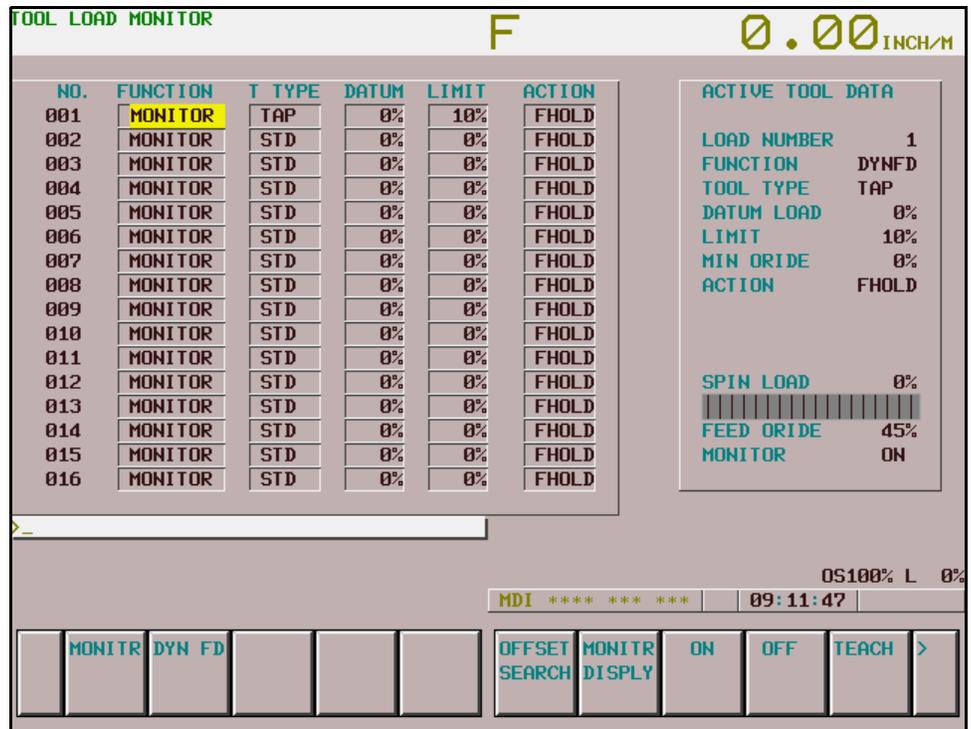


Figure 4-9: Function

Press the DYN FD softkey and the function for the selected tool becomes DYNFEED (Figure 4-10:).

Press the TEACH softkey.

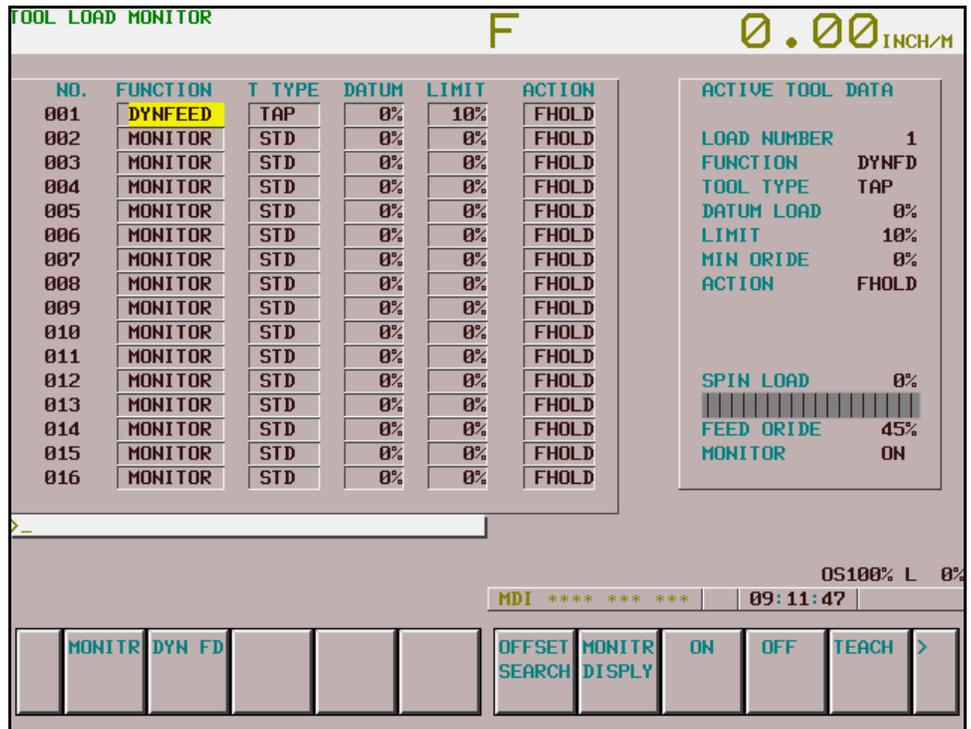


Figure 4-10: Function

4.4.2 TOOL TYPE

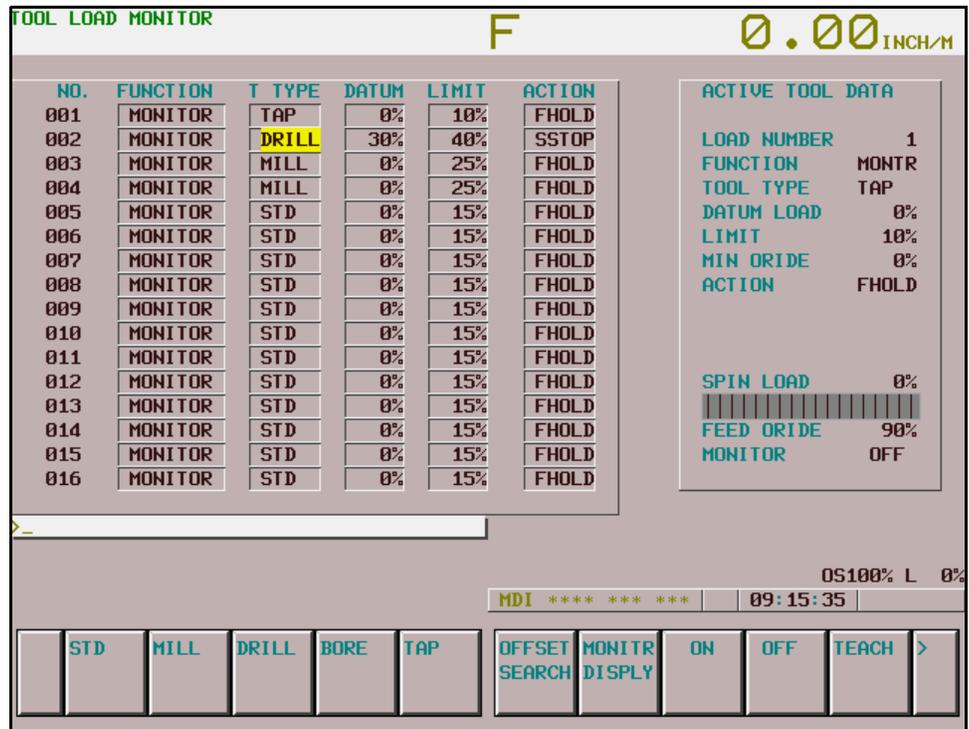


Figure 4-11: Tool Type

For ease of use, five tool types have been pre-defined in the Machine Configuration displays:

- STD Default Value 15% Increase
- MILL Default Value 25% Increase
- DRILL Default Value 40% Increase
- BORE Default Value 20% Increase
- TAP Default Value 10% Increase

To change the tool type for a tool number, use the cursor keys to highlight the tool required in the T TYPE column (Figure 4-11:) and press the relevant tool type soft key, the default value for the tool type is loaded into the LIMIT column for that tool.

The default values can be modified in the machine configuration displays.

4.4.3 DATUM

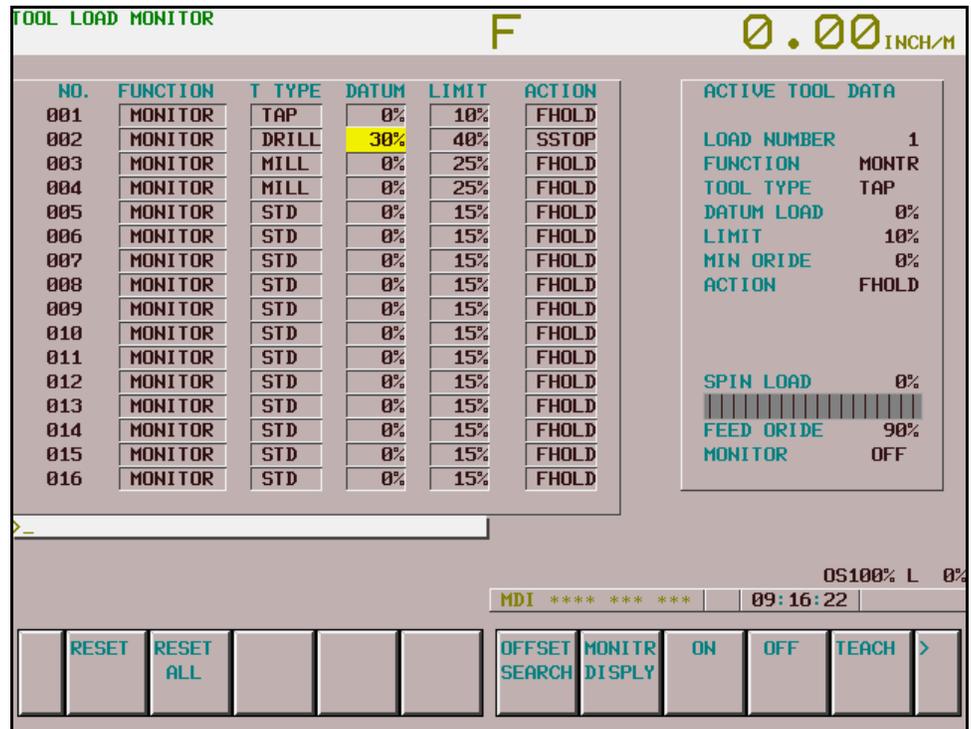


Figure 4-12: Datum

Datum is the maximum spindle load used by a tool during normal program execution, even if the tool is loaded several times within a program machining several features. The value recorded when using the teach mode can be overridden by the operator using the following procedure:

- Use the cursor keys to highlight the tool number required in the Datum column (Figure 4-12:).
- Using the numeric keypad type the value you require.
- Press the INPUT push button.

If the DATUM value is set to 0 (zero), the spindle load monitor is disabled and all available spindle power is used.

4.4.4 LIMIT

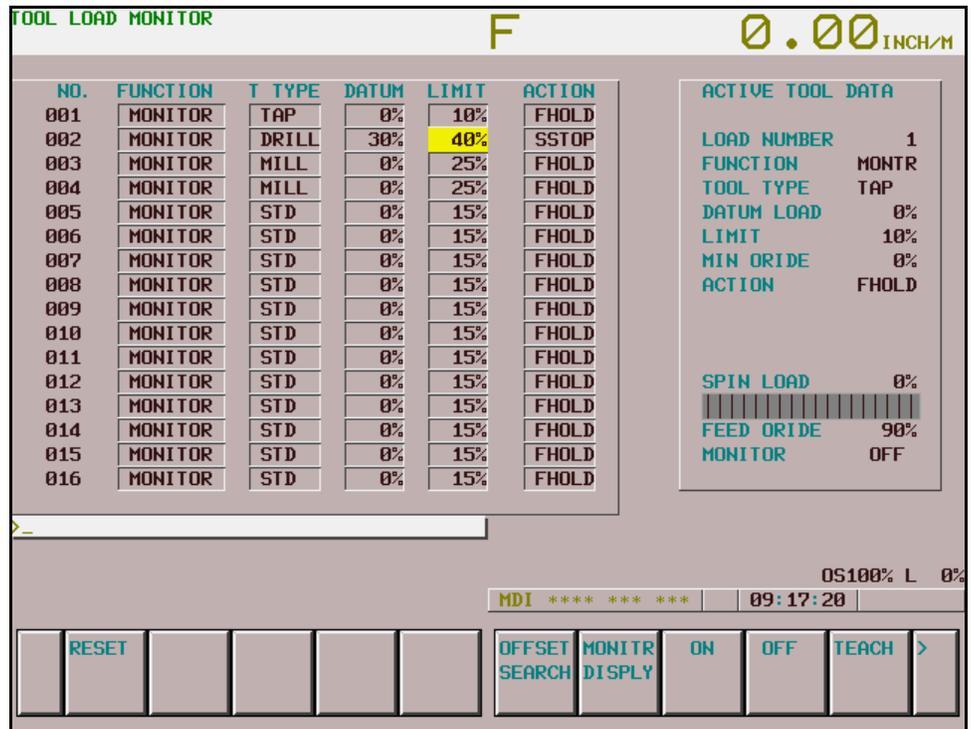


Figure 4-13: Limit

The limit is the incremental spindle load percent increase of the maximum spindle load measured for a tool. It is automatically set to the default tool type value; the operator can override the value with the following procedure.

- Use the cursor keys to highlight the tool number required in the Limit column (Figure 4-13:).
- Using the numeric keypad type the value required.
- Press the INPUT push button.

If the LIMIT value is set to 0 (zero) then the spindle load monitor is disabled & all available spindle power is used.

STD Default Value 15%

MILL Default Value 25%

DRILL Default Value 40%

BORE Default Value 20%

TAP Default Value 10%

Example 1:

The spindle load required for a milling cutter is 80%.

Using the default value for a milling cutter 25%

$$25\% \text{ of } 80\% = 20\%$$

$$80\% + 20\% = 100\%$$

The machine will function without any action from the control up to 100% spindle load.

Example 2:

The spindle load required for a drilling operation is 10%.

Using the default value for a drill 40%

$$40\% \text{ of } 10\% = 4\%$$

$$10\% + 4\% = 14\%$$

The machine will function without any action from the control up to 14% spindle load.

4.4.5 ACTION

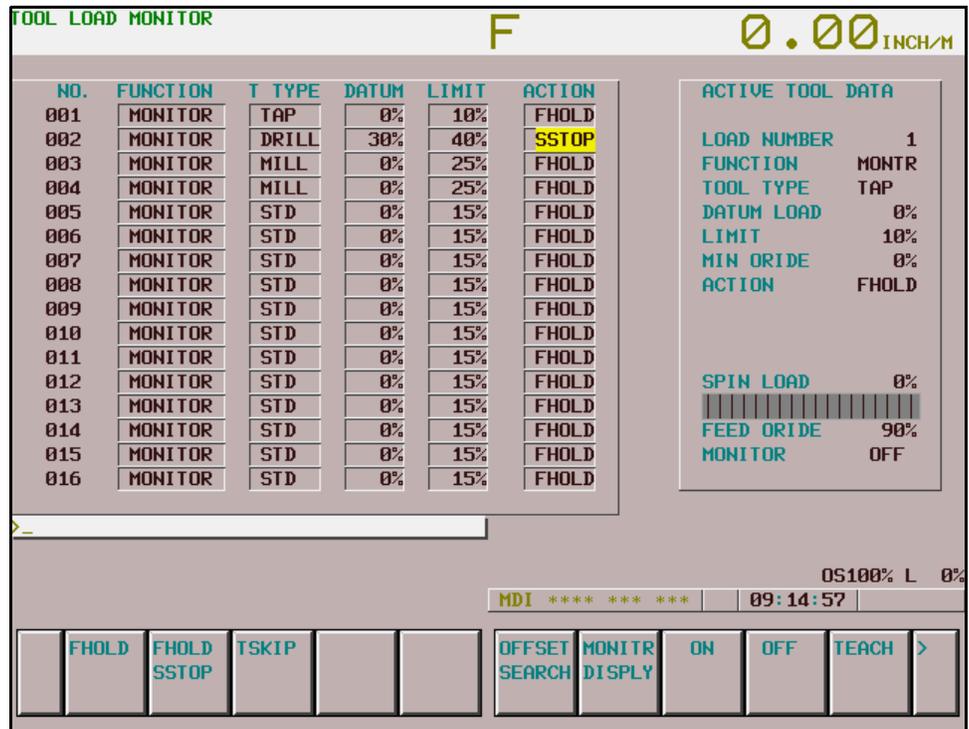


Figure 4-14: Action

There are two different actions that can be performed when reaching the tool spindle load limit:

FHOLD - Feed Hold. Applies feed-hold only - the spindle is allowed to continue running.

SSTOP - Feed Hold Spindle Stop. Applies feed-hold -the spindle stops 5 seconds after the spindle load has reduced to below the spindle load limit.

To change the action when the spindle load is exceeded use the following procedure:

Use the cursor keys to highlight the tool number required in the Action column (Figure 4-14:).

Press either the FHOLD or FHOLD SSTOP soft keys.

WARNING!

When Feed Hold Spindle Stop is selected, the spindle load increases above the action limit. The spindle will continue to rotate until the spindle load is reduced below the action limit, before the 5 second timer is started.

5.0

LENGTH SETTING

5.1 SETTING TOOL ORIGIN POINT

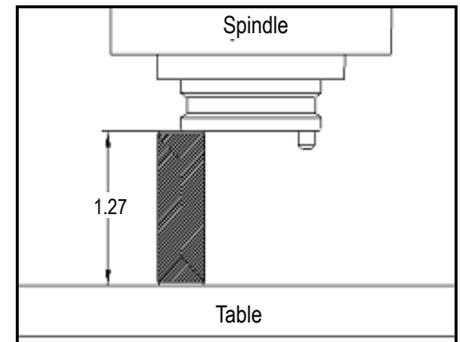
To simplify tool length setting several short-cut keys have been developed. The procedures here describe how to quickly set tool lengths.

It is advisable when setting the Tool Origin Point and Tool Lengths to set these points in the Z axis positive direction. This overcomes the possible problem of trapping the tool setting block or possible tool breakage.

A datum point must be set before tool length setting can be performed. It is recommended that a setting block of a known height is used to complete the following procedure. The minimum height of the setting block must be greater than the spindle nose clearance height between the table surface and maximum.

If there is a tool in the spindle that has been loaded from the tool drum, please open the door, press JOG pushbutton, and hold the tool in the spindle and press the FUNC and TOOL REL pushbuttons together and this will eject the tool from the spindle.

1. Press JOG mode pushbutton on the operator panel.
2. Using the axis direction pushbuttons position the spindle nose near the tool length setting block.
3. Using a combination of the HANDLE X 1, HANDLE X 10, HANDLE X 100, and the MPG Handwheel obtain a sliding fit between the spindle nose and the tool length setting block.
4. Remove the tool length setting block.
5. Press the LENGTH SET pushbutton - the following screen is displayed.



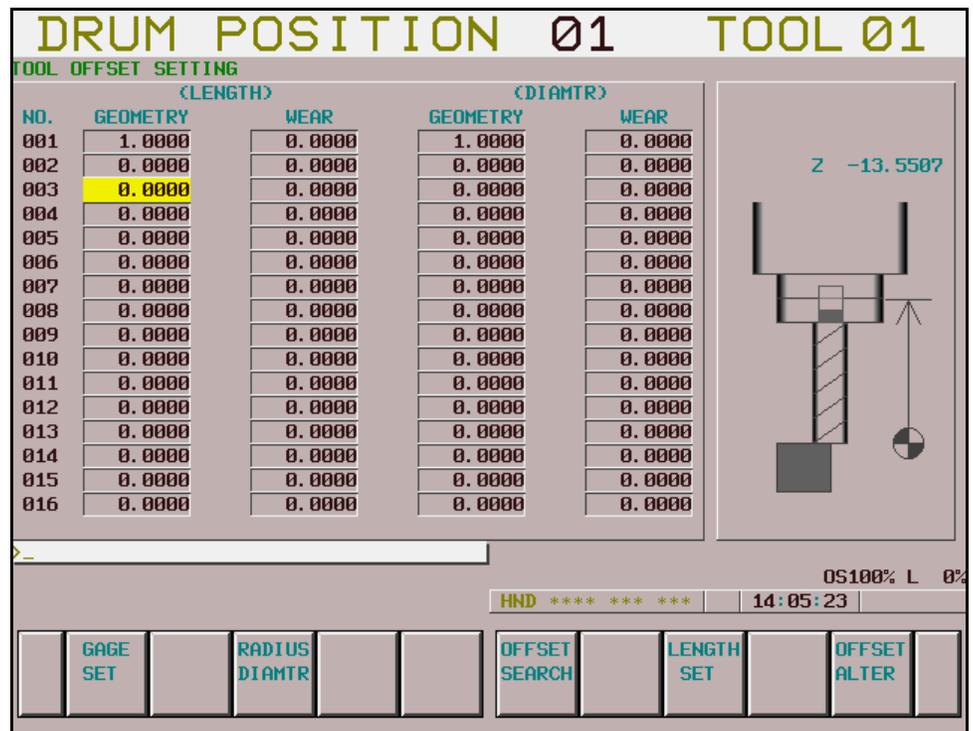


Figure 5-1:

6. Type the height of the tool height setting block .

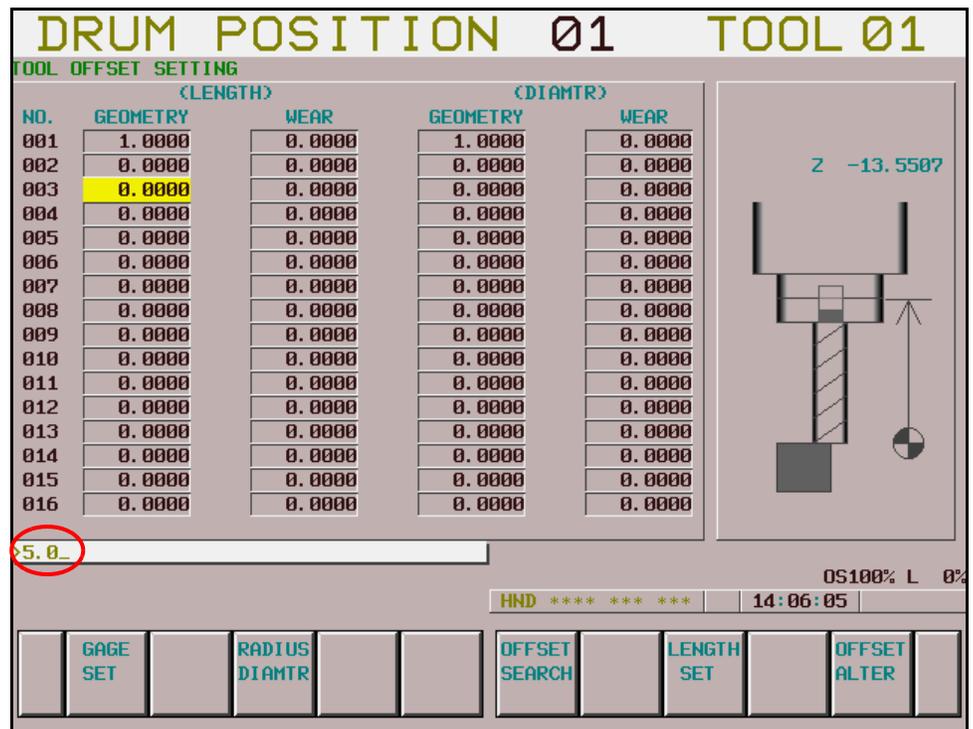


Figure 5-2:

- Press the GAGE SET soft key. The height tool setting block will be displayed on the screen and CNC will remember this value.

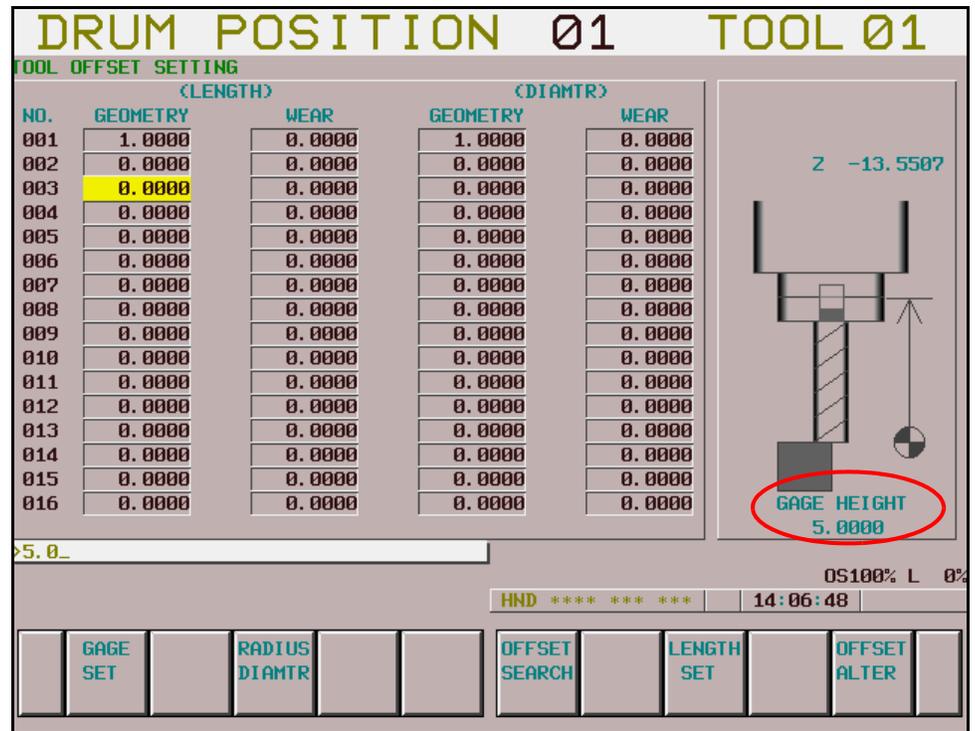


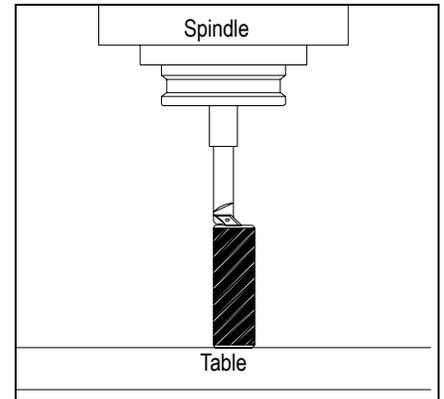
Figure 5-3:

- Press the JOG pushbutton.
- Press the Z+ pushbutton until the Z-axis is at the top of its stroke.

5.2 SETTING TOOL LENGTH

To reset a tool length after a tool breakage, follow the procedure below:

- Using the DRUM FWD or DRUM REV pushbutton index the drum to the tool you wish to load into the spindle e.g. pocket 1.
- Press the FUNC + LOAD TOOL pushbuttons.
- The tool from pocket 1 is loaded into the spindle.
- The data for the tool in the spindle are automatically highlighted.



5. Using the axis direction pushbuttons position the spindle nose near the origin point
6. Using a combination of the HANDLE X 1, HANDLE X 10, HANDLE X 100, and the MPG Handwheel obtain a sliding fit between the tool in the spindle and the tool setting block.
7. Remove the tool setting block.
8. Press the LENGTH SET softkey. The measured tool length is displayed.
9. For subsequent tools repeat steps 2 to 8.

To reset a tool length after a tool breakage, follow the above procedure.

5.3 TOOL DIAMETER/
RADIUS SETTING

1. Press the LENGTH SET softkey - the following screen is displayed.

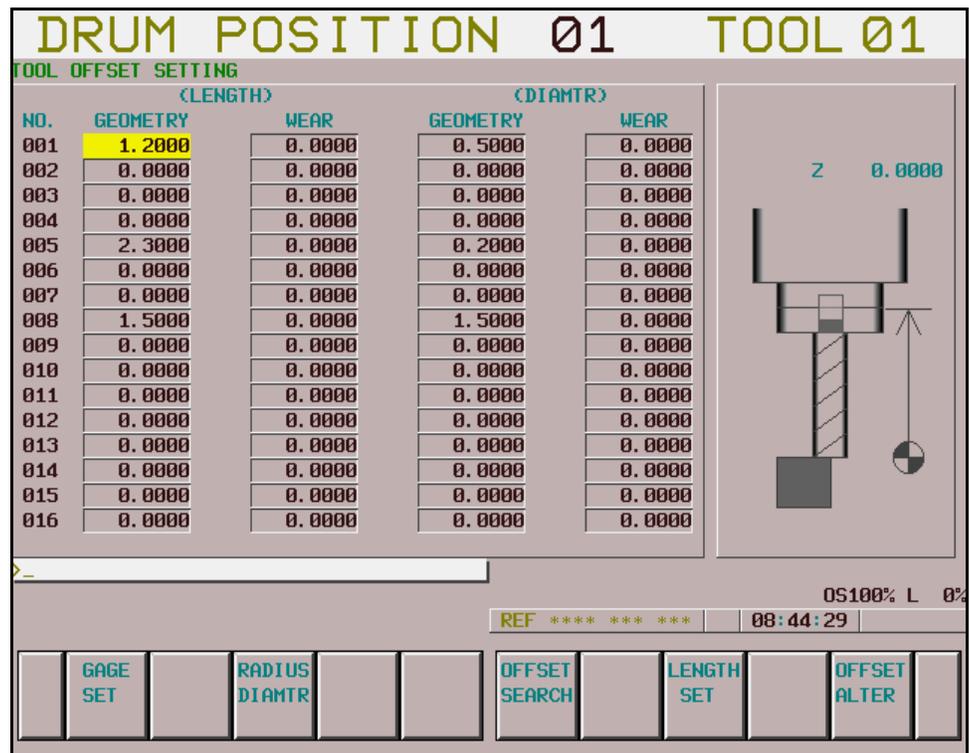


Figure 5-4: Tool Length Setting Screen

WARNING!

When inputting tool diameter/radius compensation, ensure the control is set to the correct status (see Step 3).

2. Pressing the RADIUS DIAMTR softkey will toggle the control between Radius and Diameter compensation.

5.4 TOOL LENGTH DIAMETER/ RADIUS ADJUSTMENT

3. All data in the compensation field will be re-calculated when the RADIUS DIAMRT softkey is pressed.
4. Using the cursor keys highlight the Radius/Diameter Geometry field and type in the value for the tool radius/diameter and press the INPUT key to enter the value into the Radius/Diameter Geometry field.
1. Adjustment of the values in the tooling tables is done as follows:
2. Press the LENGTH SET softkey - Figure 5-4: screen is displayed.
3. Using the cursor keys highlight the value to be altered.
4. Type-in the compensation to be applied e.g. -0.01.
5. Press the OFFSET ALTER softkey - this will subtract 0.01 from the original value.
6. Type in the compensation to be applied e.g. 0.02
7. Press the OFFSET ALTER softkey. This will add 0.02 to the original value.

6.0

DATUM SETTING

6.1 DATUM EDGE LOCATION

6.1.1 X - AXIS LOCATION

The objective of component datum setting is to inform the CNC control of the distance from the machine co-ordinate to the datum position in the X and Y axes (spindle centreline), and the distance from the machine co-ordinate to the datum position in the Z axis (spindle nose).

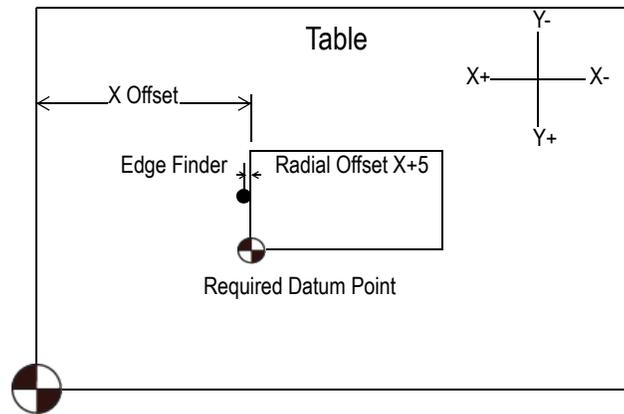


Figure 6-1: X-Axis Location

1. Press the JOG pushbutton.
2. Using the axis direction pushbuttons position the edge finder near the left-hand side of the work piece (Figure 6-1:).
3. Using a combination of the Handle X1, Handle X10, Handle X100, and the MPG Handwheel obtain contact with the work piece.
4. Press the DATUM SET pushbutton - the following screen is displayed:

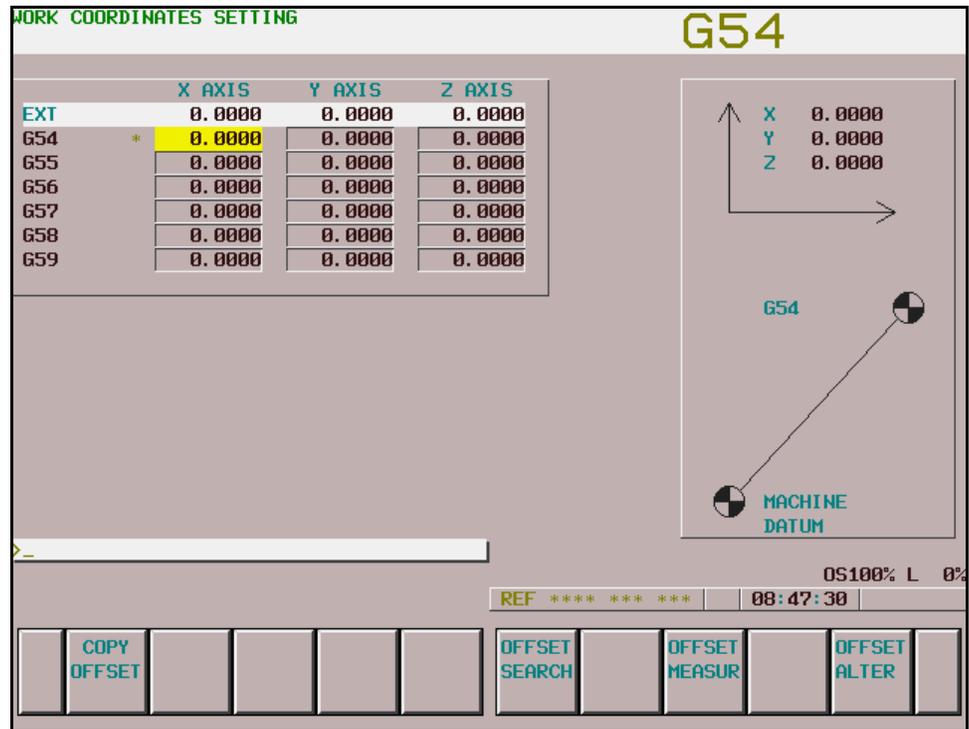


Figure 6-2: Work Coordinates Setting Screen

- Using the UP or DOWN cursor keys highlight the offset to be set, choose from G54 to G59 (but not EXT).
- Type the current programme position from datum edge to spindle centreline e.g. X+5 (Figure 6-1:)
- Press OFFSET MEASURE softkey.

The value now contained in the X axis table will be the distance from the machine coordinate to the datum edge.

6.1.2 Y - AXIS LOCATION

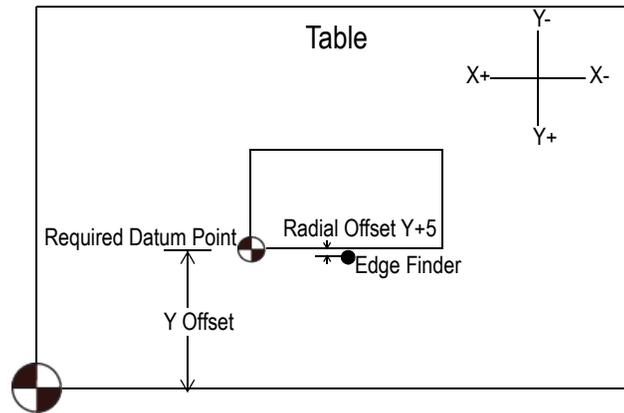


Figure 6-3: Y-Axis Location

1. Using the MPG Handwheel, raise the Z axis clear of the workpiece surface, and position the Edge Finder on the Y axis side of the workpiece.
2. Press the JOG pushbutton.
3. Using the axis direction pushbuttons position the edge finder near the lower edge of the work piece (Figure 6-3:).
4. Using a combination of the HANDLE x1, HANDLE x10, HANDLE x100 and the MPG Handwheel obtain contact with the work piece.
5. Type-in the current programme position from datum edge to spindle centreline e.g. Y+5 (Figure 6-3:).
6. Press the OFFSET MEASURE softkey (Figure 6-2:).
7. The value now contained in the Y axis table will be the distance from the machine co-ordinate to the datum edge.

6.1.3 Z - AXIS
LOCATION

1. Using the MPG Handwheel, position the spindle nose to a setting gauge block of known height on top of the required datum in the Z-axis.

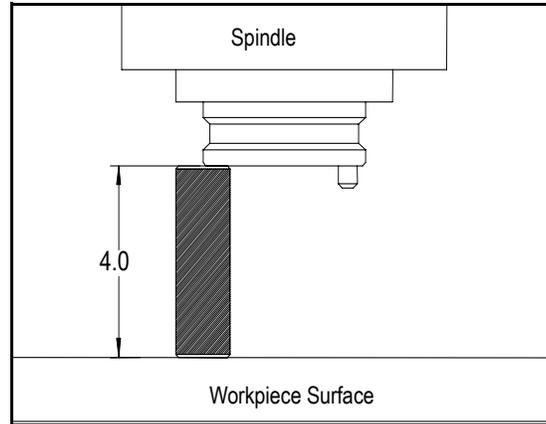


Figure 6-4: Z-Axis Location

2. Press the JOG pushbutton.
3. Using the axis direction pushbuttons, position the spindle nose near the gauge block on top of the work piece.
4. Using a combination of the Handle X1, Handle X10, Handle X100, and the MPG Handwheel obtain contact with the work piece.
5. Type-in the current programme position the spindle nose is above the workpiece e.g. Z4.0 (Figure 6-4:).
6. Press the OFFSET MEASURE softkey (Figure 6-2:).

The value now contained in the Z axis table will be the distance from the machine coordinate to the datum edge and should be a negative Z-axis value (Figure 6-5:)

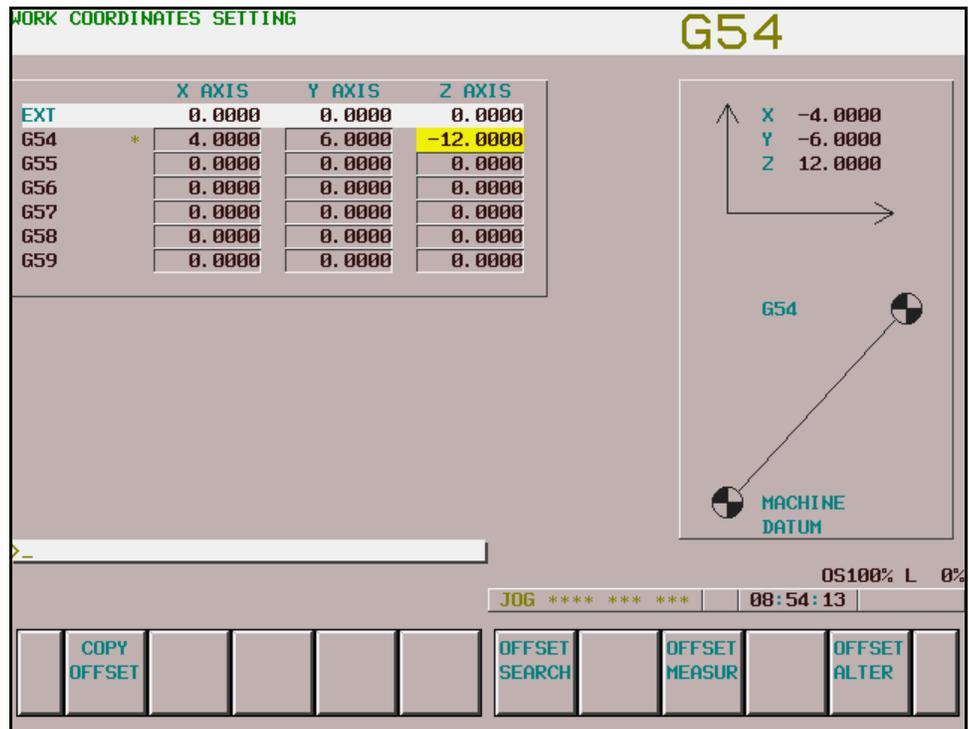


Figure 6-5: Work Coordinates Setting Screen

6.2 COPYING OFFSETS

When a multi-load fixture is in use on the machine and the distance between the various load stations is known, the Copy Offset feature can be used to reduce the offset setting time as follows:

1. Use the Component Datum Setting procedure (sections above) to set the initial position.
2. Using the cursor keys, highlight the offset just set (e.g. G54) (Figure 6-5:).
3. Press the COPY OFFSET softkey (Figure 6-5:).
4. Using the cursor keys move to the next offset to be used (e.g.. G55) and press the PASTE G54 softkey.

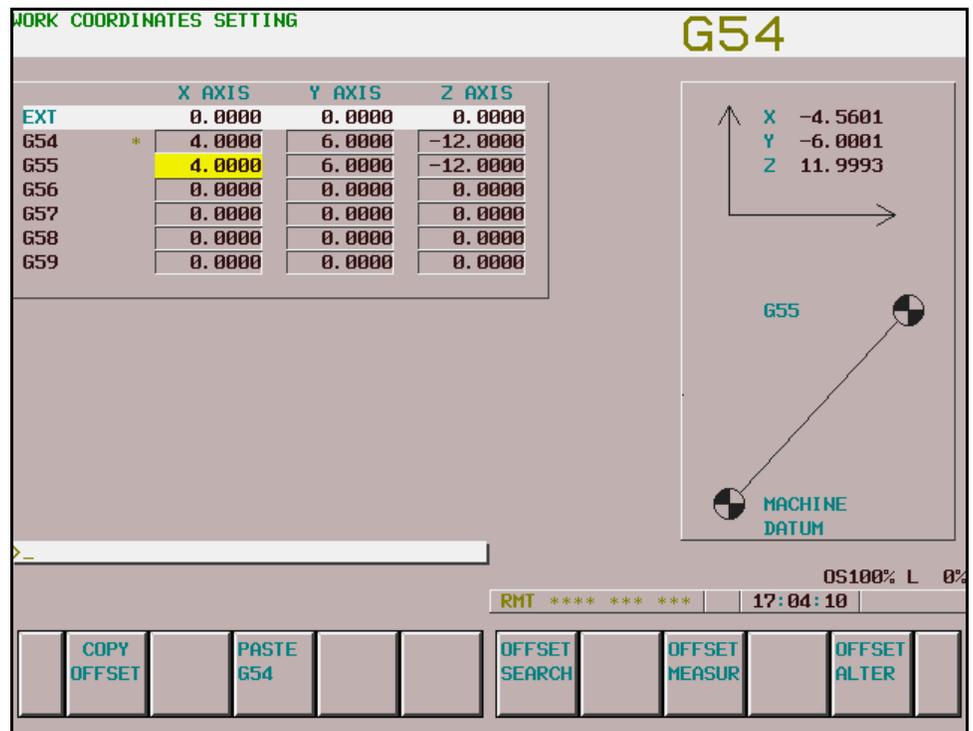


Figure 6-6: Paste G54 Softkey

Now G55 has same offsets as G54.

5. Assume the next load position is 4.0 inch to the right - highlight the X offset on G55.
6. Type-in 8.0 inch and press the OFFSET ALTER softkey (Figure 6-6:); this will add 8.0 inch to the value that was in G55 X-AXIS.

Repeat from step 2 for subsequent offsets.

6.3 GLOBAL DATUM SHIFT

1. It is useful when proving programmes to globally shift the Z-axis position in the positive direction. Using the EXT offset this can be done as follows:
2. Press the DATUM SET pushbutton.
3. Using the cursor keys move the cursor to the EXT Z offset.

4. Type 5.0 inch then press the input key - the following screen is displayed (Figure 6-7):

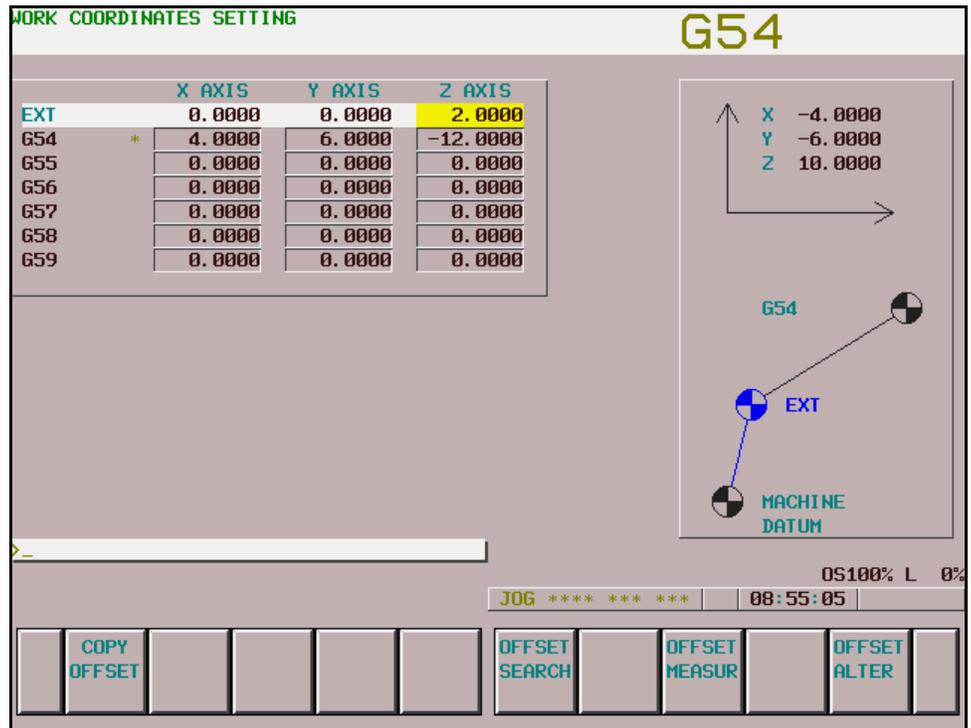


Figure 6-7: Work Coordinates Setting Screen

6.4 G92 POSITION SET

WARNING!

Do not change the work coordinate system when using G92.

Failure to heed this Warning may result in a collision with the workpiece or a fixture, and possible personal injury.

The G92 Position Set feature is used to set a local co-ordinate system without affecting any of the G54 to G59 positions. G92 is an additional offset to the active G54 to G59 position and is set as follows:

1. Using MPG Handwheel, position the Edge Finder on the X axis side of the workpiece.
2. Press the MDI pushbutton.
3. Type-in the current programme position from datum edge to the spindle centreline e.g. G92 X+5 (Figure 6-8:).

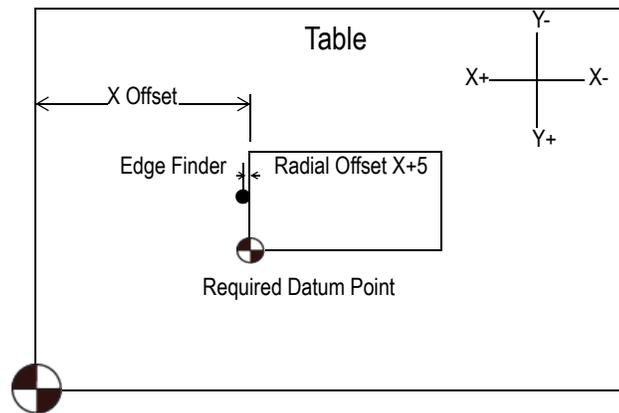


Figure 6-8: G92 Position Set

4. Press the CYCLE START pushbutton.

The X-axis position when looking at the absolute display will be X+5.

5. Using the MPG Handwheel, raise the Z-axis clear of the workpiece surface and position the Edge Finder on the Y axis side of the workpiece.

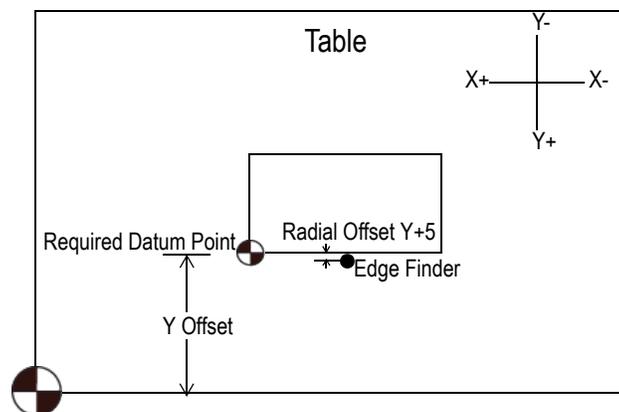


Figure 6-9: G92 Position Set

6. Press the MDI pushbutton.
7. Type-in the current programme position from datum edge to spindle center line e.g. G92 Y+5 (Figure 6-9:).
8. Press the CYCLE START pushbutton.

The Y axis position when looking at the absolute display will be Y+5.

- Using the MPG Handwheel, position the spindle nose to a setting block of a known height on top of the required datum in the Z-axis.

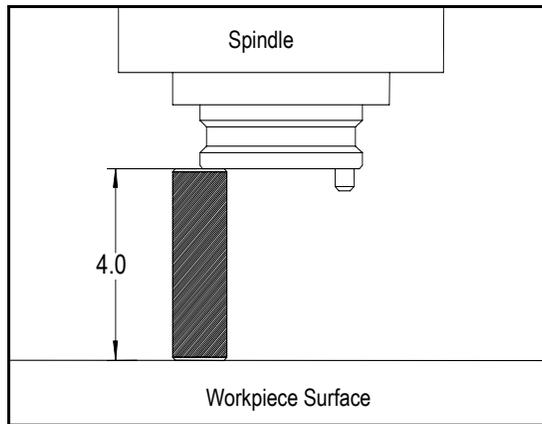


Figure 6-10:

- Press the MDI pushbutton.
- Type-in the current programme position from the work surface datum position to the spindle nose e.g. G92 Z4.0.
- Press the CYCLE START pushbutton.

The Z axis position when looking at the absolute display will be Z4.0.

6.5 G92 POSITION SET CANCEL

The G92 Position Set feature must be cancelled on completion of the task as follows:

1. Press the DATUM SET pushbutton - the following screen is displayed (Figure 6-11):

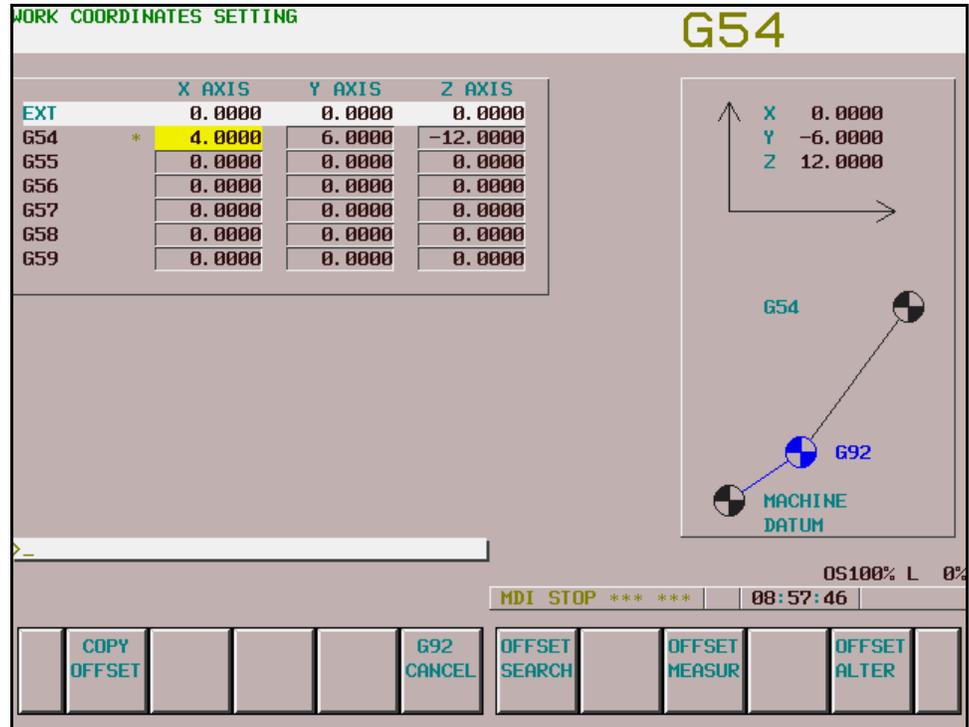


Figure 6-11:

2. Press the G92 CANCEL pushbutton.
3. The G92 Position Set feature is cancelled without any axes movement.

7.0

CALCULATOR

The calculator assists the operator to do the following common engineering calculations:

- Basic trigonometry.
 - Circular interpolation.
 - Speed and Feed calculations.
 - Addition, subtraction, multiplication, and division.
1. Press the CALC hard key to display the calculator. The display defaults to the last Calculator screen displayed.
 2. Pressing the PAGE UP/PAGE DOWN pushbuttons switches the calculator between the above functions.

7.1 TRIANGLE SOLVER

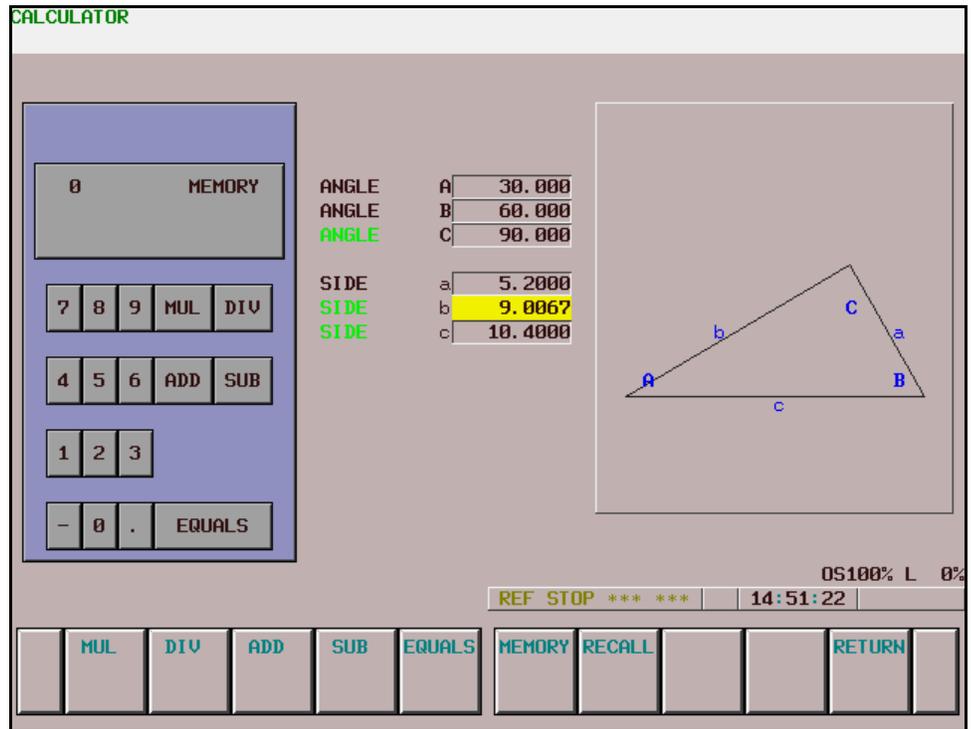


Figure 7-1: Triangle Solver

The minimum information needed to solve a triangle is 2 angles and 1 side, or 2 sides and one angle.

In Figure 7-1: the information entered was:

Angle A = 30 Angle B = 60 Side a = 5.2

The calculated results are in reverse video, and are:

Angle C = 90 Side b = 9.0067 Side c = 10.4

The operator can copy any of the triangle data into the memory by pressing the MEMORY softkey. Once the value is stored in the memory the PASTE softkey flashes, the operator can then either paste the value into a program being edited, or into MTB by using the PASTE pushbutton.

7.2 CIRCLE SOLVER

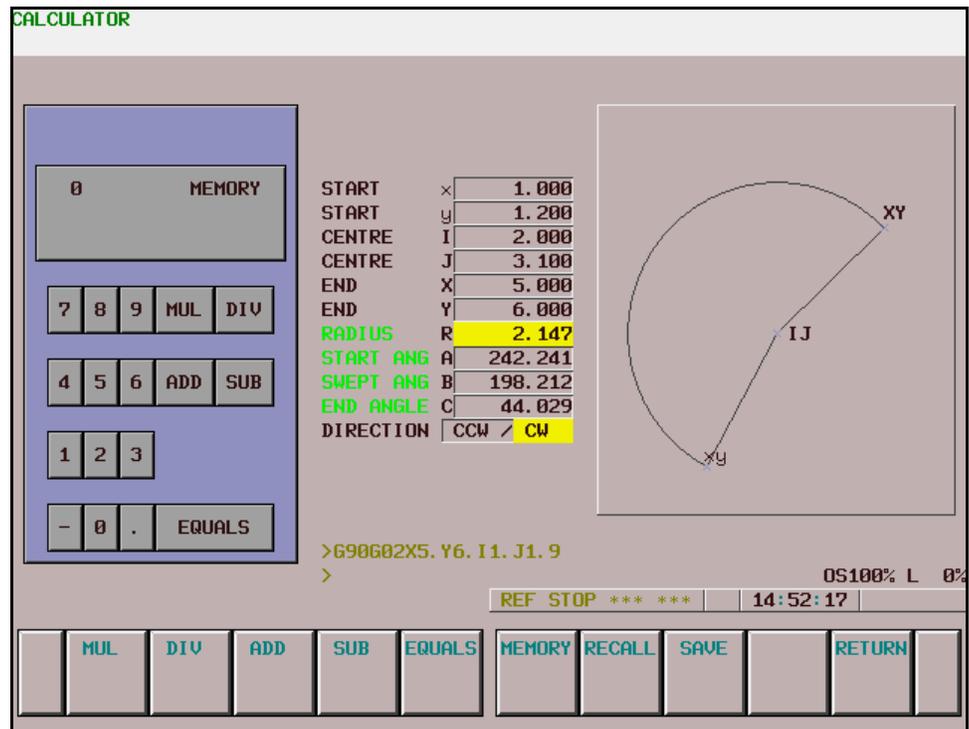


Figure 7-2: Circle Solver

In Figure 7-2: a simple quadrant is used. All data must be entered in absolute coordinates.

The information entered was:

- Start Position in the:X axis x=1.0
- Start Position in the:Y axis y=1.2
- Circle centre in the:X axis I=2.0
- Circle centre in the:Y axis J=3.1
- End Point in the:X axis X=5.0
- End Point in the:Y axis Y=6.0

7.3 SPEED AND FEED CALCULATOR

The calculated data names are in reverse video, and are:

- Radius:= 2.147
- Start Angle:= 242.241
- Swept Angle:= 198.212
- End Angle:= 44.029

The solution G code program is displayed as <G90G03X0.Y60.I-60.J0.

Pressing the SAVE softkey enables the G code solution to be pasted into either MDI or a program being edited using the MDI PASTE pushbutton.

This facility may be used as a built in cutting guide, values are input as Calculator. The facility has the ability to take lines of program from other screens and also to be able to be pasted to other screens.

Screens available under speed and feed calculator include the selection of material, work hardness, tool diameter, pilot hole diameter, cutting width, cutting depth, cutting speed and spindle speed. Screens vary slightly depending on machine specification.

NOTE

The cutting guide is intended as a general purpose guide to the selection of machining data. It is derived from average manufacturer's data and takes into account rigid tooling, workpieces and fixing. Power and force calculations are theoretical values and cannot be guaranteed. No account is taken of machine and workpiece stiffness and stability.

Consult the cutter manufacturer to establish optimum machining rates for a specific application.

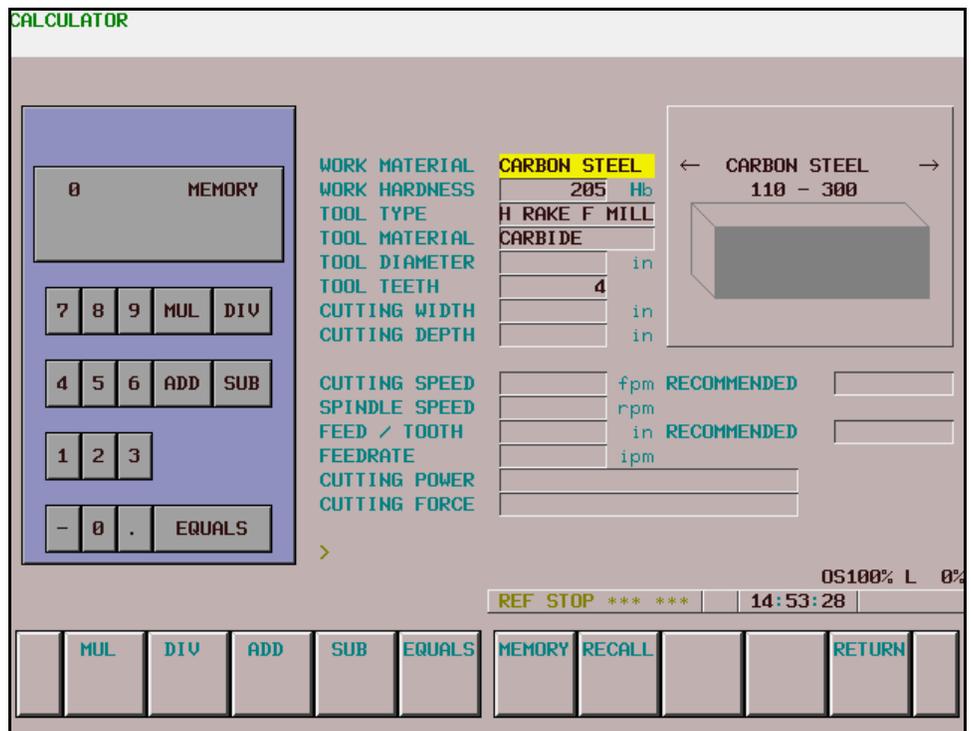


Figure 7-3: Work Material

Navigation between the screens is by using a combination of cursor keys.

Use the up/down keys to highlight the top field "WORK MATERIAL" Figure 7-3:.

Use the left/right keys to page through the available WORK materials, select by pressing a down cursor key. This action also inputs the next field "WORK HARDNESS" (Figure 7-4:).

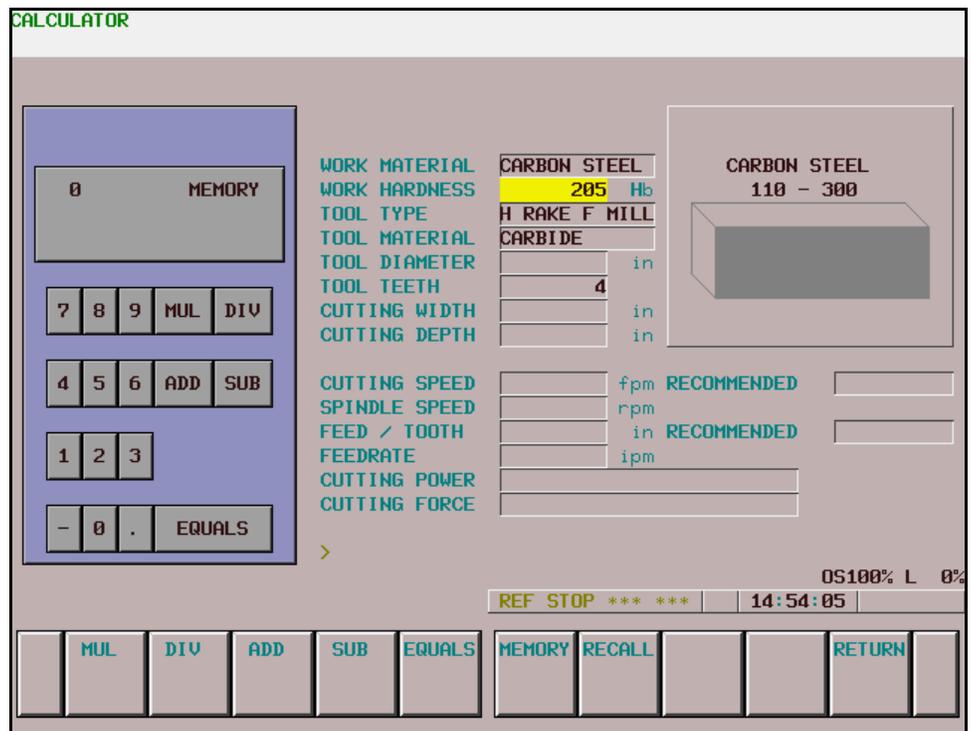


Figure 7-4: Work Hardness

Use the down cursor key to highlight "TOOL TYPE" (Figure 7-5:).

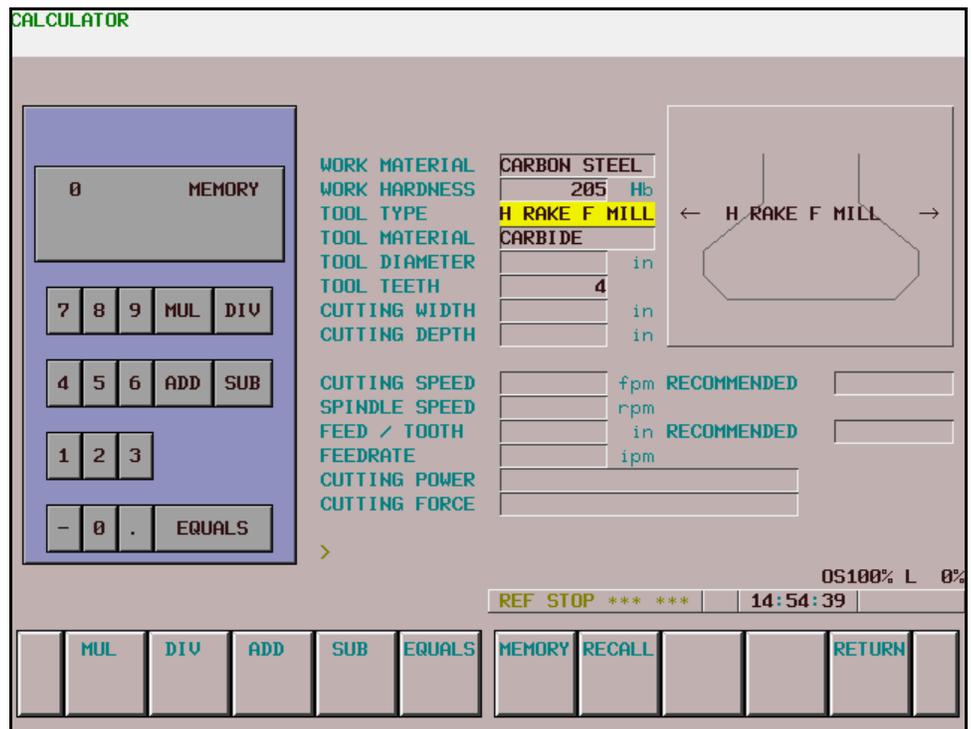


Figure 7-5: Tool Type

1. Use the left/right cursor keys to page through "TOOL MATERIAL" images, select by pressing a down cursor key.
2. Use the down cursor key to highlight "TOOL DIAMETER".
3. Input the required diameter using the numeric keypad.
4. Use the down cursor key to highlight "TOOL TEETH".
5. Input the required number using the numeric keypad.
6. Use the down cursor key to highlight "CUTTING WIDTH".
7. Input the required dimension using the numeric keypad.
8. Use the down cursor key to highlight "CUTTING DEPTH".
9. Input the required dimension using the numeric keypad.
10. Use the down cursor key to highlight "CUTTING SPEED".
11. Input the required value using the numeric keypad.
12. Use the down cursor key to highlight "SPINDLE SPEED".
13. Input the required value using the numeric keypad.
14. Use the down cursor key to highlight "FEED/TOOTH".
15. Input the required value using the numeric keypad.
16. Use the down cursor key to highlight "FEEDRATE".
17. Input the required value using the numeric keypad.
18. Press MEMORY softkey to retain the input values.
19. Press RETURN softkey to enter values into the program.

8.0

QUICK CUT



This feature enables the operator to set a Speed and Feed to allow the machine to be used under manual control. QUICK CUT and JOG pushbutton LED's are both illuminated.

Pressing the QUICK CUT pushbutton will display the following screen:

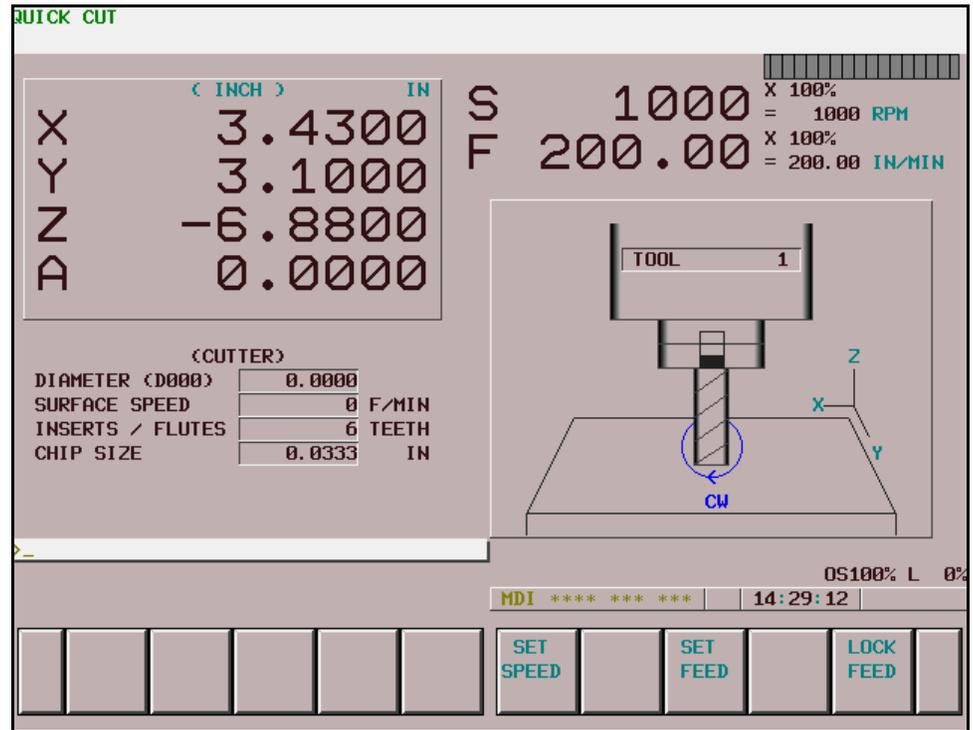


Figure 8-1: Quick Cut Screen

The speed and feed displayed are the last speed and feed used by the control system.

To set the speed, press MDI button, use the number keypad and type-in the required rpm e.g.. 100. and then press the SET SPEED soft key.

To set a feed, press MDI button, use the number key-pad and type-in the feed required in mm per min e.g.. 200, and then press the SET FEED soft key.

After entering the speed and feed, the screen is updated as shown in Figure 8-2:

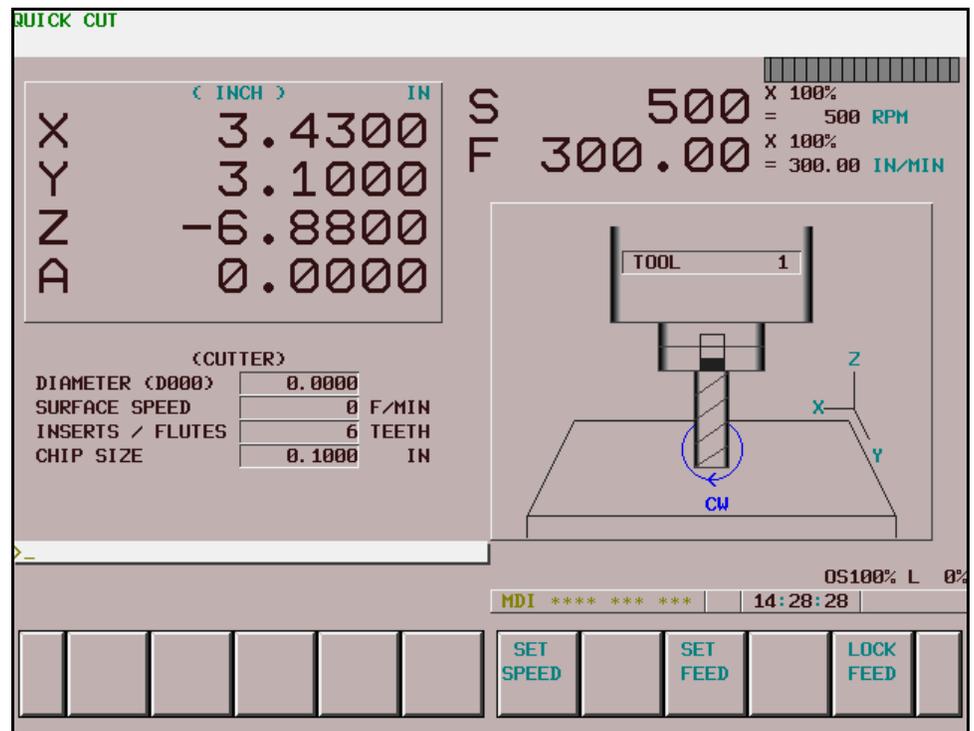


Figure 8-2: Quick Cut Screen

The machine is now ready for use under manual or MDI control.

8.1 SPINDLE CONTROL

Control the spindle as follows:

1. Press MDI pushbutton.
2. Press FUNC + SPINDLE CW or FUNC + SPINDLE CCW pushbuttons on the Operator Panel A to start spindle in the right direction.
3. Changing the spindle speed can be done in two ways:
 - Without stopping the spindle in the MDI mode, by typing in the required spindle speed e.g. 400 and then pressing the SET SPEED soft key, the spindle speed will be immediately changed.
 - The SPINDLE SPEED OVERRIDE selector switch can also be used to override the commanded speed within the range of 50 to 120%.

8.2 FEED CONTROL

The operator can use the machine in JOG mode to manually feed a cutter across a workpiece, or simply to position a cutter to the start point of each machining pass across the workpiece. It may also be used to feed a cutter across a workpiece whilst performing a machining process. This requires the operator to keep the relevant AXIS

JOG pushbutton pressed, releasing the pushbutton will stop axis motion. Axis Jog pushbuttons X+, X-, Y+, Y-, Z+ and Z- may be selected in this mode. Axis motion is at AXIS JOG feedrate (not the selected Quick Cut feedrate) which may be modified by using the FEED OVERRIDE selector switch.

Alternatively, the LOCK FEED function can be used. This feature allows a cutter to feed across a workpiece at the specified Quick Cut feedrate.

1. Press the MDI pushbutton.
2. Press FUNC + SPINDL CW or FUNC + SPINDL CCW pushbuttons to start spindle in the right direction.
3. Press the LOCK FEED softkey to change the screen display as shown below.

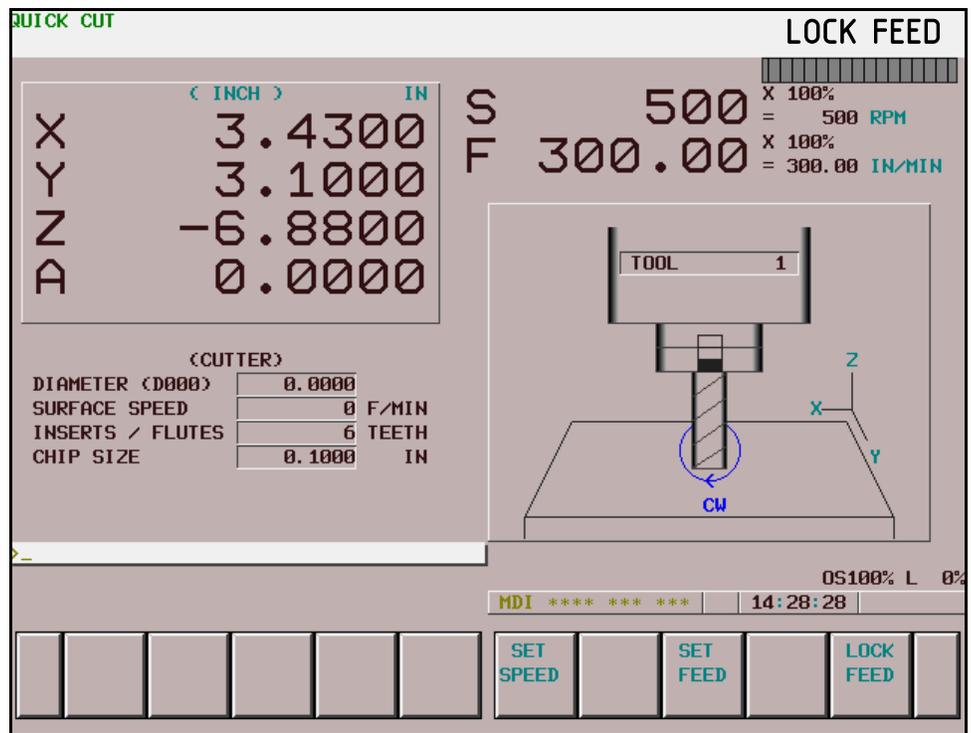


Figure 8-3: Feed Lock

4. Press the X+, X-, Y+, or Y- axis selection pushbutton for the desired feed direction. The selected feed is now locked, and the pushbutton need not be kept pressed. Axis motion is at the specified QUICK CUT feedrate which may be modified by using the FEED OVERRIDE selector switch.
5. Pressing any key on the keypad will terminate the axis feed. Terminating axis feed cancels the Lock Feed function.

NOTE

It is not possible to use Lock Feed in the Z-axis.

9.0

AICC FUNCTION

9.1 PROGRAMMING CONSIDERATIONS

AICC is provided for high-speed, high-precision machining. This function improves acceleration/deceleration and servo response during the high-speed machining. Control can look ahead up to 40 blocks to execute smooth acceleration/deceleration over multiple blocks and higher machining speed and reducing profile error.

The feature is primarily designed for Die and Mould applications where there is a high accuracy requirement and to overcome tool path control when generating sharp corners and small radii.

AICC Enable Format:- G5.1 Q1 Rx

where:-

Rxx provides the user with the option of selecting from 10 fixed settings (R1 - R10) which contrast Tool Path Speed (feedrate) with Positioning Accuracy, i.e.,

G5.1 Q1 R1	-Tool path speed has priority
G5.1 Q1 R2	over Positioning Accuracy
G5.1 Q1 R3	
G5.1 Q1 R4	
G5.1 Q1 R5	-Tool Path Speed and path
G5.1 Q1 R6	accuracy have equal priority
G5.1 Q1 R7	
G5.1 Q1 R8	
G5.1 Q1 R9	
G5.1 Q1 R10	-Positioning Accuracy has priority over Tool Path Speed

R1-R2: Smoother axis motion. Improved accuracy.

R3-R5: Die & Mould - roughing.

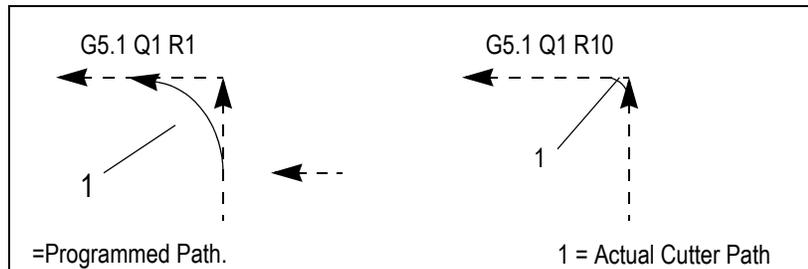
R6-R8: Die & Mould - semi-finish.

R9-R10: Best accuracy, finish, form & smooth axis motions.

R1 = Rough machining. R10 = Finish machining.

Using R1, a 90 degree corner produced at feedrate of 10,000mm/min will give an approx. deviation of 0.15mm.

Using R10, a 90 degree corner produced at feedrate of 2,000mm/min will give an approx. deviation of 0.01mm.



AICC Disable Format:- G5.1 Q0

The AICC feature is also disabled by M02, M30 and by pressing the RESET Button.

NOTE

Always enable / re-enable the AICC before a Tool Change (M06) block and G43 block; or a Program Stop (M00/M01) block. The 'G5.1 Q1 Rxx' block must be inserted immediately after activation of Work Offset (G54 - G59) and before Tool Length Offset (G43) data blocks, see Example Program below.

Always disable the AICC feature prior to processing a Tool Change (M06) block, or a Program Stop (M00/M01) block.

Example program:

```

%
O0001(PART NAME)
G0 G90 G40 G49 G94 G80
G21
G0
G5.1Q1R1(AICC ENABLED-ON) -           Tool path speed has priority
                                       over Positioning Accuracy

T1 M6 (10MM DIA ENDMILL)
(ROUGH MACHINE COMPONENT)
G54
G1 X10 Y50 F3000
S7500 M3
G43 Z100 H1
Z-10
    
```

G0 Z100 M9

M5

G5.1Q0 (AICC DISABLED - OFF)

G28 Z0.0

G49

G5.1Q1 R5(AICC ENABLED-ON) -

Tool path speed and Position Accuracy have equal priority

T2 M6 (12MM DIA ENDMILL)

(SEMI-FINISH COMPONENT)

G54 X10 Y50 F3000 S7500 M3

G43 Z100 H2

Z-10

G0 Z100 M9

M5

G5.1Q0 (AICC DISABLED - OFF)

G28 Z0.0

G49

G5.1Q1 R8 (AICC ENABLED-ON) -

Positioning Accuracy has priority over Tool path speed

T3M6 (8MM DIA BALL ENDMILL)

(FINISH MACHINE COMPONENT)

G54 X10 Y50 F3000 S7500 M3

G43 Z100 H3

Z-10

G0 Z100 M9

G5.1Q0 (AICC DISABLED - OFF)

G28 Z0.0

M30

%

**9.2 AICC PRECISION
LEVEL DATA
SCREEN**

The current Precision Level of the AICC feature may be viewed at any time by following the procedure given below:

1. Press the OFFSET SETTING pushbutton.

2. Press the [+] softkey twice or until legend "PR-LEV" appears on a screen softkey.
3. Press the [PR-LEV] softkey.
4. Press the [APC/AI] softkey.

Press the up/down cursor keys to display AICC data for the desired axis. Figure 9-1: shows AICC parameter data for the X-axis.

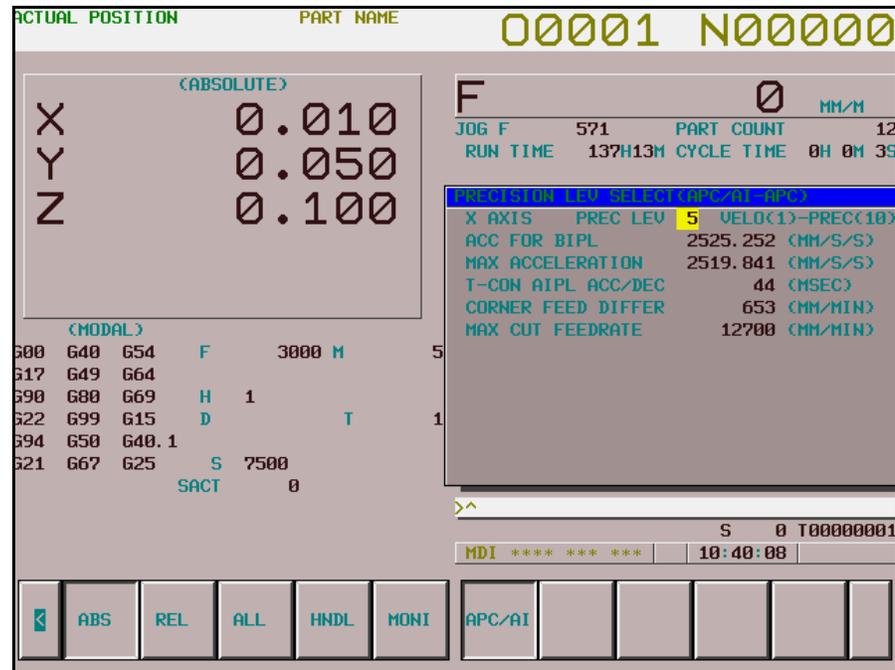


Figure 9-1: AICC Precision Level Data Screen

The screen in Figure 9-1: indicates the AICC Precision Level ("5" shown in highlight - i.e., G5.1Q1 R5 active) and all parameters associated with the X-axis at the active Precision Level. The screen is provided for viewing only.

For further information relating to this feature, refer to GE Fanuc Operator Manual publication B-64124EN- Section II Programming, Chapter 19.3 'AI ADVANCED PREVIEW CONTROLFUNCTION/AI CONTOUR CONTROL FUNCTION'.

GE Fanuc Manuals may be found elsewhere on this CD-ROM.

10.0

TOOL MANAGEMENT

10.1 TOOL LIFE MANAGEMENT

The CNC system may have an optional control feature for managing tool life.

Tool life management enables the user to locate identical (sister) tools into tool groups. A tool group T-number is specified in the NC program from which the system decides which sister tool from the group is to be used. A 4 digit T-number (e.g. T0910) is used to define Tool Group #10.

The number of tool groups, and the tool life measurement units are defined by an 8-bit parameter 6800. The default bit pattern is as follows:

Parameter	Bit	7	6	5	4	3	2	1	0
6800		0	0	0	0	0	0	0	0

This bit pattern allows for 16 tool groups with tool life measured by usage count.

NOTE

Param 6800 Bit 0 defines the Useful Tool Life Group size i.e.:

- 0 = 16 groups, each group comprising 16 tools
- 1 = 32 groups, each group comprising 8 tools

Param 6800 Bit 2 defines tool life units i.e.:

- 0 = usage count,
- 1 = cycle time count (minutes) with spindle rotating at feedrate motion. Tool Life Units may also be defined in the "Edit Tool Life" screen display.

The screenshot shows a terminal window titled "TOOL LIFE DATA :". The screen displays the following information:

NEXT ***	USE ***	SELECTED GROUP: ***
GROUP 009:C	LIFE 0000	COUNT 0000
GROUP 010:M	LIFE 0500	COUNT 0345
0001	0002	

At the bottom of the screen, it says "TO BE CHANGED :".

Figure 10-1: Tool Life Data

Figure 10-1: shows tools allocated to Tool Group 10.

EDIT LIFE DATA GROUP : 010				
TYPE :	2 (1:C 2:M)	NEXT GROUP :	***	
LIFE :	0500	USE GROUP :	***	
COUNT :	0345	SELECTED GROUP :	***	
NO.	STATE	T-CODE	H-CODE	D-CODE
01		0001	001	001
02		0002	002	002

Figure 10-2: Edit Life Data Group

Figure 10-2: shows editing of Tool Group 10.

Display shows:

- *Type* - Tool life units in minutes (type 2)
- *Life* - Life per tool is 5.00 minutes
- *Count* - Accumulated time of active tool is 3.45 minutes.
- *T code* = Tool number

In tool group 10, T01 and T02 are designated with a length offset (H1 / H2) and a radius offset (D1 / D2).

T01 and T02 are tool numbers, and are called sequentially via a dedicated 4-digit Tool Group Number T0910.

Specifying Tool Life:

Life type may be conducted in either usage count (1 = C) or machining time in minutes (2 = M)

Cycle time is measured as the spindle is turning at feedrate

Usage count is incremented on processing M02 or M30. (NB a 'loop' program will not increment the usage counter.)

When the active tool has exceeded its life time limit, and asterisk (*) appears in the STATE column for that tool. The affected tool is allowed to continue in-cycle until reaching the next tool change block. The next time the affected tool group is selected, the system allows the next available tool in the group to become the active tool.

An active tool is designated with the symbol (@) in the STATE column. When all tools in a tool group have exceeded their tool life limit (i.e. cycle time or usage count) and the tool is again called, this results in an alarm.

Programming Tool Life:

When the Tool Life Management feature is active, the Tool Call (M6) block specifies the Tool Life Group number, not the Tool number. Hence, "T0901 M6" calls for the first available tool from Tool Life Group 1. "T0901 - T0916" is the range of Tool Life Group tool numbers when the control is arranged with 16 Tool Life Groups.

Tools from tool group 10 may be accessed by programming:

T0910 M6

The program extract shown here utilizes the first available tool from Tool Life Group 10. The system automatically identifies the associated Length Offset H-register by programming H99. Similarly, the system identifies the Cutter Radius Offset D-register by processing D99 in a G41/G42 block.

N210	Tn*	M6				
N220	G0	G90	G40	G17	G94	G80
N230	G21					
N240	G54	X---	Y---	S---	M3	
N250	G43	Z---	H99			
N260	G41	X---	Y---	D99		
N270	X---					
N280	Y---					
N290	G40	X---				
N300	G0	Z---				

* Tn = 0910

The actual Hxx and Dxx codes are specified along with the Tool numbers in the EDIT LIFE DATA screen of the Tool Life Group.

H00 cancels the active Tool Length Offset

D00 cancels the active Cutter Radius Offset

10.2 TOOL LIFE MANAGEMENT ACCESS

To access Tool Life Management, select Tool Life Group following the procedure bellow:

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad.
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the "EDIT TOOL DATA" screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor pushbuttons control to advance to the desired tool group.

10.3 ADD A TOOL

To add a Tool to the selected Tool Group follow the procedure bellow:

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the "EDIT TOOL DATA" screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor pushbuttons control to advance to the desired tool group.
8. Press [OPRT] softkey
9. Press [EDIT] softkey (this displays EDIT LIFE DATA screen for the selected group)
10. Enter Tool No (e.g. 0005)
11. Press [INSERT] softkey.

10.4 ADD MORE TOOLS

Tool number 0005 is now displayed in the T-CODE column, as are the H-Code (tool length offset register field) and D-Code (tool radius offset register field), all may now be added.

To add more tools to the Tool Group follow the procedure below:

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the "EDIT TOOL DATA" screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor pushbuttons control to advance to the desired tool group.
8. Press [OPRT] softkey
9. Press [EDIT] softkey (this displays EDIT LIFE DATA screen for the selected group)
10. Enter Tool No (e.g. 0005)
11. Press [INSERT] softkey.
12. Add offset number in H-Code and D-Code fields as required.

10.5 TOOL LIFE LIMIT SETTING

To set Tool Life Limit follow the procedure below:

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the [EDIT TOOL DATA] screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey

6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor controls to advance to the desired tool group.
8. Press [OPRT] softkey
9. Press [EDIT] softkey (this displays EDIT LIFE DATA screen for the selected group)
10. Cursor pushbuttons control to advance to the TYPE field.
11. Type [1] for usage count life or 2 for life in minutes.
12. Press INPUT keypad to confirm data entry.
13. Cursor pushbuttons control to advance to the LIFE field
14. Enter Tool Life (0200 = 2 minutes or 200 cycles of the part program)
15. Press INPUT keypad to confirm data entry

The accumulated usage time in minutes or the accumulated usage count of the active tool is displayed in the COUNT field.

NOTE

The input of there Tool Life Limit by Time or by Usage Count to the active group (as above) overwrites the default setting of the Tool Life Limit by Time.

10.6 DELETE A TOOL

To delete a tool from the selected Tool Life Group follow the procedure bellow:

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the "EDIT TOOL DATA" screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor pushbuttons control to advance to the desired tool group.
8. Press [OPRT] softkey

9. Press [EDIT] softkey (this displays EDIT LIFE DATA screen for the selected group)
10. Cursor pushbuttons control to advance to the selected tool to be deleted.
11. Press [DELETE] softkey
12. Press [<CRSR>]softkey

Selected tool is deleted from the Tool Life Group.

10.7 DELETE ALL TOOLS

To delete all the tools from the selected Tool Life Group follow the procedure bellow'

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the "EDIT TOOL DATA" screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor pushbuttons control to advance to the desired tool group.
8. Press [OPRT] softkey
9. Press [EDIT] softkey (this displays "EDIT LIFE DATA" screen for the selected group)
10. Press [DELETE] softkey
11. Press [GROUP] softkey
12. Press [EXEC] softkey

All tools are deleted from the selected Tool Life Group.

10.8 RESET "TOOL LIFE EXPIRED" STATUS FOR THE TOOL

To reset "Tool Life Expired" status for selected Tool in the Group follow the procedure below:

1. Press [MDI] key on the operator panel.

10.9 RESET "TOOL LIFE EXPIRED" STATUS FOR ALL TOOLS

2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the "EDIT TOOL DATA" screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor controls to advance to the desired tool group.
8. Press [OPRT] softkey
9. Press [EDIT] softkey (this displays EDIT LIFE DATA screen for the selected group)
10. Cursor pushbuttons control to highlight tool identified with * in state column
11. Press [STATE] softkey
12. Press [CLEAR] softkey

To reset "Tool Life Expired" Status for all the Tools in the Group follow the procedure below:

NOTE

The Tool Group Life Expired LED is illuminated on operator control panel.

The control screen displays alarm "TOOL GROUP EXPIRED". The illuminated LED and alarm occur: at the end of program if Usage Count is the life measure. During the machining cycle of the tool whose life expires if Cycle Time is the life measure.

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the [EDIT TOOL DATA] screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey

5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor pushbuttons control to advance to the desired tool group.
8. Press [CLEAR] softkey
9. Press [EXEC] softkey

Asterisks (*) in the STATE column are removed. All tools in the group are given a full tool life period.

10.10 TOOL SKIP

To skip a Tool from a Tool Life Group follow the procedure below:

softkey operation in the EDIT LIFE DATA screen may be used to designate a tool as having no useful life, i.e.:

1. Press MDI pushbutton on the operator panel.
2. Select OFFSET/SETTING keypad
3. Press [+] softkey on the LCD screen until [TOOLLF] softkey comes out.
4. If the "EDIT TOOL DATA" screen is present, but does not display the desired Tool Life Group, then:
 - Press [OPRT] softkey (if present)
 - Press [END] softkey
5. Press [>] softkey
6. Press [TOOLLF] softkey (this displays two of the tool groups)
7. Cursor pushbuttons control to advance to the desired tool group.
8. Press [OPRT] softkey
9. Press [EDIT] softkey (this displays EDIT LIFE DATA screen for the selected group)
10. Cursor pushbuttons control to highlight tool identified with (*) in STATE column
11. Press [STATE] softkey
12. Press [SKIP] softkey.

11.0

MANUAL GUIDE I

11.1 MANUAL GUIDE I FUNCTION



MGI pushbutton enables the CNC to enter the Manual Guide i mode.

MANUAL GUIDE i is the operation guidance to support an operator in many situations such as creating a part program, checking by machining simulation, set-up and actual machining. These operations can be done on the only one screen. ISO code form is adopted as the part program format, it is widely used on many CNC machine tools, furthermore, machining cycles, which can realize complicated machining motions by simple programming, are provided.

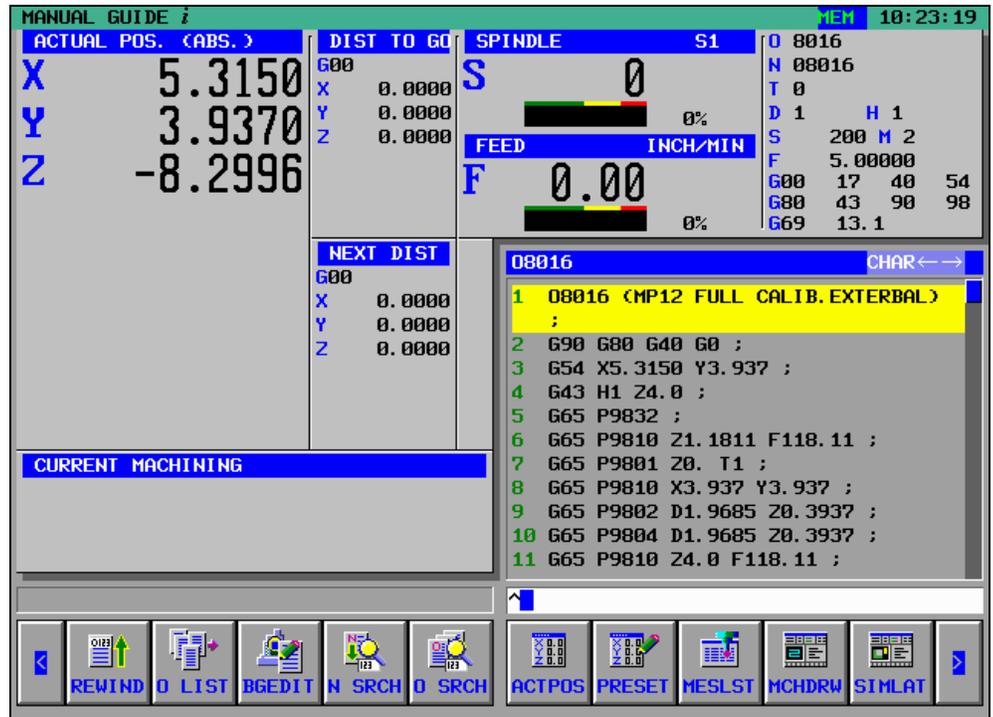


Figure 11-1: Manual Guide i Screen

By using Manual Guide i, the operator can carry out routine machining easily.

1. Integrated operation screen that enables almost all routine machining operations

A single integrated operation screen enables routine machining operations including machining program input/editing, animated simulation-based machining program checks, production machining, MDI operations, and manual operations with JOG and HANDLE.

2. Machining programs in ISO code format

Using ISO code machining programs, which are in wide use, enables the operator to specify simple operations with simple commands, such as those for straight lines and arcs, and complicated machining operations with machining cycles easily.

3. High affinity with CAD/CAM

ISO code machining programs created using CAD/CAM can be used without modification. Adding advanced machining cycles to these machining programs makes them perfect machining programs. They can be checked easily, using animated simulation.

4. Advanced machining program editing

Using advanced editing functions, such as substring search and cut/paste via the clipboard, enables easy editing of machining programs.

5. Advanced machining using machining cycles (option)

Advanced machining cycles are available which cover various types of machining including milling and turning. With these machining cycles, it is possible to perform complex machining by creating and running programs easily.

6. Fixed format program menu-driven simple program entry

Register a series of frequently used machining operations previously as a menu, and select necessary machining operations from the menu when creating a machining program. This method can eliminate the trouble of entering similar machining operations repeatedly.

7. M code menu

It is possible to input M codes easily by referencing explanations displayed in an M code menu. Machine tool builders can create the explanations easily.

8. Realistic animated simulation (option)

Machining programs can be checked easily, using an animated simulation method that can realistically show what the surface machined with a specific type of tool tip is like. In addition, you can check a simulated workpiece as if you were looking at a real workpiece because the animated simulation method uses solid models for all operations, from milling to turning, for the workpiece.

9. Advanced set-up guidance (option)

It is possible to set up machining operations and check the precision of machined workpieces easily, using an advanced set-up guidance function that can handle all measurements, from tool offset measurement to the measurement of workpieces in machine tools.

10. Supporting a wide variety of machine types including lathes and machining centers.

In MANUAL GUIDE i, basically, only one screen called the All-in-one Screen is used for all the operations from trial machining to actual machining.

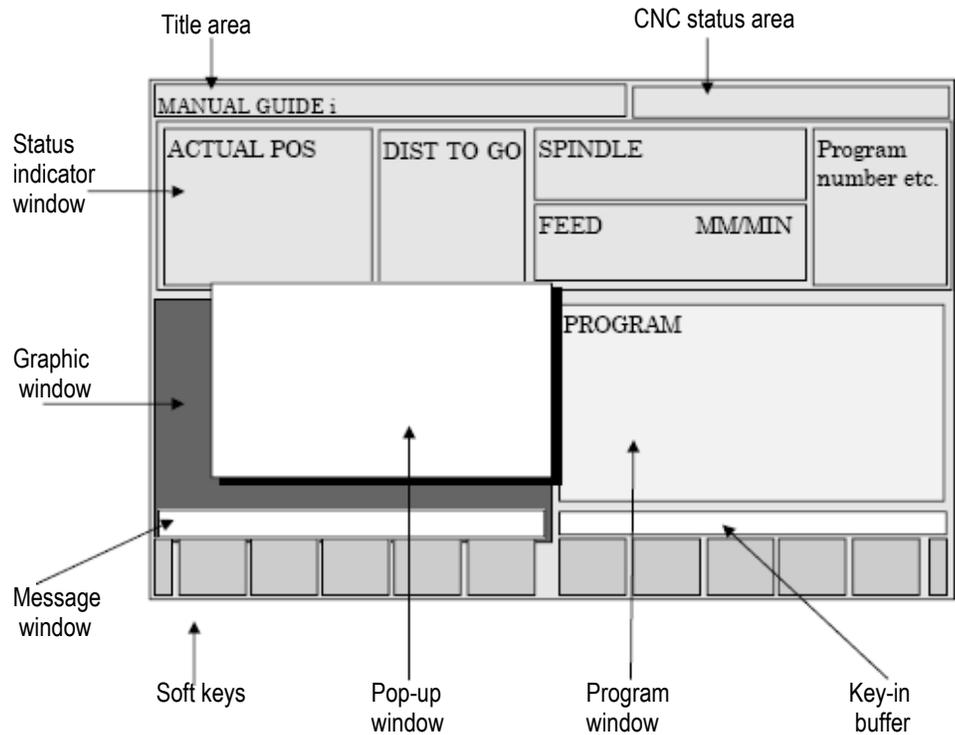


Figure 11-2: Screen windows and displays

1. Title area of MANUAL GUIDE i is always displayed.
2. CNC status area displays the following CNC statuses:
 - Mode
 - Alarm status
 - Reset or emergency stop status
 - Actual time
3. Status indicator window displays the following information:
 - Actual machine position
 - Remaining moving distance of the actual block
 - Actual speed and load meter (for the axis with the maximum load)
 - Spindle rotating speed and spindle load meter

- Program number and process number
 - Command values during automatic operation (M,S,T,F)
4. Graphic window displays the following graphical drawing:
 - Animated drawing with a solid model (machining simulation)
 - Tool path drawing
 5. Program window displays the machining program.
 6. Pop-up window displays the following supplemental screens:
 - M-code menu
 - Fixed format statement menu
 - Set data (workpiece coordinate system, tool offset, fixed format statement registration)
 - Program list
 - Data input window for machining cycles
 7. Key in buffer displays comments on data and input numerical data.
 8. Message window displays the following messages:
 - Meaning of the word which a cursor is allocated (Guidance Message)
 - Meaning of Machine Cycle which is executing
 - Content of Warning and Alarm
 9. Soft keys displays the comments on the following soft keys:
 - Editing operation menu
 - Machining cycle menu
 - Pop-up window menu

12.0

DATA I/O

12.1 DATA I/O FUNCTION



Data Input/Output screen displays list of programs that are stored in the CNC memory.

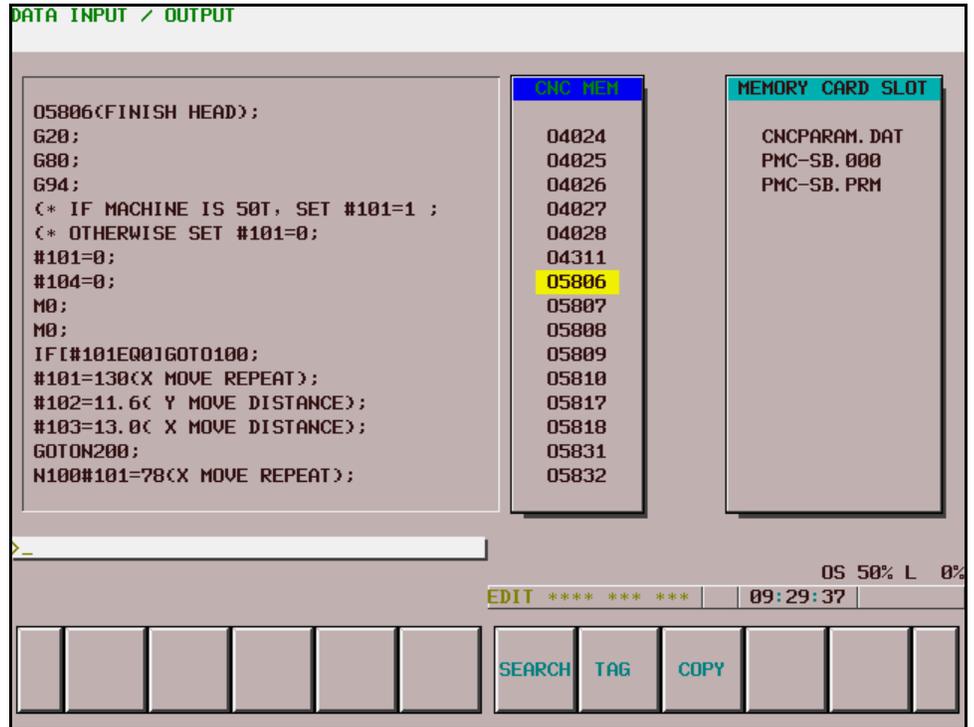


Figure 12-1: Data Input/Output Screen

Data Input/Output screen displays the CNC memory list of programs and memory card slot list of programs.

Once the program is selected, contents of the program is displayed in the left box of the screen.

It is possible to copy the program from CNC memory and paste it in the memory card slot and vice verse.

1. Select the program.
2. Press Copy softkey.
3. Paste the program in the desired memory window.

To search for a program:

1. Input program number.
2. Select SEARCH softkey.

If this program is in one of the memories, screen will display it.

To select more than one program, select TAG softkey. If program is selected, little star appears in front of the program indicating that the program is selected. To copy all the selected programs select COPY softkey, and it will copy all the selected programs.

13.0

COOLANT CONTROL

13.1 AUTO COOL

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

AUTO COOL pushbutton toggles on/off, as indicated by LED. While AUTO COOL is active, manual coolant control pushbuttons MIST COOL and FLOOD COOL are inhibited. AUTO COOL "OFF" enables control of coolant manually. 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

13.2 FLOOD COOLANT

FLOOD COOL turns ON/OFF the flood coolant pump when the AUTO COOL is OFF.

13.3 MIST COOL

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

13.4 COOLANT THROUGH SPINDLE (OPTIONAL)

MIST COOL pushbutton is used to turn ON/OFF the coolant through spindle when the AUTO COOL is OFF. When the AUTO COOL pushbutton LED is ON, M7 is used to turn ON coolant through spindle and M9 is used to turn it OFF.

When the coolant through spindle is in OFF position (by pushbutton or M9 command), the coolant through spindle pump is turned OFF immediately. After coolant through spindle is in OFF position for 4 seconds, the air seal of the coolant through spindle turns OFF also.

NOTE

FLOOD COOL and MIST COOL/Coolant through spindle may be turned on simultaneously.

14.0

DATA SERVER

14.1 DATA SERVER OPTION

Moving Data Around on the Data Server

There are many different ways to move data around using the Data Server option. They vary depending on your specific CNC setup. Please refer to the document "Ethernet/ Data Server for i-Series Described" for a more detailed description of the different Data Server modes.

For this document we will cover only the case was you have a true Data Server with storage.

Section #a Punch a Part Program between CNC and Data Server

For Series 0iMC

1. Verify that parameter 20 (or 21 depending CNC setup) is 5
2. Place the CNC in "EDIT" mode.
3. Press function key "PROG"
4. Press the continuation menu soft key "+" at the right end of the soft key menu.
5. When you press soft key [PRGRM], the Program screen appears.
6. Press soft key [(OPRT)].
7. Press the continuation menu key at the right end of the soft key menu.
8. Press soft key [PUNCH].
9. Enter the file number or name of the NC program using the MDI keys.
 - a. [Input format] <O-number>
10. Press soft key [EXEC].
11. During output, "OUTPUT" blinks in the lower-right part of the screen.
12. Once complete, display the Data Server Storage screen. The program should now be listed there.

Section #b Transfer a Part Program between Data Server Storage and the Host

For Series 0iMC

1. Display the Hard Disk File Dir screen.
2. Press soft key [(OPRT)].
3. Press the continuation menu key at the right end of the soft key menu.
4. Press soft key [PUT].
5. Enter the following items using the MDI keys: number or name of the file on the transfer source Fast data server built-in ATA card, and name of the file on the transfer destination host computer built-in hard disk.
 - a.[Input format] Use one of the following:
 - i.<transfer-source-file-number>, <transfer-destination-filename> or
 - ii.<transfer-source-file-name>, <transfer-destination-filename> or
 - iii.<transfer-source-file-number> or
 - iv.<transfer-source-file-name>
6. Press soft key [EXEC].
7. During PUT operation, "PUT" blinks in the lower-right part of the screen.
8. To stop a PUT operation, press soft key [STOP].
9. Once complete, display the Host Directory screen. The program should now be listed there.

Section #2c Read/Punch Other CNC Data to the Host

Any CNC data that can normally be Read/Punched through the CNC RS-232 can now be read and punched through the Data Server connection to the host.

1. Verify that parameters 20, 21, 22, and 23 are set correctly for the CNC. (5 for 0iMC, 14 for 15i)
2. Execute a normal Read or Punch like you would if you were sending the data out the RS-232 port. Each set of data will be given a unique filename on the Host PC.

15.0

APPENDIX

15.1 M CODE LIST

Table 14-1: M Code

M CODE	DESCRIPTION	
M00	Program Stop	See unit III, Section 11.1 of 0i-MC Operator's Manual
M01	Optional Program Stop	
M02	End of Program	
M30	End of Program	
M98	Sub-Program Call	
M99	Sub-Program End	
M03	Spindle Start Forward	
M04	Spindle Start Reverse	
M05	Spindle Stop	
M06	Run Tool Change Cycle	
M07	Coolant ON - Mist Coolant/Coolant thru Spindle	Mist Coolant (Coolant 2) Receptacle
M08	Coolant ON - Flood Coolant	
M09	Coolant OFF	
M11	Set Tool Carousel position to 1 (ATC Tool Changer ONLY)	Need to take off the tool from spindle and cycle the CNC power
M19	FANUC Canned Cycle Positioning, (Not for Spindle Orientation)	Canned Cycles, Spindle Orients normal to X/Y/ axes
M20	Wash Down, Toggle ON/OFF	When K15.7=1
M20	MIDACO Pallet Changer 1	When K15.7=0, K15.2=1
M20	does nothing	When K15.7=0, K15.1=0
M22	Chip Conveyor Toggle ON/OFF	Set K15.6=1 when activating Conveyor by M22
M29	Rigid Tap	M29 Sxxxx in block prior to G84
M41	Low Gear Select	
M42	High Gear Select	
M48	100% Spindle Speed Override Forced	Spindle Speed will override selection
M49	100% Spindle Speed Override Released	
M60	A Axis Brake ON	
M61	A Axis Brake OFF	

Table 14-1: (Continued) M Code

M CODE	DESCRIPTION	
M62	B Axis Brake ON	
M63	B Axis Brake OFF	
M64	M64/65 Output ON	
M65	M64/65 Output OFF	
M66	M66/67 Output ON, Chip Conveyor ON	When K15.6=0
M67	M66/67 Output OFF, Chip Conveyor OFF	When K15.6=0
M68	M68/69 Output ON, Wash Down ON	When K15.7=0
M69	M68/69 Output OFF, Wash Down OFF	When K15.7=0
M78		
M79		
M80	MIDACO Pallet Changer 2	
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	Positioning Tool Drum to Pocket by T-Word Ex: "M100 T05"	
M190	Spindle Orient for Tool Changer	Orient tool to angle required for Tool Change.

15.2 PLC ALARM LIST

Table 14-2: PLC Alarms

ALARM	DESCRIPTION
1002	DUAL DOOR SWITCH FAILURE
1003	M02 POWER SAVE, RESET TO CLEAR
1005	DOOR OPEN, MDI TOOL CHG ALRM
1016	TOOL LIFE END REPLACE TOOL AND RESET ON TOOL LIFE SCREEN
1040	CAN NOT CHANGE TOOL WHEN SPINDLE IS RUNNNG
1041	CAN NOT CHANGE TOOL WHEN Z IS NOT IN HOME POSITION
1042	CAN NOT CHANGE TOOL WHEN SPINDEL IS NOT ORIENTED
1050	INIT SHIFT FAIL, CYC STRT INHIBT
1051	LOST LOW GEAR DURING OPERATION
1052	LOST HIGH 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 DRUM 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'T EXTEND DURING SP-OP
1104	ATC TOOL COUNT SWITCH MISSING
1105	INHIBIT SPINDLE CMM, ATC IS NOT AT HOME POS.
1106	ATC Z AXIS IS NOT AT REF.POSTION PLEASE REFERENCE Z AXIS

Table 14-2: (Continued) PLC Alarms

ALARM	DESCRIPTION
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 AXS BRAKE ON, COMMAND HALTED
1132	5TH AXS BRAKE ON, COMMAND HALTED
1134	INCH METRIC CONVERSIO ERROR CHANGE MODE IN SETTINGS SCREEN
1143	CYCLE CNC POWER
1145	DOOR PARAMETER WRITE ALARM
1147	ILLEGAL TOOL SEE MESSAGE
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

Table 14-2: (Continued) PLC Alarms

ALARM	DESCRIPTION
1172	M84 LOW GEAR SHIFT ERROR
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 HIGR CALIB DATA ZERO ERROR
1191	M84 HIGR SPDL SPEED CALC ERROR
1192	M84 HIGH GEAR SHIFT ERROR
1193	M84 HIGR WINDR READ SAMPLE ERROR
1194	M84 HIGR ARBITRARY DATA ERROR
1195	M84 HIGR REV COUNT WINDR ERROR
1196	M84 HIGR FDBK COUNT WINDR ERROR
1197	M84 HIGR DATA OUT OF RANGE ERROR
1200	M84 HIGR DISP RATIO MATH ERROR
1201	M84 HIGR RATIO RANGE ERROR
1202	M84 HIGR WINDW PAR SET ERROR 1
1203	M84 HIGR WINDW PAR SET ERROR 2
1204	M84 HIGR CALIB SPDL START ERROR
1205	NEED TO REFERENCE ALL AXES FIRST
1206	DOOR OPEN/CLOSE REQUIRED
1210	M84 #3741 CALC ERROR
1211	M84 #3742 CALC ERROR
1212	M84 MAX LOW WINDW ERROR

Table 14-2: (Continued) PLC Alarms

ALARM	DESCRIPTION
1213	M84 MAX HI WINDW ERROR
1214	M84 #3751 CALC ERROR
1215	M84 SHIFT SPD WINDW ERROR
1216	M84 #3736 CALC ERROR
1217	M84 #3736 WINDW ERROR
1220	M84 #3752 WINDW ERROR
1221	M84 #3762 WINDW ERROR

15.3 PLC MESSAGES

Table 14-3: PLC Messages

MESSAGE	DESCRIPTION
2004	OPERATOR FUNCTION INHIBITED CHECK DOOR ALM OPERATIONAL MODE
2005	MACHINE DOOR OPEN. FEEDHOLD 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 PB 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
2014	EXIT PROGRAM RESTART BEFORE CYCLE START
2015	DOOR UNLOCK INHIBITED
2016	TOOL CAGE DOOR IS OPEN, PLEASE CLOSE IT. FEED HOLD IS ON
2019	RESET E-STOP WITH ALARM PB.
2020	LOW LUBE OIL LEVEL REFILL VACTRA-2 OR EQUIVALENT
2021	TOOL LIFE NEAR END OR HAS EXPIRED
2023	TOOL DATA SWAP ERROR CHECK TOOL LOCATION
2024	LOW PROBE BATTERY
2026	AIR-OIL FAULT, ILLEGAL TABLE DATA FOR AIR-OIL INTERVALS

Table 14-3: PLC Messages

MESSAGE	DESCRIPTION
2027	AIR-OIL FAULT FEEDHOLD/SPINDLE STOP FORCED.SEE OTHER MESSAGES
2030	AIR-OIL LOW OIL PRESSURE FAULT
2031	AIR-OIL UPPER BEARING PRESSURE FAILED FAULT
2032	AIR-OIL LOWER BEARING PRESSURE FAILED FAULT
2033	AIR-OIL HIGH OIL PRESSURE WHILE PUMP TURNED OFF
2034	AIR-OIL FAULT. SAFELY STOP MACHINE, OR STOP WILL BE FORCED
2035	AIR-OIL SPINDLE AIR/VAC PRESSURE FAILED FAULT
2036	AIR-OIL OIL LEVEL LOW PLEASE REFILL RESEVOIR
2037	CORRECT FAULT & USE ALARM-MSG PBTO RESET ALL AIR-OIL MESSAGE'S
2044	TOOL ARM OVERLOAD TRIP MAINTENANCE MUST RESET OVERLOAD
2045	TOOL MAGAZINE OVERLOAD TRIP 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 ALRM 1050 CHECK AIR PRESSURE 80-90 PSI
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 POSN
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 SPNDLE
2064	DRAWBAR NOT RELEASED FOR REMOVAL OF TOOL FROM SPNDLE
2065	TOOL ARM NOT IN POSITION FOR TOOL INSERTION INTO SPNDLE
2066	DRAWBAR NOT CLAMPED ON NEW TOOL IN SPNDLE
2067	TOOL ARM NOT RETURNED TO PARK POSN AT END OF TOOL CHANGE
2068	USING "FUNC+CYCLE START" BUTTON TO RESTART MAGAZINE

Table 14-3: PLC Messages

MESSAGE	DESCRIPTION
2070	TOOL CHANGE COMMAND ABORTED TOOL CHANGE SERVICE MODE ACTIVE
2071	TOOL CHANGE COMMAND ABORTED SPINDLE FWD, REV, ORIENT ACTIVE
2072	ARM NOT AT HOME POSITION
2073	ARM AT SPINDLE
2074	Z MOVE TO THE PROHIBIT AREA
2075	TOOL CHNG & SPNDL, E-STOP FORCED ALARM PB TO CLEAR
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
2082	TOOL DRUM AT POCKET # OF # PRESS DRUM PSN TO CLEAR MSG
2087	STORED TOOL VALUE OF # IS NOT IN LEGAL RANGE OF 1 TO #(maximum tool number)
2100	NO TOOL CHANGER SELECTED CORRECT K5.X & CYCLE POWER
2101	ILLEGAL VMC TOOL CHANGR SETTING CORRECT K5.X & CYCLE POWER
2102	VMC & HMC TOOL CHANGER BOTH SET CORRECT K5.X & CYCLE POWER
2127	ORIENTATION INHIBIT, SLIDE EXTENDED
2130	LOW COOLANT LEVEL OR PUMP FAILURE
2131	USE M61 TO RELEASE 4TH AX BRAKE PRIOR TO AXIS COMMAND BLOCK
2133	USE M62 TO RELEASE 5TH AX 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	AXIS INHIBIT BY MIDACO PALLET CHANGER
2142	DOOR IS CLOSED. PRESS ALARM PB TO RESET
2150	ATC IS NOT AT HOME POSITION USE M186 RETURN;Z INHIBITED
2151	NEW TOOL VALUE OF # IS NOT IN LEGAL RANGE OF #(minimum tool value) TO (maximum tool value).
2165	PRESS UNLOCK BUTTON FOR 60S TO UNLOCK DOOR
2187	M84 BELT CALIBRATION ACTIVE

Table 14-3: PLC Messages

MESSAGE	DESCRIPTION
2222	IO OUTPUT OVERCURRENT
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

15.4 G CODES

For the G-Codes list refer to GE Fanuc Operator's Manual B-64124EN_01, II Programming, sections 3. Preparatory Function (G Function), Table 3 G Code List.

15.5 FINAL BACK UP PROCEDURE

1. Reference all the axes.
 - a). Press "REF RETURN" pushbutton on the operator panel .
 - b). Press "Z+" pushbutton on the operator panel (Manual Z axis zero return).
 - c). Press "X+" pushbutton on the operator panel (Manual X axis zero return).
 - d). Press "Y+" pushbutton on the operator panel (Manual Y axis zero return).
 - e). CNC moved X/Y/Z axis to zero position. CNC screen will display X0Y0Z0.
2. Use flash card.
 - a). Prepare a blank flash card to use.
 - b). Smoothly put the COMPACT FLASH CARD inside the M-CARD slot of LCD
3. Backup all the programs of CNC memory.
 - a). Press "SYSTEM" pushbutton on pendant.
 - b). Press "EDIT" pushbutton on pendant.
 - c). Press softkey "+" twice.
 - d). Press softkey "ALL IO".
 - e). Press softkey "PRGRM".
 - f). Press softkey "OPRT".
 - g). Press softkey "PUNCH".
 - h). Type "-9999"
 - i). Press "O SET" softkey.

- j). Press softkey "EXEC" (OUTPUT will be displayed on the screen, all the programs will output to the Flash Card as PROGRAM.ALL)
4. Backup CNC parameters.
- a). Press "SYSTEM" pushbutton on pendant.
 - b). Press "EDIT" pushbutton on pendant.
 - c). Press softkey "+" twice.
 - d). Press softkey "ALL IO".
 - e). Press softkey "PARAM".
 - f). Press softkey "OPRT".
 - g). Press softkey "PUNCH".
 - h). Press softkey "EXEC". (OUTPUT will be display on the screen, all the parameter will output to the Flash Card as "CNCPARAM.DAT")
5. Backup CNC offset.
- a). Press "SYSTEM" pushbutton on pendant.
 - b). Press "EDIT" pushbutton on pendant.
 - c). Press softkey "+" twice.
 - d). Press softkey "ALL IO".
 - e). Press softkey "OFFSET".
 - f). Press softkey "OPRT".
 - g). Press softkey "PUNCH".
 - h). Press softkey "EXEC" (OUTPUT will be display on the screen, all the offsets will output to the Flash Card as "TOOLOFST.DAT")
6. Backup CNC macros.
- a). Press "SYSTEM" pushbutton on pendant.
 - b). Press "EDIT" pushbutton on pendant.
 - c). Press softkey "+" twice.
 - d). Press softkey "ALL IO".
 - e). Press softkey "+".
 - f). Press softkey "MACRO".
 - g). Press softkey "OPRT".

- h). Press softkey "PUNCH".
 - i). Press softkey "EXEC" (OUTPUT will be display on the screen, all the macros will output to the Flash Card as "MACROVAR.DAT")
7. Backup CNC pitch.
- a). Press "SYSTEM" pushbutton on pendant.
 - b). Press "EDIT" pushbutton on pendant.
 - c). Press softkey "+" twice.
 - d). Press softkey "ALL IO".
 - e). Press softkey "+".
 - f). Press softkey "PITCH".
 - g). Press softkey "OPRT".
 - h). Press softkey "PUNCH".
 - i). Press softkey "EXEC" (OUTPUT will be display on the screen, all the pitch compensation data will output to the Flash Card as "PITCHERR.DAT")
8. Backup CNC work offsets.
- a). Press "SYSTEM" pushbutton on pendant.
 - b). Press "EDIT" pushbutton on pendant.
 - c). Press softkey "+" twice.
 - d). Press softkey "ALL IO".
 - e). Press softkey "+".
 - f). Press softkey "WORK".
 - g). Press softkey "OPRT".
 - h). Press softkey "PUNCH".
 - i). Press softkey "EXEC" (OUTPUT will be display on the screen, all the work offsets will output to the Flash Card as "WORK-G54.DAT")
9. Backup the PLC software
- a). Press "EDIT" pushbutton to CNC control to edit mode.
 - b). Press "SYSTEM" pushbutton on the MDI panel until parameter screen comes out.
 - c). Press softkey "PMC" .
 - d). Press softkey "I/O" .

- e). Move the cursor and select the contents of the screen as following (move the cursor, and use the softkey to select the correct data type)

DEVICE= M-CARD

FUNCTION= WRITE

DATA KIND= LADDER

FILE NO.= @PMC-SB.000

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

10. Backup the PLC parameter

- a). Press "EDIT" pushbutton to CNC control to edit mode.
- b). Press "SYSTEM" pushbutton on the MDI panel until parameter screen comes out.
- c). Press softkey "PMC" .
- d). Press softkey "I/O" .
- e). Move the cursor and select the contents of the screen as following (move the cursor, and use the softkey to select the correct data type)

DEVICE= M-CARD

FUNCTION= WRITE

DATA KIND= PARAM

FILE NO.= @PMC-SB.PRM

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

11. Backup SRAM file.

- a). Power off the machine.
- b). Press the most right side two softkey on the LCD screen and hold
- c). Press the CNC power on pushbutton.
- d). Wait until the CNC system boot screen comes out.
- e). Press the "DOWN" softkey to move the cursor to the item 5 (is highlighted)

5. System Data Backup

- f). Press "SELECT" softkey will display two choices

"1. SRAM BACKUP(CNC 'Flash Card)"

"2....."

- g). Make sure the cursor is in the first line.
- h). Press "SELECT" softkey and you will be asked "Backup SRAM Data OK?"
- i). Select "YES" to continue. (or "NO" to return to previous menu.)
- j). Wait until the screen displays completed information.
- k). Press softkey "SELECT".
- l). Press the "DOWN" softkey until the cursor moves to item : END
- m). Press softkey "SELECT".
- n). Press the "DOWN" softkey until the cursor moves to item : END
- o). Press softkey "SELECT".
- p). Press softkey "YES".
- q). Finish, the CNC system will boot up.

12. Copy all the backup files form your flash card to your PC or storage disk.

**15.6 COLD START
POSITION SETUP
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" code to change machine to Metric mode.
 - a). Press the "MDI" pushbutton on the operator panel A.
 - b). Press "PROGRAM" pushbutton one/two times on the MDI panel until PROGRAM (MDI) screen is displayed.
 - c). Press "G21" using MDI panel.
 - d). Press "EOB" pushbutton on the MDI panel.
 - e). Press "INSERT" pushbutton.
 - f). Press "CYCLE START" pushbutton on the operator panel .
 - g). Make sure the screen is changed to Metric Mode (X***,*** is displayed).
2. Reference all the axes.

- a). Press "REF RETURN" pushbutton on the operator panel .
 - b). Press "Z+" pushbutton on the operator panel (Manual Z axis zero return).
 - c). Press "X+" pushbutton on the operator panel (Manual X axis zero return).
 - d). Press "Y+" pushbutton on the operator panel (Manual Y axis zero return).
 - e). CNC moved X/Y/Z axis to zero position. CNC screen will display X0Y0Z0.
3. Load a tool to spindle.
- a). Press "JOG" pushbutton on the operator panel .
 - b). Press "FUNC" + "TOOL REL" pushbutton together on the operator panel and keep it. (Draw Bar will be released)
 - c). Put the tool inside of spindle nose.
 - d). Release the "FUNC" + "TOOL REL" pushbutton.
 - e). Make sure that tool is loaded into spindle.
4. Find out X axis zero position manually and makes the cold start.
- a). Press "HANDLE X100" or "HANDLE X10" or "HANDLE X1" to select MPG mode and right resolution of MPG.
 - b). Press "X+" pushbutton to select to select X axis.
 - c). Use TAPE MEASURE find out the X axis middle point of the table, and make a mark.
 - d). Use MPG move the X axis to the X axis middle position mark.
 - e). Record the X coordinate displayed on the CNC screen as "A".
 - f). Calculate the data "B" as following formula $B=(A-(A/10))*10000$
 - g). Press "MDI" pushbutton on the operator panel.
 - h). Press "SYSTEM" pushbutton on the MDI panel.
 - i). Press "PARAM" softkey below the LCD screen.
 - j). Type "1850" using MDI panel.
 - k). Press "NO.SRH" softkey below the LCD screen (Parameter 1850 will be displayed).
 - l). Move the cursor to X axis section of parameter No.1850.
 - m). Type data B from MDI panel, example 34560 (3.456mm).
 - n). Press "INPUT" pushbutton on the MDI panel (34560 will be displayed on the parameter 1850X column)

*****Alarm 000 will pop up on the alarm window.**

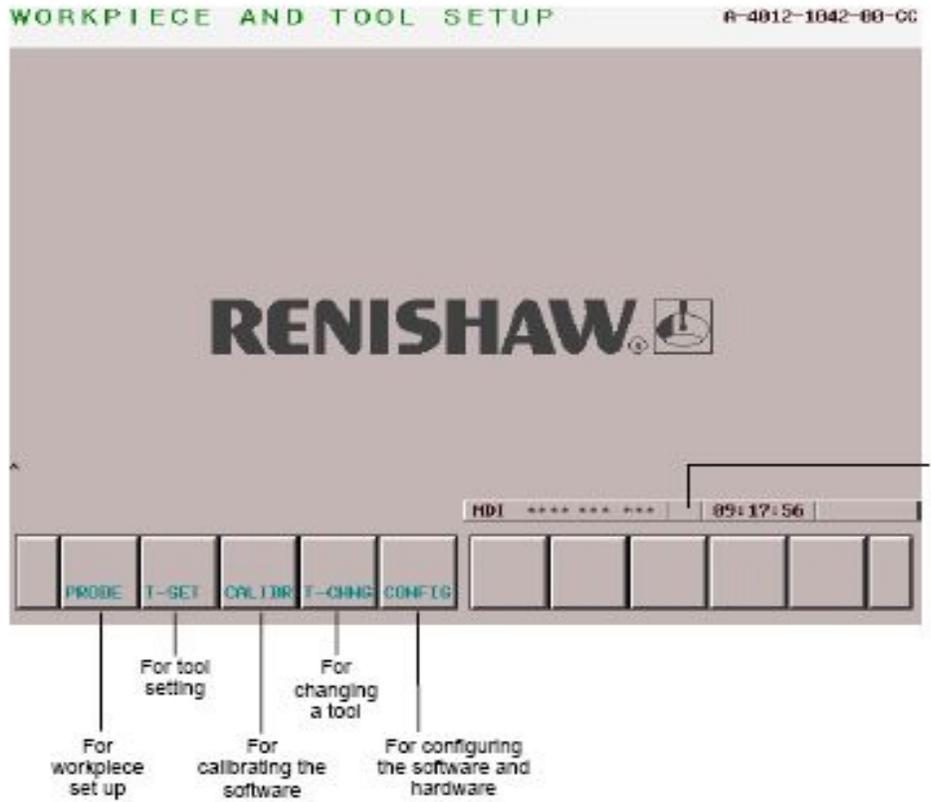
- o). Press "Power OFF" button to power off the CNC (Press [O] button on the operator panel).
- p). Press "Power ON" button to Power ON the CNC (Press [I] button on the operator panel B).

*****Alarm 300X will be displayed on the screen (it is normal).**

- q). Press "JOG" pushbutton on the operator panel.
 - r). Press "X-" pushbutton on the operator panel to move the X axis 25.0mm or more.
 - s). Press "X+" plus pushbutton to move the X axis middle position to the position that is very close to the physical reference mark.
 - t). Press "REF RETURN" pushbutton.
 - u). Press "X+" pushbutton. (X axis will automatically move to the zero position)
 - v). In case if zero position is wrong, repeat the previous steps starting with step 4/e).
 - w). Double check the cold start position using tape measure.
 - x). Press "RESET" pushbutton on the MDI panel. That will reset the 300 alarm.
5. Find out Y axis zero position manually, make the cold start.
 - a). Install the indicator on the spindle.
 - b). Repeat the same procedure as 4/a). - 4/x). to find out the Y axis cold start position.
 6. Find out Z axis zero position manually, make the cold start.
 - a). Install tool on the DATC arm (DATC) or ATC carousel (ATC).
 - b). Measure the height of the tool.
 - c). Install the tool on the spindle.
 - d). Use MPG move the Z axis to the same height as the tool on the DATC ARM/ ATC carousel, measure the same point of the tool.
 - e). Repeat the same procedure as 4/e). - 4/x). to find out the Z axis cold start position.
 7. Repeat the same procedure for A axis id it needs.
 8. Cold Start setup is finished.

15.7 PROBE

For further instruction please view PDF attachment H-2000-6430-00-A from Renishaw



INDEX

0-9

4TH (A) AXIS AMP FITTED 71

4TH (A) AXIS FITTED 71

A

A- AXIS 47

A+ AXIS 47

ACTION 80

ADD A TOOL 121

ADD MORE TOOLS 122

AICC FUNCTION 111

AICC PRECISION LEVEL DATA SCREEN 114

AIR PRESSURE 2

ALARM MESSAGE 39

APPENDIX 141

AUTO COOL 138

AUTO MODE 26

AUTO TOOL CHANGER 72

AXES SET UP 71

AXIS DIRECTION SELECTION 56

B

BECON 23

BLOCK DELETE 40

C

CALCULATOR 99

CIRCLE SOLVER 101

CNC CONTROL 7

COLD START POSITION SETUP PROCEDURE 154

CONTROL FUNCTION 25

CONTROL LAYOUT 8

COOLANT CONTROL 137

COOLANT THROUGH SPINDLE (OPTIONAL) 138

COPYING OFFSETS 92

CYCLE START 62

D

DATA I/O 133

DATA I/O FUNCTION 134

DATUM 77

DATUM EDGE LOCATION 88

DATUM SETTING 87
DELETE A TOOL 123
DELETE ALL TOOLS 124
DOOR OVERRIDE 39
DRUM FWD 43
DRUM REV 43
DRY RUN 41

E

EDIT MODE 28
ELECTRICAL CABINET DOOR 5
EMERGENCY STOP 39
EXECUTION FUNCTION 62

F

FEED CONTROL 109
FEED HOLD 63
FEED/RAPID/JOG OVERRIDE 61
FINAL BACK UP PROCEDURE 150
FLOOD COOLANT 138
FLOOD COOLANT 3
FUNCTION KEY 42

G

G CODES 150
G92 POSITION SET 94
G92 POSITION SET CANCEL 97
GE FANUC LCD 9
GE FANUC MANUAL DATA INPUT 9
GLOBAL DATUM SHIFT 93

H

HANDLE 62
HANDLE MODE 38

J

JOG MODE 37

L

LENGTH SETTING 81
LIMIT 78

M

M CODE / S CODE 58
M CODE LIST 142
M/C SET UP 65
MANUAL GUIDE I 127

MANUAL GUIDE I FUNCTION 128
MANUAL PULSE GENERATOR 23
MANUAL PULSE GENERATOR 62
MANUAL SPINDLE SPEED 48
MC LOCK 41
MDI MODE 29
MIST COOL 138

O

OIL RESERVOIR 2
OPERATION CONTROL 61
OPERATION MODE SELECTION 26
OPERATOR PANEL A 10
OPERATOR PANEL B 21
OPTION STOP 40

P

PCMCIA CARD PORT 23
PLC ALARM LIST 144
PLC MESSAGES 147
POWER OFF 5
POWER ON 4
POWER ON/OFF 1
POWER ON/OFF 4
POWER SET 74
PRE-START CHECKING STEPS 2
PROGRAM COORDINATE UNITS 68
PROGRAM RESTART 40
PROGRAMMING CONSIDERATIONS 112

Q

QUICK CUT 107

R

RAPID OVERRIDE 57
REF RETURN MODE 35
REMOTE MODE 31
RESET "TOOL LIFE EXPIRED" STATUS FOR ALL TOOLS 125
RESET "TOOL LIFE EXPIRED" STATUS FOR THE TOOL 124
RS 232 23

S

SAFETY FUNCTIONS 39
SETTING TOOL ORIGIN POINT 82
SETTING TOOL LENGTH 84
SETUP MODES 42
SINGLE BLOCK 40

SOFTKEYS 9
SPEED AND FEED CALCULATOR 102
SPINDLE CLOCKWISE 58
SPINDLE CONTROL 109
SPINDLE COUNTER-CLOCKWISE 58
SPINDLE OPERATION 58
SPINDLE ORIENTATION 60
SPINDLE OVERRIDE 60
SPINDLE SPEED OVERRIDE 60
SPINDLE SPEED RANGE 59
SPINDLE STOP 58

T

T/H/D ALIGNMENT CHECK 69
T-COMMAND (TOOL NUMBER) 44
TEST FUNCTION MODES 40
TOOL CHANGE ADDS D/H OFFSETS 70
TOOL CHANGE TYPE 72
TOOL CHANGER COMMAND 45
TOOL DIAMETER/RADIUS SETTING 85
TOOL LENGTH DIAMETER/RADIUS ADJUSTMENT 86
TOOL LIFE LIMIT SETTING 122
TOOL LIFE MANAGEMENT 118
TOOL LIFE MANAGEMENT ACCESS 121
TOOL LOAD MONITOR 73
TOOL MANAGEMENT 117
TOOL OPERATION 43
TOOL REL 43
TOOL SKIP 126
TOOL TYPE 76
TRIANGLE SOLVER 100

U

USER PREFERENCE 67

W

WATER RESERVOIR 3
WORKLIGHT 39

X

X- AXIS 56
X - AXIS LOCATION 88
X+ AXIS 56

Y

Y - AXIS LOCATION 90
Y- AXIS 57

Y+ AXIS 56

Z

Z - AXIS LOCATION 91

Z - AXIS 57

Z+ AXIS 57

RENISHAW

Workpiece and tool set up software for Fanuc controllers with GUI

Draft No. 4 – circulated 8/05/06

**for Fanuc controls 0i, 16i, 18i,
20i, 21i, 30i and 300i series**

© 2006 Renishaw plc. All rights reserved.

This document may not be copied or reproduced in whole or in part, or transferred to any other media or language, by any means, without the prior written permission of Renishaw.

The publication of material within this document does not imply freedom from the patent rights of Renishaw plc.

Disclaimer

Considerable effort has been made to ensure that the contents of this document are free from inaccuracies and omissions. However, Renishaw makes no warranties with respect to the contents of this document and specifically disclaims any implied warranties. Renishaw reserves the right to make changes to this document and to the product described herein without obligation to notify any person of such changes.

Trademarks

RENISHAW® and the probe emblem used in the RENISHAW logo are registered trademarks of Renishaw plc in the UK and other countries.

apply innovation is a trademark of Renishaw plc.

All other brand names and product names used in this document are trade names, service marks, trademarks, or registered trademarks of their respective owners.

Contents

Introduction	3
Software cycles.....	3
Before using Renishaw software cycles	3
Starting the Renishaw software	4
Using the software: an overview	4
Selecting a screen option	4
Moving between screens	4
Entering a cycle input value.....	5
Moving between cycle inputs	5
Using a typical cycle	5
Calibrating a probe's stylus.....	7
When to calibrate an inspection probe	7
When to calibrate a TS27R tool setting probe	7
Selecting a calibration cycle	8
Calibrating an inspection probe's stylus	9
Calibrating a tool setting probe's stylus	9
Setting a workpiece	10
Selecting a workpiece set up cycle.....	10
Bore setting	10
Boss setting	10
Web setting.....	10
Pocket setting	11
Edge setting.....	11
Z surface setting	11
Corner setting	11
Levelling a 4th axis (when fitted)	12
Tool setting	12
Selecting a tool setting cycle	12
Tool length setting	13
Tool length and diameter setting	13
Sequential tool length setting.....	14
Random tool length setting	14
Broken tool detection (manual and automatic)	14
Changing a tool.....	14
Configuring the software (when access is permitted)	15
Selecting the software configuration screen	15
Configuring controller settings	15

Configuring inspection probe settings.....	16
Configuring TS27R tool setting probe settings	17
Inputs	19
Inputs for workpiece set up cycles	19
Inputs for tool setting cycles	20
Inputs for inspection probe calibration cycles	21
Inputs for TS27R tool setting probe calibration cycles	21
Workpiece set up NC macro variables.....	21
Calibration cycle variables	21
Cycle output variables	22
Alarms and error messages.....	23
Additional TS27R messages.....	24

Introduction

This Renishaw software is designed for running on Fanuc controls that use a graphical user interface (GUI). It provides you with a quick and effective way to carry out many of the workpiece and tool setting set up tasks that are commonly performed on machine tools. The cycles are based on those provided by current Renishaw software.

Fanuc controls on which the software runs are as follows:

- 0i, 16i, 18i, 20i, 21i, 30i and 300i series.

Software cycles

The following cycles are provided by the software:

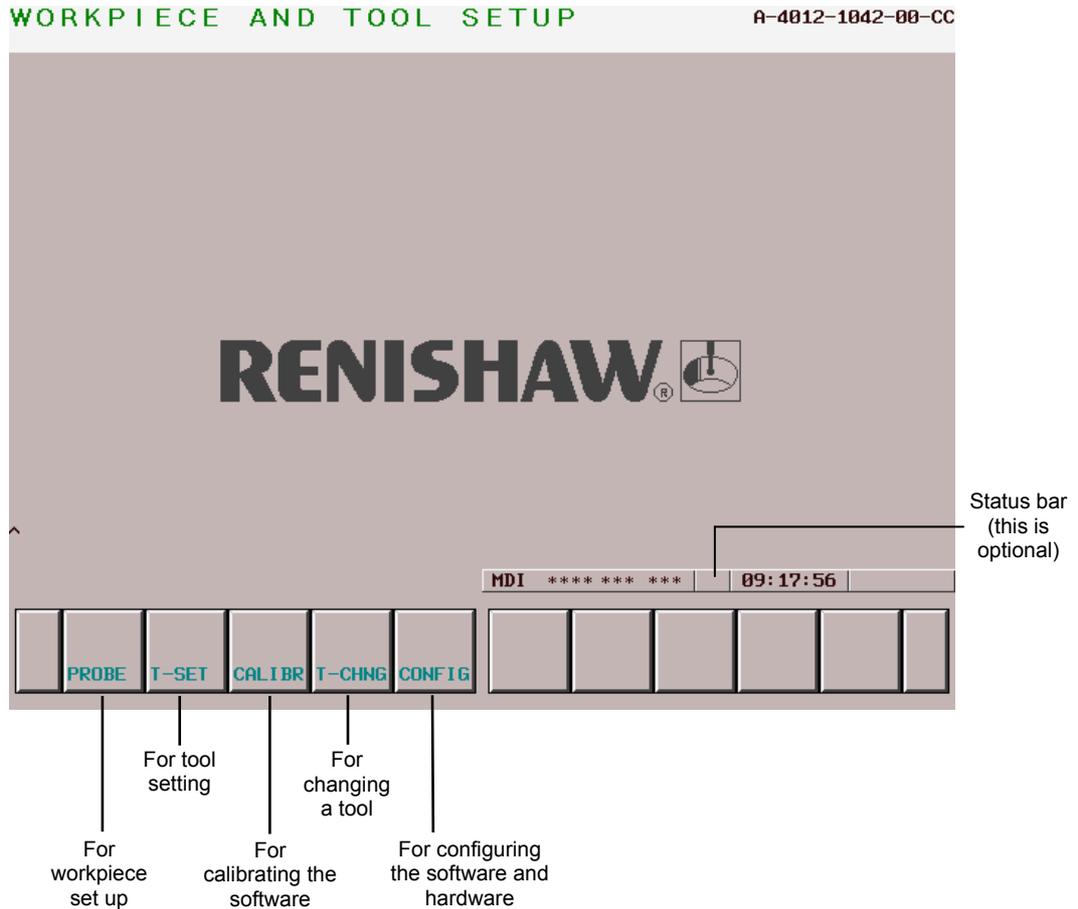
- Workpiece set up cycles. For a description of how to select and use these cycles, see “Setting a workpiece” on page 10.
- Tool setting cycles. For a description of how to select and use these cycles, see “Tool setting” on page 12.
- A tool change cycle. For a description of how to select and use this cycle, see “Changing a tool” on page 15.
- Probe stylus calibration cycles. For a description of how to select and use these cycles, see “Calibrating a probe’s stylus” on page 7.
- Software configuration options. These are used to configure the controller, inspection probe and Renishaw TS27R tool setting probe. For a description of how to select and use these options, see “Configuring the software” on page 16.

Before using Renishaw software cycles

Before using any of the cycles it is important that the software has been correctly prepared and configured for the machine. For details of configuration settings, refer to “Configuring the software” on page 16.

Starting the Renishaw software

- ▶ To start the Renishaw software, press the CUSTOM 1 button on the control. The Renishaw splash screen is displayed, as shown below. Note that your control might have been configured to use a different button.



Using the software: an overview

Selecting a screen option

- ▶ To choose a screen option, press the soft key associated with the option.

Moving between screens

- ▶ To exit a screen and move to the previous screen, press the ◀ soft key.
- ▶ To move to the next screen, press the ▶ soft key.

Entering a cycle input value

- ▶ To enter a value for the inputs of most cycles, use the control's numeric keypad.
- ▶ To enter a value for some of the inputs of the configuration cycles, repeated press the SELECT soft key that will be displayed.

Moving between cycle inputs

- ▶ After you enter the value of a cycle input the cursor moves to the next input in the sequence. To move between the inputs in a random order, use the control's cursor keys.

NOTE: If you press ◀ to exit the screen after you have entered the input values, the values will be lost.

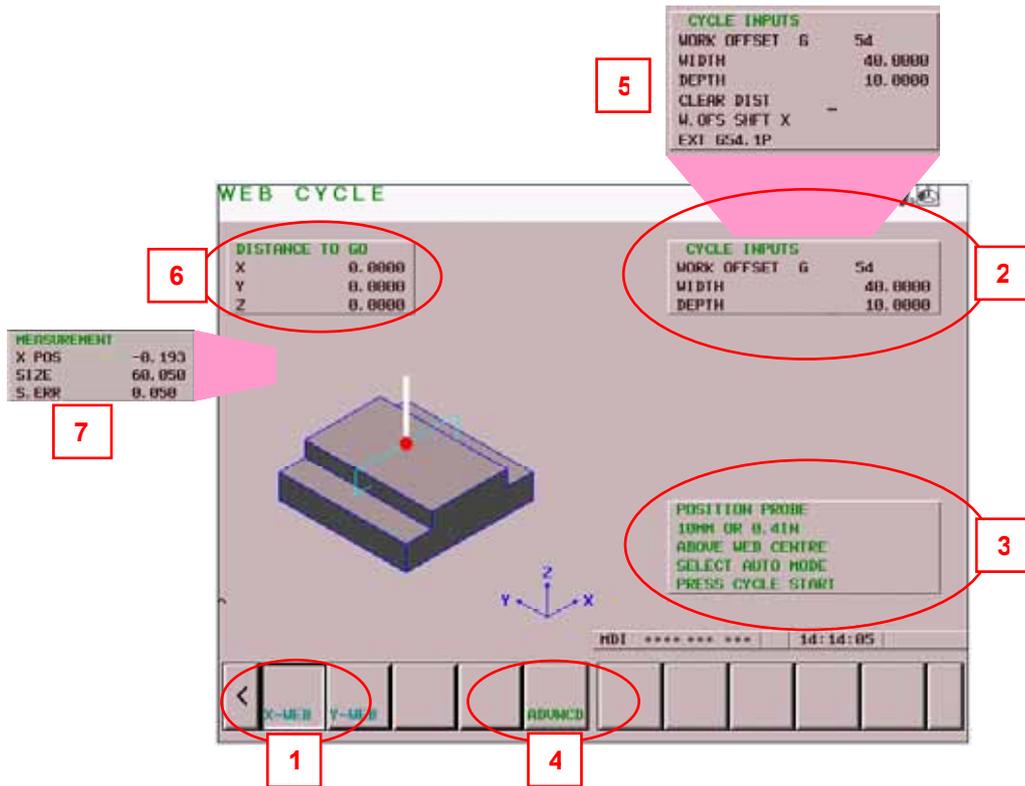
Using a typical cycle

This topic describes how to select a typical cycle (the web set up cycle in this example), enter the input values then run the cycle. All software cycles are used in a similar way.

Selecting the cycle



1. From the Renishaw splash screen (shown on the previous page), press PROBE. The WORKPIECE SETUP screen is displayed.
2. From the WORKPIECE SETUP screen, press WEB. The WEB CYCLE screen is displayed.



Entering inputs and running the cycle

1. To choose a cycle option, press the required soft key ①.
2. Use the control's keypad to enter the values of compulsory inputs. They are shown in the CYCLE INPUTS box ②.

For further information about the inputs, both compulsory and optional, for workpiece set up, tool setting and calibration cycles, see "Inputs" on page 20.

3. After you have entered all compulsory input values, a message box that contains operating instructions is displayed ③. A cycle cannot run until this message appears.

When the cycle supports additional optional inputs, an ADVNC'D (advanced options) soft key ④ is also displayed.

You can now choose to either enter advanced optional inputs, when supported by the cycle, or run the cycle.

4. To enter the additional optional inputs, press ADVNC'D ④.

Enter the values of the optional inputs. They are shown in an expanded CYCLE INPUTS box ⑤.

5. To run the cycle, select AUTO mode on the control then press CYCLE START.
Refer to the DISTANCE TO GO box ⑥ to see live axis values.
Output variables that are set by the cycle are described in the section “Cycle output variables” on page 23.
6. When the cycle finishes, a MEASUREMENT pop-up box ⑦ is displayed.
To display the previous screen, press ◀.

Calibrating a probe's stylus

Before a probe is used it is important that the stylus is calibrated correctly. Two calibration cycles are provided for calibrating the stylus of an inspection probe and one cycle for calibrating a TS27R tool setting probe.

When to calibrate an inspection probe

After you have fitted an inspection probe into its machine shank/holder it is good practice to adjust the probe's stylus so that it is true to the spindle centre-line. This helps reduce the effects of spindle and tool orientation errors. However, a small amount of run-out can be tolerated.

Calibrating the probe ensures that run-out is automatically accounted for. If you do not calibrate the probe you will get inaccurate results.

It is important that you calibrate a Renishaw inspection probe in the following circumstances:

- When a probe is to be used for the first time.
- Whenever a new stylus is fitted to the probe.
- When it is suspected that the stylus has become distorted or that the probe has crashed.
- At regular intervals to compensate for mechanical changes of your machine tool.
- If, following a tool change, the repeatability of accurately relocating the probe shank is poor. In this case, the probe may need to be recalibrated each time it is selected.

When to calibrate a TS27R tool setting probe

After a TS27R tool setting probe has been mounted on the machine table, the stylus faces must be aligned with the machine axes to avoid probing errors when setting tools. Take care with this operation – for normal use you should try to get the faces aligned to within 0.010 mm (0.0004 in). Do this by manually aligning the stylus with the adjusting

screws, and using a suitable instrument such as a DTI clock mounted in the machine spindle.

After the probe has been correctly set up on the machine, it must be calibrated. This establishes the trigger point values of the probe stylus measuring faces under normal measuring conditions. The calibration values are stored in macro variables for computation of the tool size during tool setting cycles.

Values obtained are axis trigger positions (in machine co-ordinates). Any errors due to machine and probe triggering characteristics are automatically compensated for. These values are the electronic trigger positions under dynamic operating conditions and are not necessarily the true physical stylus face positions.

It is important that you calibrate a Renishaw TS27R probe in the following circumstances:

- When a probe is to be used for the first time.
- Whenever a new stylus is fitted to the probe.
- When it is suspected that the stylus has become distorted or that the probe has crashed.
- At regular intervals to compensate for mechanical changes of your machine tool.

Selecting a calibration cycle



1. From the Renishaw splash screen (shown on page 4), press the CALIBR soft key. The HARDWARE CALIBRATION screen is displayed.
2. To calibrate the length of an inspection probe stylus, press the PRBLEN soft key.
To calibrate the radii and stylus offsets of an inspection probe stylus, press the PRBRAD soft key.
To calibrate a TS27R tool setting probe stylus, press the TS27R soft key.
For further information about inputs for the calibration cycles, see "Inputs" on page 20.

The following sections each provide a brief description of how to use a calibration cycle. If you need more information about using a cycle, see "Using the software: an overview" on page 4.

Calibrating an inspection probe's stylus

Calibrating the length



This cycle is used to calibrate the length of the stylus of an inspection probe.

A reference feature having a known Z-axis position is used for calibrating. This may be the machine table surface or a reference surface on a component or fixture. Enter the probe tool offset used and the position of the Z surface.

Note that the Z REF POSITION is relative to the active work offset. The active work offset number (G54 etc.) is displayed on the screen so also check the Z-axis value of the work offset register.

The probe is positioned above the reference surface. When the cycle runs, the surface is measured and the probe tool offset is reset to this new value. The probe then returns to the start position.

Calibrating the radius and stylus offsets



This cycle is used to calibrate the radius (diameter) of the stylus ball and the offsets of the stylus in the X and Y axes.

A reference feature having a known size is used for calibrating. This may be a ring gauge or a suitable external cylinder feature.

When the machine has an M-code for 180 degrees spindle orientation positioning, the cycle will calibrate using this feature. This avoids the necessity to position the probe's spindle exactly on the feature centre-line before running the cycle.

The probe is positioned at a suitable start position, as shown in the onscreen graphic. Depending on the method used for calibrating, the probe's spindle may or may not need to be exactly on the feature centre-line with the spindle orientation active. When the cycle runs, complete calibration of the stylus radii, including the vector radii, and stylus offsets are established. The probe then returns to the start position.

Calibrating a tool setting probe's stylus



This cycle is used to fully calibrate the TS27R probe on the machine table to establish the trigger position of each of the stylus faces.

A master tool (reference arbor) having a known length and diameter is used for calibrating the probe. This must be fitted in the machine spindle.

The SELECT soft key is used to select the onscreen orientation that matches the orientation of your TS27R probe.

The master tool is positioned to a suitable start position approximately 10 mm (0.4 in) above the stylus as shown in the onscreen graphic. When the cycle runs, the stylus is fully calibrated.

Setting a workpiece

This section provides a brief description of how to select a workpiece set up cycle and then use each of the cycles. If you need more information about using a cycle, see “Using the software: an overview” on page 4.

Selecting a workpiece set up cycle



1. From the Renishaw splash screen (shown on page 4), press PROBE. The WORKPIECE SETUP screen is displayed.
2. From the WORKPIECE SETUP screen, press a soft key to select the cycle you require. The appropriate cycle screen is displayed.

Bore setting



The bore set up cycle measures a bore using four measuring moves in the X and Y axes.

Before running the cycle, the probe is positioned on the expected centre-line of the feature and at a suitable position in the Z-axis, as shown in the onscreen graphic. The probe and probe offset are to be active. When the cycle runs, the stylus measures the bore. The probe then returns to the start position.

Boss setting



The boss set up cycle measures a boss using four measuring moves in the X and Y axes.

Before running the cycle, the probe is positioned on the expected centre-line of the feature and at a suitable position in the Z-axis, as shown in the onscreen graphic. The probe and probe offset are to be active. When the cycle runs, the stylus measures the boss. The probe then returns to the start position.

Web setting



The web set up cycle measures a web using two measuring moves in either the X or Y axis.

Before running the cycle, the probe is positioned on the expected centre-line of the feature and at a suitable position in the Z-axis, as shown in the onscreen graphic. The

probe and probe offset are to be active. When the cycle runs, the stylus measures the web. The probe then returns to the start position.

Pocket setting



The pocket set up cycle measures a pocket using two measuring moves in either the X or Y axis.

Before running the cycle, the probe is positioned on the expected centre-line of the feature and at a suitable position in the Z-axis, as shown in the onscreen graphic. The probe and probe offset are to be active. When the cycle runs, the stylus measures the pocket. The probe then returns to the start position.

Edge setting



The edge set up cycle measures an edge in the X+, X-, Y+ or Y- axis to establish either its size or position.

Press the appropriate soft key to select the onscreen orientation that matches the orientation of the edge to be measured.

Before running the cycle, the probe is positioned adjacent to the edge and at a suitable position in the Z-axis, as shown in the onscreen graphic. The probe and tool offset are active. When the cycle runs, the stylus measures the surface. The probe then returns to the start position.

Z surface setting



The Z- set up cycle measures a surface to establish either its size or Z-axis position.

Before running the cycle, the probe is positioned adjacent to the surface and at a suitable position in the Z-axis, as shown in the onscreen graphic. The probe and tool offset are active. When the cycle runs, the stylus measures the surface. The probe then returns to the start position.

Corner setting



The corner set up cycle is used to establish the corner position of a feature. Both internal and external corners can be set. A true corner intersection can be found even if the corner is not 90 degrees.

The following options are available:

Using inputs D1 and D2 only: A single measuring point is taken on each surface to find the corner. This assumes the surfaces are square to each other and to the machine axis.

Using inputs D1, D2 and either I or J: Two measuring points are taken on one surface and one is taken on the other surface to find the corner. This assumes the surfaces are square to each other but not necessarily to the machine axis.

Using inputs D1, D2, I and J: Two measuring points are taken on each surface to find the corner. The true corner intersection point is found.

Before running the cycle, the probe is positioned at the start position, as shown in the onscreen graphic. The tool offset is active. The SELECT soft key is used to select the onscreen orientation that matches the orientation of the corner to be measured.

When the cycle runs, the probe first measures the Y-axis surface followed by the X-axis surface. The probe then returns to the start position. If an error occurs during the cycle, the probe returns to the start position.

Levelling a 4th axis (when fitted)



Note that this cycle is available only if a 4th axis is provided on the machine.

The level set up cycle is used to find the slope of a surface between two points (Z1 and Z2). The 4th axis can then be rotated to compensate for the surface error. The 4th axis must be positioned to the expected angular position of the feature; that is, the surface normal to the Z axis. The work offset register is adjusted by the error amount.

To make the new work offset active on most machines, it is normally necessary to re-state the work offset and move to the angular position after the cycle.

Tool setting

This section provides a brief description of how to select a tool setting cycle and then use each of the cycles. If you need more information about using a cycle, see “Using the software: an overview” on page 4.

Selecting a tool setting cycle



1. From the Renishaw splash screen (shown on page 4), press T-SET. The TOOL MEASURE screen is displayed.
2. From the TOOL MEASURE screen, press a soft key to select the cycle you require. The appropriate cycle screen is displayed.

Tool length setting



The manual and automatic length cycles are used to measure the effective cutting length of a rotating or non-rotating tool by taking a measurement on the tool setting stylus. The manual cycle can set the length of on-centre tools only; off-centre tools cannot be set. The auto cycle can set the length of on-centre and off-centre tools.

Manual cycle (for non-rotating tools only)

Before running the cycle, the spindle is jogged until the tool is directly over the probe's stylus and within 10.0 mm (0.4 in) of the surface. The cycle runs and the tool is moved to the stylus, where the measurement is made. At the end of the cycle, the tool returns to the Z clearance position above the stylus.

Auto cycle

When the cycle runs, the tool is selected from the tool changer and is moved to the stylus, where the measurement is made. At the end of the cycle, the tool returns to the Z home position.

Tool length and diameter setting



The manual and automatic length and diameter cycles are used to measure the effective cutting length of a rotating (or non-rotating) tool and the effective cutting radius of a rotating tool. The manual cycle can set the length and diameter of on-centre tools only; off-centre tools cannot be set. The auto cycle can set the length and diameter of on-centre and off-centre tools.

Manual cycle

The effective cutting length is determined from one measurement, taken on the top of the tool setting stylus. The effective cutting radius is determined from a measurement taken on each side of the tool setting stylus.

Before running the cycle, the spindle is jogged until the tool is directly over the probe's stylus and within 10.0 mm (0.4 in) of the surface. The cycle runs and the tool is moved to the stylus, where the measurements are made. At the end of the cycle, the tool returns to the Z clearance position above the stylus.

Auto cycle

The effective cutting length is determined from one measurement, taken on the top of the tool setting stylus. The effective cutting radius is determined from a measurement taken on each side of the tool setting stylus.

When the cycle runs, the tool is selected from the tool changer and is moved to the stylus, where the measurements are made. At the end of the cycle, the tool returns to the Z home position.

Sequential tool length setting



The sequential cycle is used to automatically set the effective cutting length of a range of tools in order. It can set the length of static on-centre tools only; off-centre tools cannot be set by this cycle.

Before running the cycle, the numbers of the first and last tools to be set are defined. Tool changing occurs automatically as the cycle runs.

Random tool length setting



This cycle sets the effective cutting lengths of up to 10 tools in the order in which the tools are entered in the list. This means that a random sequence of tools (e.g. T1, T8, T4 and T2) can be set if required.

The cycle can set the length of static on-centre tools only; off-centre tools cannot be set by this cycle. Tool changing occurs automatically as the cycle runs.

Broken tool detection (manual and automatic)



The manual and automatic check cycles are used to detect a broken or excessively worn tool. When a tool is found to be out of tolerance, the program raises a BROKEN TOOL alarm.

The auto cycle allows a tool change to take place as the cycle runs.

Changing a tool



This section provides a brief description of how to select and use the tool change cycle. If you need more information about using a cycle, see “Using the software: an overview” on page 4.

TIP: Before running a manual cycle, use this tool change cycle to select the tool into the spindle to provide an active tool /offset. This is particularly important when running a cycle after the machine has just been powered up and a tool has not yet been changed into the spindle.

1. From the Renishaw splash screen (shown on page 4), press T-CHNG. The TOOL CHANGE screen is displayed.

The tool change cycle can also be selected from the TOOL MEASURE screen.

2. Enter the number of the new tool.

Configuring the software (when access is permitted)

After default configuration values have been set for the first time as the software is commissioned, you are able to change them to suit the application provided you have been granted the necessary access status.

This section provides a brief description of how to select a software configuration screen and then use each of the configuration options. If you need more information about using a screen, see “Using the software: an overview” on page 4.

Selecting the software configuration screen



- ▶ From the Renishaw splash screen (shown on page 4), press the CONFIG soft key. The SOFTWARE CONFIGURATION screen is displayed.

Configuring controller settings



- ▶ To choose this option from the SOFTWARE CONFIGURATION screen, press the SYSTEM soft key.

Use this screen to configure parameters associated with the controller:

TOOL OFFSET TYPE: Select the correct tool offset to suit the machine. You can choose from offset types A, B or C.

Type A: One set of registers per tool offset number.

Type B: Two sets of registers per tool offset number. They are geometry and wear registers.

Type C: Four sets of registers per tool offset number. They are length geometry, length wear, radius geometry and radius wear.

Default: C

INCH OR METRIC UNITS Select the units to be used for measuring. This changes the units used for the setting data held in the P-code variables. All current setting data is automatically converted to the newly selected unit.

Default: Metric

Configuring inspection probe settings

- To choose this option from the SOFTWARE CONFIGURATION screen, press the PROBE soft key.



Use this screen to configure following parameters associated with an inspection probe.

IN POSITION ZONE	<p>This is the zone used for checking whether a probe open or probe fail condition exists.</p> <p>Default: 5 μm</p>
FAST FEEDRATE	<p>This is the fast feedrate that is used when an inspection probe approaches the surface of the feature.</p> <p>Default: 3000 mm/min (118 in/min).</p>
BACK OFF DISTANCE	<p>This is a multiplication factor and does not have any units associated with it. The default value usually ensures that a cycle will run first time without generating a PROBE OPEN alarm.</p> <p>After running a cycle, the back-off distance may be optimised by gradually reducing the value while checking that cycles continue to run without causing a PROBE OPEN alarm.</p> <p>Default: 0.25 mm (0.01 in)</p>
180 DEGREE ROTATION M-CODE	<p>This is the M-code number that activates the second spindle-axis orientation (180°) position.</p> <p>Your application might not use the M-code method for spindle positioning. In this case, treat the setting as a flag to select the type of calibration method used, as follows:</p> <p>0 This turns off the automatic centring feature during calibration.</p> <p>(Any value) This turns on the automatic centring feature during calibration. There is no need to clock a ring gauge.</p> <p>Note that this entry may have been disabled (i.e. the calibration method has been is fixed).</p> <p>Default: 0</p>
PROBE OFFSET NO.	<p>This defines the tool offset number that is assigned to the inspection probe. You should note that this entry fixes the tool offset register used for the probe whenever a workpiece set up cycle is run. If the tool pocket/offset is changed, the new offset register number must then be entered here.</p> <p>Default: 0</p>

VARIABLE BASE NO.	This is the P-code variable base number that is used by the workpiece cycles. Default: 21500
STORE RESULTS IN MACRO VARS	This allows the probing output results and, optionally, the calibration data to be output to NC macro variables as follows: NO Data is not sent to any of the NC macro variables. YES The probing output results are sent to NC macro variables #135 to #149. AND CALDATA The probing output results and the workpiece set up calibration data are sent to the NC macro variables (see also BASE NO. FOR CALIBR. DATA on the screen). Default: No
4TH AXIS NUMBER	The number of the 4th axis that is defined by the controller. Default: 0 (not used)
4TH AXIS UPDATE DIRECTION	Set to either CW or CCW to suit how the 4th axis has been configured. Default: CW DIR
BASE NO. FOR NC CALIBR. DATA	This is the P-code variable base number that is used by the calibration cycles. Default: 500

Configuring TS27R tool setting probe settings



- To choose this option from the SOFTWARE CONFIGURATION screen, press the T-SET soft key.

Use this screen to configure parameters associated with the TS27R tool setting probe.

Z APPROACH – LONG TOOL LENGTH	The first fast positioning move to the position where the tool offset is applied (the height above the stylus). This position is usually set to the longest tool length plus clearance. Default: 100 mm (3.94 in)
Z CLEARANCE POSITION	The position above the stylus for clear moves around the stylus (the height above the stylus). Default: 10 mm (0.394 in)

ROTATE TOOLS ABOVE DIA.	Tools above this diameter are rotated. Default: 10 mm (0.394 in)
MAXIMUM TOOL DIAMETER	The maximum diameter of a tool. Default: 80 mm (3.15 in)
STYLUS DIAMETER OR WIDTH	The nominal diameter of a round stylus or nominal width of a cube stylus. Default: 12.7 mm (0.5 in)
DEFAULT OVERTRAVEL	The distance past the expected surface the tool will travel as it searches for the surface of the probe. Default: 5 mm (0.197 in)
VARIABLE BASE NO.	This is the P-code variable base number that is used by the TS27R cycles. Default: 21600
BACK OFF DISTANCE	This is a multiplication factor and does not have any units associated with it. The default value usually ensures that a cycle will run first time without generation of a PROBE OPEN alarm. If a PROBE OPEN alarm is generated after the first measuring touch when running a cycle for the first time, increase the back-off distance. Run the cycle again. If a PROBE OPEN alarm is again generated, repeat this process until a PROBE OPEN alarm is no longer generated. To optimise the cycle times it may be possible to reduce the default value. Default: 0.5 mm (0.02 in)
SHORT TOOL LENGTH	This is the length of the shortest tool to be used. Default: 30 mm (1.18 in)
ON-CENTER APPROACH FEED	This is the fast feedrate that is used when an on-centre tool approaches the surface of the probe. Default: 2000 mm/min (78.7 in/min).
OFF-CENTER APPROACH FEED	This is the fast feedrate that is used when an off-centre tool approaches the surface of the probe. Default: 3000 mm/min (118 in/min).

Inputs

Inputs for workpiece set up cycles

BALL DIAMETER	The nominal diameter of the stylus ball.
CLEAR DIST	The clearance distance when moving around an external feature.
DEPTH	The depth to be measured. This is the incremental distance from the start point.
DIAMETER	The nominal diameter of the feature.
DISTANCE D1	The external corner find distance to the first measuring point along the X axis.
DISTANCE D2	The external corner find distance to the first measuring point along the Y axis.
DISTANCE I	The incremental distance to the second measuring point along the X axis.
DISTANCE J	The incremental distance to the second measuring point along the Y axis.
EXT G54.1 P	The extended work offset number (P1 to P48).
FEATURE DIAM	The nominal diameter of the feature.
RAD CLEARANCE	The clearance distance when moving around an external feature.
SEARCH DIST	The distance past the expected surface the probe will travel as it searches for a surface (the probe overtravel distance).
TOLERANCE	The tolerance value of a feature dimension to be set.
WIDTH	The nominal width of the feature.
WK. OFFS SHFT X	The work offset shift value in the X axis. This can be used to adjust the new work offset position by a set amount.
WK. OFFS SHFT Y	The work offset shift value in the Y axis. This can be used to adjust the new work offset position by a set amount.
WK. OFFS SHFT Z	The work offset shift value in the Z axis. This can be used to adjust the new work offset position by a set amount.
WORK OFFSET G	The standard work offset number (G54 to G59).

Levelling the 4th axis cycle inputs (when fitted)

ANGLE TOLERANCE	The tolerance value of the angle to be measured.
SPAN X	The distance between measurements along the X axis.
SPAN Y	The distance between measurements along the Y axis.
W. OFFS SHIFT	The work offset shift value in the 4th axis. This can be used to adjust the new work offset position by a set amount.
Z DISTANCE	The expected search distance to the surface.

Inputs for tool setting cycles

FIRST TOOL	The number of the first tool in the sequence to be set.
NEXT TOOL	The number of the next tool in the sequence to be set.
INC. Z DIST	The depth for measuring a diameter taken from the top of the stylus.
LAST TOOL	The number of the last tool in the sequence to be set.
RAD CLEARANCE	The clearance distance when moving around an external feature.
R EXPERIENCE	The experience value. Specify the number of a spare tool offset where an adjustment value to the measured size is stored.
SEARCH DISTANCE	The distance past the expected surface the tool will travel as it searches for the surface of the probe (the tool overtravel distance).
TOLERANCE	The tolerance value of the tool to be set.
TOOL DIAMETER	The nominal diameter of the cutting tool.
TOOL LENGTH	The nominal length of the cutting tool.
TOOL NUMBER	The number of the tool to be set or changed.
TOOL OFFSET D	The diameter offset number. This is the offset location in which the measured tool diameter is stored.
TOOL OFFSET H	The height offset number. This is the offset location in which the measured tool height is stored.

Inputs for inspection probe calibration cycles

BALL DIAMETER	The nominal diameter of the stylus ball.
DEPTH	The absolute Z-axis measuring position when calibrating on an external feature, e.g. a boss.
FEATURE DIAM	The diameter of the reference feature, e.g. a calibrated ring gauge.
RAD CLEARANCE	The clearance distance when moving around an external feature.
SEARCH DIST	The distance past the expected surface the probe will travel as it searches for a surface (the probe overtravel distance).
TOOL OFFSET H	The height offset number. This is the offset location in which the height of the active tool is stored.
Z REF POSITION	The position of the reference surface that is used for calibrating the probe's stylus, relative to the active work offset.

Inputs for TS27R tool setting probe calibration cycles

ARBOR LENGTH	The reference length of the calibration tool.
ARBOR DIAM	The reference diameter of the calibration tool.
INC. Z DIST	The depth for measuring a diameter taken from the top of the stylus.
RAD CLEARANCE	The clearance distance when moving around an external feature.
TOOL NUMBER	The number of the tool to be set.

Workpiece set up NC macro variables

Calibration cycle variables

The following calibration data is written across into NC macro variables. You should note that depending on how your system has been set up, these outputs might not be available.

NC variable used	Default	Description
# base number +0	#500	X+, X+ stylus ball radius (XRAD)
# base number +1	#501	Y+, Y+ stylus ball radius (XRAD)

NC variable used	Default	Description
# base number +2	#502	X-axis stylus offset
# base number +3	#503	Y-axis stylus offset

Cycle output variables

Variable outputs that are produced by some of the workpiece set up macros are listed here. You should note that depending on how your system has been set up, these outputs may not be available.

Variable No.	Z surface cycle	Bore/boss cycles	Web/ pocket cycles	Corner cycles	Edge cycle	4th axis cycle
# 135		X position	X position	X position	X position	
# 136		Y position	Y position	Y position	Y position	
# 137	Z position					
# 138	Size	Size	Size		Size	
# 139				X surface angle		4th angle
# 140		X error	X error	X error	X error	
# 141		Y error	Y error	Y error	Y error	
# 142	Z error			Y surface angle		
# 143	Size error	Size error	Size error	Y angle error	Size error	Height error
# 144				X angle error		Angle error
# 145	True position error	True position error	True position error	True position error	True position error	
# 146	Metal condition	Metal condition	Metal condition		Metal condition	
# 147	Direction indicator					
# 148	Out of tolerance flag (1 to 7)					
# 149	Probe error flag (0 to 2)					

Alarms and error messages

When an error occurs during use of the software, an alarm number and message is displayed on the screen of the controller. The meaning and likely cause of each alarm message and typical actions you should take to clear the fault are described here.

Some critical data entry fields are monitored. Entry of incorrect data immediately causes an alarm message to be displayed on the screen. To cancel the message, position the cursor at the input data causing the problem then enter correct data.

Secondary checking during run time of a cycle may cause an NC alarm message to be displayed. Further information on these messages is described in the relevant programming manual.

Message #3000 = 86 (PATH OBSTRUCTED)

Cause The probe was triggered during a positioning move.

Action Clear the obstruction and start again from a safe position.
Edit the program.
This is a reset condition.

Message #3000 = 88 (NO FEED RATE)

Cause This error occurs in the protected positioning cycle only.

Action Edit the program.
Insert the F___ code input and start again from a safe position.
This is a reset condition.

Message #3000 = 89 (NO TOOL LENGTH ACTIVE)

Cause G43 or G44 must be active before the cycle is called.

Action Edit the program and start again from a safe position.
This is a reset condition.

Message #3000 = 91 (T INPUT MISSING)
#3000 = 91 (X INPUT MISSING)
#3000 = 91 (Y INPUT MISSING)
#3000 = 91 (Z INPUT MISSING)
#3000 = 91 (XY INPUT MISSING)
#3000 = 91 (H INPUT NOT ALLOWED)
#3000 = 91 (SH INPUT MIXED)
#3000 = 91 (TM INPUT MIXED)
#3000 = 91 (XYZ INPUT MIXED)
#3000 = 91 (FORMAT ERROR)

Cause The input specified is either missing from the program or is incorrect.

Action Edit the program and start again from a safe start position.
This is a reset condition.

Message #3000 = 92 (PROBE OPEN)

Cause The probe is already triggered before a move.
The stylus may be in contact with a surface or the probe failed to reset.
This could be due to swarf trapped around the probe eyelid.

Action Clear the fault and start again from a safe start position.
This is a reset condition.

Message #3000 = 93 (PROBE FAIL)

Cause The probe did not trigger during the move.
The surface was not found or the probe failed.

Action Edit the program and start again from a safe start position.
This is a reset condition.

Message

		#148 flag	
#3006 =	1 (OUT OF TOLERANCE)	If the cycle start button is pressed to continue, the offset is updated.	1
	1 (OUT OF POSITION)		2
	1 (ANGLE OUT OF TOLERANCE)		4
	1 (DIA OFFSET TOO LARGE)		5
	1 (UPPER TOL EXCEEDED)	If the cycle start button is pressed to continue, no offset is updated.	3

Additional TS27R messages

Message #3000 = 99 (BROKEN TOOL)

Cause The tool is out of tolerance and the Mm input has not been used. A worn or broken tool is assumed.

Action Replace the defective tool and establish the correct tool offset value.

Renishaw plc
New Mills, Wotton-under-Edge,
Gloucestershire, GL12 8JR
United Kingdom

T +44 (0)1453 524524
F +44 (0)1453 524901
E uk@renishaw.com
www.renishaw.com

RENISHAW 
apply innovation™

**For worldwide contact details,
please visit our main web site at
www.renishaw.com/contact**



H - 2000 - 6430 - 00