



KEEN CHIEF

光鉛企業股份有限公司

MT SERIES OPERATIONAL MANUAL

SMALL design
HIGH SPEED performance
ACCURATE production



www.keenchief.com

MT SERIES



CHAPTER 1: SAFETY	3
1. SAFETY PRACTICES FOR SETUP	3
2. CHECKPOINTS FOR SAFE OPERATION	4
3. SAFETY PRACTICES FOR MACHINE OPERATION	5
4. SAFETY PRACTICES FOR MAINTENANCE AND INSPECTION	8
5. TO ENSURE HIGH ACCURACY	9
CHAPTER 2: INSTALLATION.....	10
1. PREPARATIONS	10
2. INSTALLATION.....	12
CHAPTER 3: INITIAL SETUP.....	17
1. CONTROL PANEL LAYOUT.....	17
2. POWER ON & SETUP.....	29
3. SETTING UP TOOLS	29
4. DRAWBAR ACTUATOR & COLLET NOSES	33
CHAPTER 4: BASIC PROGRAMMING	35
1. THE EDITOR	35
2. PROGRAM MANAGEMENT	39
3. SERIAL PORT (RS-232) COMMUNICATION	39
4. MEMORY CARD (PCMCIA) COMMUNICATION	39
5. PROGRAMMING BASICS	39
6. SIMULATE & GRAPHICS FUNCTIONS	39
CHAPTER 5: CANNED CYCLES.....	40
1. MOST COMMONLY USED CANNED CYCLES	40
2. G71 - ROUGHING IN THE Z AXIS	40
3. G74 - DRILLING CYCLE.....	42
4. G76 - THREADING CYCLE	44
CHAPTER 6: M&G CODE.....	46
1. COORDINATE TYPES (CARTESIAN)	46
2. ABSOLUTE POSITION	46
3. INCREMENTAL POSITION	47
4. RADIUS AND DIAMETER PROGRAMMING	47
5. MACHINE COORDINATE SYSTEM	48
6. M&G CODE LISTING	50
7. SPINDLE ORIENTATION	62

8. OTHER ADDRESSES.....	63
CHAPTER 7: TOOL NOSE COMPENSATION.....	66
1. G40, G41, G42 COMPENSATION CODES.....	66
2. G40 COMPENSATION CODE.....	66
3. G41 & G42 COMPENSATION CODES	66
4. R AND F COMPENSATION REGISTERS	66
5. COMPENSATION DIRECTIONS	68
6. ENTRANCE AND EXIT AMOUNTS FOR TNRC	68
CHAPTER 8: MAINTENANCE	69
1. DAILY CHECKUP AND MAINTENANCE	69
2. FORBIDDEN RULES (CAUTION).....	69
3. START-UP CHECK	70
4. CLOSING CHECK.....	71
5. LUBRICATION	71
6. PROXIMITY SWITCH CLEARANCE	77
7. HYDRAULIC SYSTEM	78
8. COOLANT EQUIPMENT	80
9. SPINDLE DRIVE BELTS	82
10. COOLING FAN FILTER	83
11. FRONT COVER WINDOW GLASS.....	83
APPENDIX A: SPECIFICATION.....	84
1. MACHINE SPECIFICATION	84
2. ELECTRICAL SPECIFICATION	85
APPENDIX B: SCHEMATICS	86
1. LUBRICATION SCHEMATIC	86
2. HYDRAULIC SCHEMATIC	87
3. COOLANT SCHEMATIC.....	88
APPENDIX C: PARTS LISTS	89

CHAPTER 1: SAFETY

Each Machine is shipped with a variety of built-in safety devices. However, careless handling of the machine may lead to serious injuries or death.

All operators must carefully read the manuals supplied by CNC unit manufacturer and KEEN CHIEF so that they understand the machine before trying to operate it.

Because there are so many “things which cannot be done” and “things which must not be done”, all prohibited information cannot be specified in the manual. Assume that something is impossible unless the manual specifically states that “it can be done”.

The following pages describe fundamental safety information.

All the items described must be carefully observed when operating the machine or performing maintenance work. Failure to observe fundamental safety information can lead to serious injury and machine damage. All operators must strictly follow the information.

1. *Safety practices for setup*



CAUTION

Never touch any switches or electronic parts with wet hands.

- The machine should be operated by one, well-trained person only. Injury is more likely to occur if more than one person operates the machine. The machine could be started by one operator while the other operator is changing the fixture or chuck jaws. If more than one operator is absolutely necessary, all involved operators must cooperate and be able to communicate continually.
- Cover your hair and do not wear loose clothing or jewelry to avoid becoming tangled or caught in the machine. Always wear proper shoes when operating the machine.

**CAUTION**

Never stand in front of the spindle. The workpiece, tools, or chuck jaws could fly out.

- When losing bolts on tool holders and cutting tools, be sure to loosen gradually. Do not tighten bolts excessively.
- Specify a spindle speed which is permissible for the chuck, cylinder, and fixture. If this condition is not satisfied, the workpiece may come off the spindle, injuring operators and damaging the machine.
- Select the proper chucking pressure for the desired type of machining.
- Clamp the workpiece and cutting tools securely. Depth of cut and cutting feed must be tested with conservative values first.
- Carefully check the workpiece chucked conditions.
- Make sure that tool holders, tools, soft jaws, and tailstock are all tightened securely. They should be mounted and well balanced so that they will not interfere with the workpiece or the machine.
- Be careful not to operate the wrong switch. Visually check the switches on the operation panel before operating them.
- Always select the most appropriate tool for the material and shape of the workpiece. Selecting a wrong tool will cause the workpiece to fly out of the chuck or will result in poor machining.
- The work lamp will be very hot after it has been lit for a long period. Be careful not to touch it.

2. Checkpoints for safe operation

**CAUTION**

When using your CNC lathe, always make sure the following conditions are met and operations followed. Failure to do this may cause serious bodily injury or death.

- When chucking a workpiece, check both the chucking method and the pressure while considering the rigidity of the workpiece so as not to cause

chucking distortion.

- Make sufficient chucking allowance so that the workpiece cannot detach from the chuck, either by cutting force or centrifugal force of the spindle.
- If the workpiece is of an eccentric or irregular shape, so that the centre of gravity of the workpiece is outside the centre of rotation, this eccentricity will cause the spindle vibrate during rotation, adversely affecting overall cutting accuracy. To counteract this, attach a balancer for the workpiece.
- If chip is suck to the workpiece or to the tool, the desired accuracy of cutting face may not be obtained. Select appropriate tools to prevent chips from sticking to the tools and workpiece.
- Before selecting tools, check the tools thoroughly so as to prevent any interference between the tools and workpiece, spindle nose, chuck, jaw, cover, etc.
- A wide variety of materials and shapes are used as workpiece to be cut. Always set the appropriate cutting conditions for each material and shape to obtain the desire accuracy of the particular product.
- At actuation of the machine and before cutting, warm up the spindle and slideways for 30 minutes to reach proper operating temperature. This is necessary to reduce the influence ton the workpiece by thermal expansion.
- When cutting the bar material using a bar feeder and a hole through spindle, use only absolutely straight bar material. The accuracy of the workpiece maybe adversely affected by any curvature in the bar material.
- With forged or cast material, the cutting allowance is often uneven. Create the program in anticipation of this uneven allowance or set a constant allowance by preparatory cutting.
- The machine can not be sued in a potential explosive environment. It can not be used to cut flammable or explosive material (such as magnesium alloy).

3. Safety Practices for Machine Operation



CAUTION

Never touch the moving units of the machine while the machine is operating. Serious injury can occur by being caught in the rotating parts.

**CAUTION**

Never touch any switches or electronic parts with wet hands.

- Do not insert bar stock into the spindle while the spindle is rotating.
- Keep the front door closed while the machine is operating. The area inside the front door contains many sources of potential danger the spindle rotating at a high speed with a workpiece clamped in it, splashing coolant and flying chips.

**CAUTION**

Never try to open the front door while the spindle is rotating to remove chips or try to touch the workpiece or cutting tools.

**CAUTION**

Never stand in front of the spindle. The workpiece, tools, or chuck jaws could fly out.

**CAUTION**

Never remove the machine covers unless absolutely necessary.

**CAUTION**

Never start the machine without the safety devices in place.

- Cover your hair and do not wear loose clothing or jewelry to avoid becoming tangled or caught in the machine. Always wear proper shoes when operating the machine.

**CAUTION**

Do NOT lean on the machine while it is operating.

- Specify a spindle speed which is permissible for the chuck, cylinder, and fixture. If this condition is not satisfied, the workpiece may come off the spindle, injuring operators and damaging the machine.
- After the completion of a cycle, before removing a machined workpiece and setting a new workpiece, always check that machine has topped moving (i.e., not in EXECUTION mode)
- Clamp the workpiece and cutting tools securely. Depth of cut and cutting feed must be tested with conservative values first.
- Always use straight bar stocks. When bar stock is machined using a bar feeder and through-spindle hole, the bend of the bar stock will cause vibration which, in turn, will deteriorate the accuracy of the finished workpiece.
- When machining bar stock longer than the spindle length, you should always use a bar feeder.
- When running a new program for the first time, check the program number. Never attempt to start a new program in the automatic mode, carefully run the program one block at a time using the single block function.
- During automatic mode operation, be careful not to touch any switches inadvertently.
- If a hard over travel alarm occurs, the axis interlock must be released to move the axis. In this case, never move the axis in opposite direction.

**CAUTION**

Never put any tools or instruments on the machine operation panel or on any machine part.

- Be careful not to operate the wrong switch. Visually check the switches on the operation panel before operating them.

4. Safety Practices for Maintenance and Inspection



CAUTION

Never put any tools or instruments on the machine operation panel or on any machine part.

- Wiring work for 200 VAC or higher voltage circuits must be performed only by an authorized electrical technician.



CAUTION

Never change parameter settings without consulting your representative. If changed inadvertently, some parameters may cancel interlock safeties.



CAUTION

Do NOT touch the conveyor when chips are being discharged.

- Disassemble and clean the chuck periodically. Apply grease to the chuck as necessary.
- The solenoid valves become very hot when the machine is operating. Be very careful not to touch them even after the power is turned OFF. Wait for them to cool down.
- Supply or change hydraulic oil or lubricating oil as specified in the manual.
- The fan the filters in the electrical cabinet must be kept clean.



CAUTION

Do NOT open the door of the electrical cabinet, the NC unit, or the operation panel unless absolutely necessary. Opening the door allows dust, foreign matter and moisture to enter the enclosure and can cause machine malfunctions.

**CAUTION**

Do NOT climb on the machine unless absolutely necessary.

- The work lamp will very hot after it has been lit for a long period. Be careful not to touch it.

5. *To Ensure High Accuracy*

When operating a CNC lathe, the accuracy of finished product cannot be maintained unless the following check points are observed. Failure to observe these points can also cause to accidents.

- Allow a sufficient chucking amount so that the workpiece will not detach from the chuck due to the cutting force or centrifugal force generated by spindle rotation.
- When chucking a workpiece, determine the chucking method and chuck pressure considering the rigidity of the workpiece so that the workpiece will not be distorted by the chuck.
- Machine vibration will result when a workpiece's center of gravity is not aligned with spindle center. This in turn, will deteriorate the accuracy of the machined workpiece. It is necessary to balance the workpiece with a balancing weight.
- Careless tooling will cause crashes between the tools and the workpiece or other parts of the machine. Check the tooling carefully to avoid interference.
- Before starting a production run, warm-up the machine properly. This will minimize the influence of thermal distortion on workpiece accuracy.
- When bar stock is used, its bend has critical influence on the accuracy of the machined workpiece. Use straight workpiece only.
- If any chips are entangle on the workpiece or cutting tool, surface roughness will be deteriorated. Select a cutting tool with good chip breaking capability.
- Workpiece materials and shapes will vary widely. It is necessary to select the cutting conditions best suited to each workpiece in order to obtain the required accuracy.

CHAPTER 2: INSTALLATION

1. Preparations

(1) Power Supply Facilities

Three-phase AC power line of 220 (+10%, -15%) volts, 50/60 Hz is used. If the power supply voltage in the user's facility differs from 220V AC, a power transformer (available as option) will be needed. The required power size varies depending on the integrated devices and equipment.



CAUTION

- Only an authorized electrical technician should perform work with the power cable connections.
- No electrical noise generating sources, such as electric welder or electric discharge machines can be near the machine. Take care to isolate the machine from any adverse effects that might be caused by nearby equipment.
- An excessive voltage drop due to an insufficient power capacity will cause a malfunction of the NC unit. The power cable must be connected directly and independently to the plant power distribution panel.

(2) Lubricants and Hydraulic Oil

Prepare all lubricants and hydraulic oil whose brands and quantities are specified on Lubrication section under the Maintenance chapter.

(3) Coolant

For the standard coolant equipment, prepare 15 gallon of coolant. It is recommended to use water-soluble coolant of the emulsion type or water insoluble coolant that does not corrode metals, rubber and plastics nor dissolve paint.

(4) Floor Space

MT requires 2090mm width x 1200mm deep of floor space for installation. The height is 1900mm.

Note: The above floor space requirement is a net space occupied by the

machine and its chip conveyor, and does not include any space for maintenance work. The above space enlarges as optional devices and equipment such as bar feeder; robot and work stocker; etc. are installed.

(5) Environment for Installation

The machine and equipment should be installed at a place where the following environmental conditions are satisfied:

Ambient Temperature	5 to 40°C
Relative Humidity	10 TO 90 % (non-condensing)



WARNING

- Avoid a place where the ambient temperature changes suddenly or frequently to prevent thermal deformation.
- Avoid a place where vibrations or impacts arise. They maybe cause not only machining quality failures but also loosening of tightened parts, malfunction of the relays and in extreme cases, damage to the machine and equipment.
- Avoid dusty and dirty areas.
- Select a place where the machine and equipment are not exposed to direct sunlight.

(6) Ground-fault Interrupter



WARNING

- When using a ground-fault interrupter for the user’s power distribution board, select an inverter-response type interrupter having the minimum leak current sensitivity of 100mA or less.

(7) Grounding of Machine

Grounding of the machine should be performed using a ground cable of not less than 5.5mm²(0.009 sq. in.). Generally, it is required to carry out grounding work (by driving a grounding rod or the like in the ground) independently to provide a single grounding element for each machine.

When conducting the grounding work, observe the following minimum precautions.



WARNING

- The grounding should be performed in conformity with the local electrical standards.
- Connect only one grounding cable to an earth terminal. Never share one earth terminal with other equipment. Otherwise, if poor contact arises on the earth terminal, the earth current resulting from other equipment failure may flow through the machine to cause a serious accident.
- Ground the machine to a single grounding electrode. Never share the earth terminal with other noise-producing machines. Otherwise, the CNC unit of the machine may malfunction due to electrical noise.
- The machine must be grounded independently of other machines.
- If electric welder or electric discharge machine are grounded to the frame of plant building, do not connect the ground wire of the machine to the plant frame.
- The ground wire must be as short as possible and have the same diameter as the input cable.

2. Installation

(1) Site Selection

- Allow space for maintenance. Install the machine so that the doors of the machine and the NC unit can be opened without interference.
- Do not put things on the floor around the machine. Keep the floor dry. If coolant or lubricating oil is spilled, wipe it up immediately.
- The machine and the NC unit must not be subject to direct sunlight, Chips, coolant, and oil must not be splashed on the machine or the NC unit. The machine and the NC unit must not be subject to any excessive vibrations.
- Make sure that the floor is strong enough to support the machine. The floor must not be sloped or irregular in any way.
- A number of cooling fans are used inside the machine. Therefore, dust and mist must be kept to a minimum.

NOTE: Allow space for easy removal of the chip conveyor and the coolant tank.

(2) Lifting

To lift or hoist the machine, be sure to follow the precautions below:

- Only an authorized technician should perform work with the machine hoisting or lifting.
- Before hoisting the machine, make sure that unit is fixed securely.
- Before hoisting the machine, make sure that nothing unnecessary is left on the machine.
- Be sure that the machine is well balanced both lengthwise and crosswise while hoisting the machine slightly above the floor.
- When more than one worker are moving or rigging the machine, be sure to call attention each other as necessary.

(3) Unpacking

- The tool slide is supported in place with wooden block when the machine is shipped. This wooden block must be removed before turning ON the power.



- All desiccants must be removed completely.



(4) Floor Foundations

For best operating results, the machine should be installed on a rigid foundation made of reinforced concrete above a layer of crushed stones in order to prevent vibration and maintain proper levelness. When the ground fails to provide sufficient withstanding level, the layers of concrete and stone should be increased as necessary. When installing the machine without using anchor bolts, place a piece of the supplied floor leveling pads under each jack bolt of the machine. When fixing the machine with anchor bolts, provide hollows of 200mm (8") square and 450mm (18") deep to embed the anchor bolts. At the same time, provide a bevel for each hollow to facilitate pouring of mortar.

(5) Leveling

The reading accuracy (graduation) of a precision level used for leveling should be 0.001mm per meter or better.

Put precision levels on an OD tool block attached to the tool plate and adjust the 4 jack bolts of the machine to obtain the machine levelness accurate to 0.02mm per meter (0.0007" per yard) or better in both front-to-back (X-axis) and left-to-right (Z axis) directions.



NOTE: Install the machine carefully and accurately to bring out the best performance and maintain it for a long time. After completing the machine installation, check the levelness once again before starting

operation. Recheck the level one month after putting the machine into production run, and thereafter every half year or thereabouts.

(6) Connection of Power Cable

When connecting a power cable to the machine, an attached cable clamber or compatible clamber should be installed in order to prevent coolant mist from entering into the electrical cabinet through the opening for the power cable. The cable size should be larger than 22mm^2 (0.034 sq. in) (OD 22 to 32mm) for the attached clamber.

(7) Before Turning ON the Power after Installation

After completing machine installation, check the following points before turning ON the power:

- Make sure that bolts are tightened securely.
- Make sure that all hydraulic hoses are connected securely.
- If the machine is equipped with any optional external equipment (bar feeder), make sure that each electrical cable and hydraulic pipes are connected correctly.
- Check for an input voltage of 220 V using a multimeter and check all the L1/L2/L3 phases of input power.

(8) After Turning ON the Power after Installation

- Check for oil leaks. Make sure that all of the gages indicate the correct values.
- Repeatedly open and close the chuck to break-in the chuck operating cylinder. Then, break-in the spindle.
- To break in the spindle, run the spindle for 20 minutes at a low speed. Increase spindle speed in five steps to the maximum speed, running the spindle for 20 minutes at each break-in spindle speed.

(9) Setup Before Automatic Operation

**WARNING**

- Before starting the automatic operation of the machine always check the following points and perform the war-up procedure.
- If any anomaly is found in the machine, immediately stop the machine and completely remove the cause of the problem before restarting automatic operation.
- Move each CNC axis by manual operation and check for unusual sound and vibration.
- Perform the warm-up operation.
- If the warm-up operation is not performed, the machine will fail to deliver its best performance.
- Check the machining program, tooling on the table, tool offset data, workpiece type another setup elements to be sure they correspond.
- Check to be sure that the reference point return operation has been completed for all CNC axes. If it has not been done, perform it.

(10) Warm-up Operation

For best performance, the machine needs the temperature of its mechanical units to come up to a constant degree. Therefore, it is necessary to warm up the machine for about 30 minutes before starting automatic operation (production run) to thermally stabilize the CNC feed axes (X and Z), main spindle and the other moving parts, and to check the functioning of the machine.

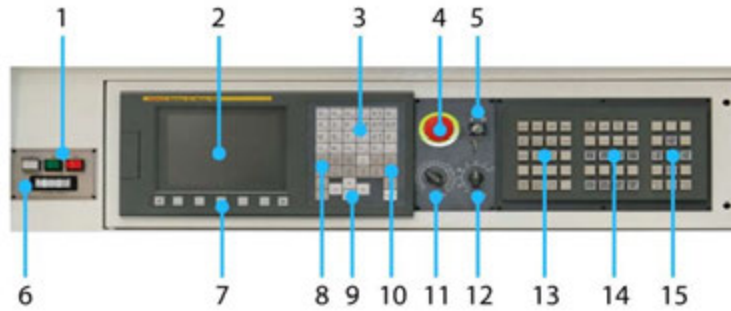
- Usually the warm-up operation is carried out in the automatic operation mode. So, it is recommended to make a special program for warm-up operation and store it in the memory of control.
- This warm-up operation should include the main spindle rotation at 500 rpm or so, the table feeds in the X-axis and Z-axis directions (first at slow federates, then at gradually increased speeds, and finally at the rapid traverse rates).

NOTE: After the machine has been left unused for more than a few days, double the warm-up operation time (i.e., 1 hour).

CHAPTER 3: INITIAL SETUP

1. Control Panel Layout

The MT Utilizes the Fanuc Oi Mate-TC control Panel



- | | |
|---------------------------|----------------------------|
| 1. On & Off Buttons | 9. Cursor Keys |
| 2. LCD Screen | 10. Editing Keys |
| 3. Address-numeric Keypad | 11. Feedrate Override |
| 4. Emergency Stop Button | 12. Spindle Speed Override |
| 5. Door Interlock Switch | 13. Button Panel 1 |
| 6. Overtravel Button | 14. Button Panel 2 |
| 7. Menu Softkeys | 15. Button Panel 3 |
| 8. Function Keys | |

(1) On & Off Buttons

The On button is used to power up the machine and of course the Off button is used to power down the machine. The spindle load meter shows the percent of allowable electrical load currently on the spindle.

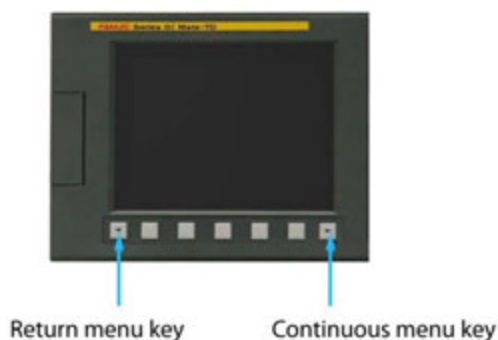


NOTE: After pressing the On button, there is a delay of a few seconds before anything happens on the LCD screen.

(2) Overtravel Button

If the slides are run off their maximum axis positions, the machine will disable the slides to prevent damage. To enable the slides again, Overtravel Button (labeled O.T. Release) must be pressed while using axis jog keys to move the slides back into their normal positions.

(3) LCD Screen

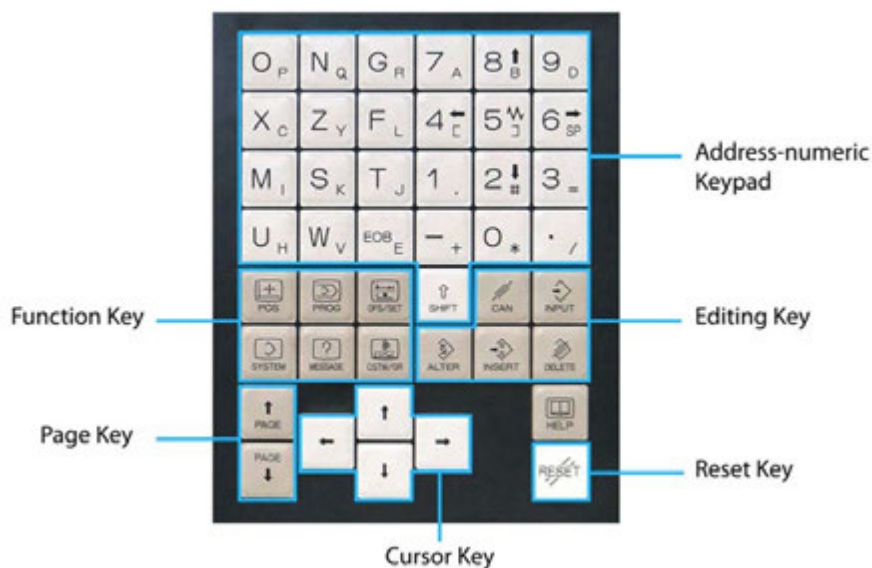


The LCD screen is the main interface for the MT-SERIES. This screen shows user input and displays the status of the machine. The Continuous Menu Key is used to reveal additional menu items than can be shown on one screen. The Return Menu Key move the menu up one level.

(4) Menu Softkeys

The menu softkeys are used to select menu items that show up on the bottom of the LCD screen. If there are more menu items than will fit on the screen, press the Continuous Menu Key to see the additional items. Use the Return Menu Key if you want to go back to the previous menu selection.

(5) Keypad









(6) Address-Numeric Keypad

These keys are used to enter information into the MT. The lower characters

on the keys can be entered by using the Shift Key.

(7) Function Keys

The function keys are used to access different screens on the Fanuc MT.

	Displays the position screen. Can toggle through absolute and relative position of the slides.
	Displays the program screen. Used to create new programs and edit existing ones.
	Displays the offset/set screen. Used to set tools and provide tool offsets.
	Displays the system screen. Used to set CNC parameters and to configure the machine.
	Displays the message screen. Shows all alarms and status messages from the MT-SERIES.
	Displays a custom/graphics screen. Used for setting graphics parameters.

(8) Cursor Keys

The cursor keys are used to scroll through screens and parts of a program.

(9) Page Keys

These keys are used to scroll up and down through menu screens one page at a time.

(10) Editing Keys

These keys are used when writing or editing a program. The Cancel Key (labeled “Can”), is used to delete the last character or symbol in the input buffer.



The Input Key is an exception among the editing keys as it isn’t used for writing and editing programs. The Input Key is only used when changing values outside of programming environment such as changing parameter or offset values.

The Alter Key (alteration) replaces characters or symbols under the cursor with characters from the input buffer.

The Insert Key inserts characters or symbols from the input buffer to after the current cursor location.

The Delete Key deletes characters and symbols that are under the cursor.

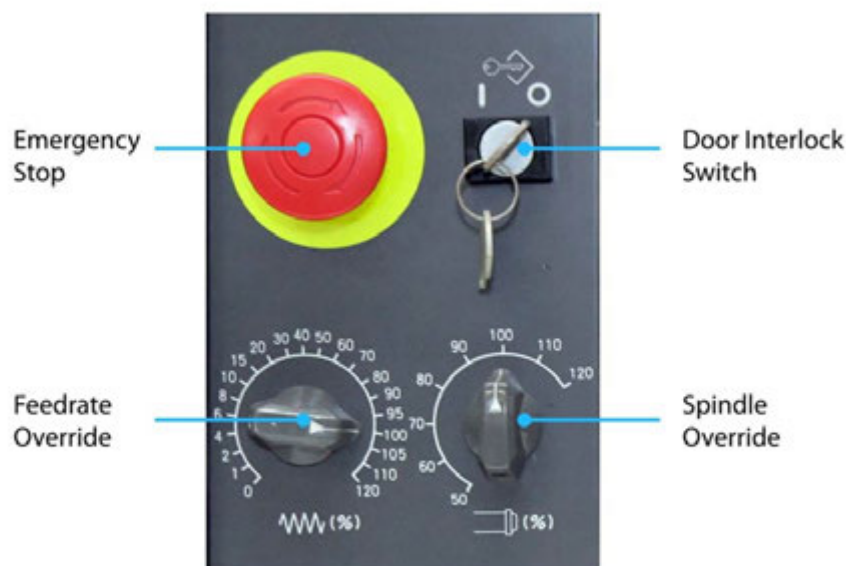
(11) Reset Key

The Reset Key is used to set any given mode to its original settings. In editing mode, pressing the Reset Key will rewind the program pointer to the beginning.

The Reset Key also stops a running program when in automatic mode.

(12) Emergency Stop

The Emergency Stop button immediately stops the machine from operating in the case of an emergency. The slides and spindles are stopped in place and the machine will not function until the Emergency Stop button is twisted to the right.



(13) Door Interlock Switch

This is a keyed switch that prevents the machine from running when the door is open. The switch toggles the door interlock on and off. When the door interlock switch is turned to the left, the machine will freeze the slides when the door is open. Turning the key to the right disables door interlock allowing the slides to move while the door is open.

(14) Feedrate Override

The feedrate override knob is only enabled when the machine is in execution and jog mode. This knob allows the operator to override the programmed slide speed. The knob has a range from 0% to 120%. The 0% position effectively freezes the slides and the 120% position moves the slides 20% faster than the programmed speed. The feedrate override knob is useful in testing out new programs at slow speed.

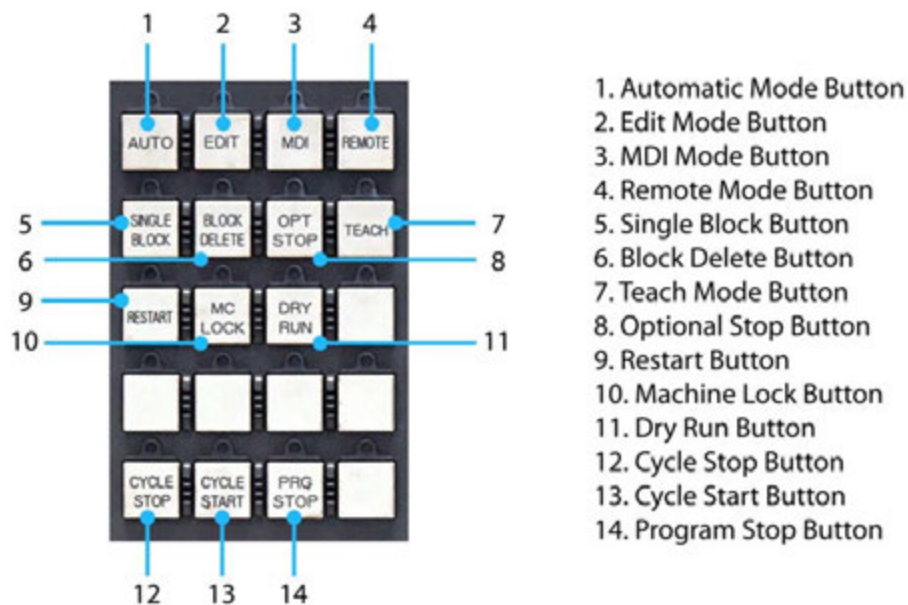
Once the program has been verified, the operator can use the feedrate knob to speed up program execution.

NOTE: The feedrate override knob also affects rapid traverse speeds when running a program.

(15) Spindle Override

The spindle override knob allows the operator to override the programmed spindle speed. The knob has a range from 50% to 120%. The 50% setting slows the spindle speed by half and the 120% setting runs the spindle 20% faster than programmed.

(16) Button Panel 1



<1> Automatic Mode Button

This mode sets the machine for automatic execution of a program. To start a program, the automatic mode button must be enabled.

<2> Edit Mode Button

The edit mode brings up a screen for creating new program and for altering existing ones.

<3> MDI Mode Button

The MDI button enables MDI mode which is used for simple testing operations. In MDI mode, you can create programs up to 10 lines long and execute them.

<4> Remote Mode Button

The remote mode button has no relevant function on the MT-SERIES and can be ignored.

<5> Single Block Button

This button enables single block execution where the machine executes one line of code at a time and then stops. Pressing the cycle start key will execute the next line of code and the stop again.

Single block mode is great for testing programs by stepping through each line of code and checking the results before moving on.

<6> Block Delete Button

When the block delete button is enabled, the machine will skip execution of lines that begin with a forward slash (/).

For example, given the following code:

```
X-1  
/ Z-2  
X+1
```

When the block delete button is turned on, the machine will only execute the X-1 and X+1 lines of code.

<7> Teach Mode Button

This button enables “Teach In” mode. The “Teach In” mode allows you to write a program by manually specifying coordinates. That is, you can write a program using “Teach In” mode by jogging the machines slides to their desired positions in sequence and store the coordinate values in the program. (BUTTON DISABLED)

<8> Optional Stop Button

When this button is pressed, it will cause program execution to stop on lines where there is an M01 code. The M01 code must be on its own line. The spindle and coolant will continue to run, but the slides will be stopped.

Pressing Cycle Start Button will restart program execution.

<9> Restart Button

This button enables the program restart mode and allows you to restart a program from a specified block number. This mode is useful when a tool breaks and you’d like to restart from the last position before the tool broke. This button has no relevant function on the MT-SERIES and can be ignored. (BUTTON DISABLED)

<10> Machine Lock Button

When this button is turned on, the screen will display changes in position, but the machine slides will not move. In other words, the machine lock mode will simulate running the program without actually moving the tools.

<11> Dry Run Button

The dry run button moves the machine slides at a fixed federate overriding the feedrates specified in the program. The dry run mode is used without a workpiece in place to test the tool path.

The dry run mode can be combined with the machine lock mode to run through the program at a fixed speed, but without moving the slides.

<12> Cycle Stop Button

The cycle stop button will halt an executing program.

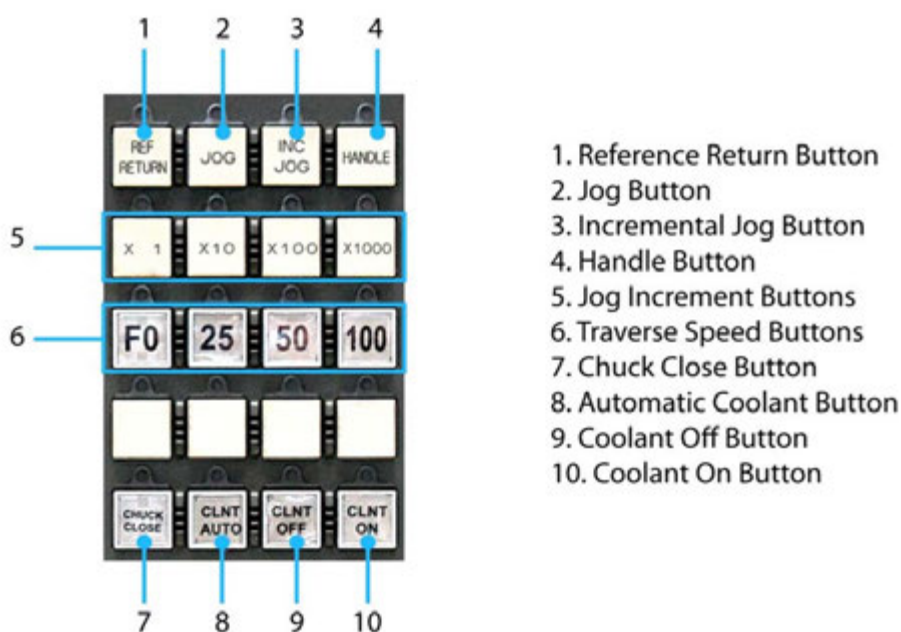
<13> Cycle Start Button

The cycle start button will start the execution of a program that is currently selected.

<14> Program Stop Button

This button will light up if during automatic execution of a program there is an M00 program stop code.

(17) Button Panel 2



<1> Reference Return Button

The reference return button is used to home the X and Z axes. When the MT-SERIES is turned on for the first time, it will need to be homed. This is done by pressing the Reference Return Button and then using the Jog Axis Buttons “X+” and “Z+” buttons. Detailed instructions on homing the axis is given later in the section.

<2> Jog Button

The job button enables continuous jog mode where pressing a jog axis button will continuously move the slides until the key is released. The X+ and X- buttons on Button Panel 3 move the slide up and down

towards the spindle respectively (along the X+ and X- direction). The Z- and Z+ arrow keys move the slide into and away from the spindle (along the Z- and Z+ direction).

In continuous jog mode, the slide's speed is controlled by the federate override knob. The higher the feedrate percentage, the faster the slides will move. The slides can be moved at their maximum speed by pressing the rapid traverse button (the button between the cursor keys) and cursor keys at the same time.

<3> Incremental Jog Button

The incremental jog button is tied to the handle mode. Pressing the button will light up the Handle Button and enable the manual pulse generator (MPG).


<4> Handle Button

The Handle Button enables the use of the manual pulse generator (MPG) to control the slides. Incremental jog is done using the MPG. Details about the MPG are noted later in this section.

<5> Jog Increment Buttons

These buttons are status indicators for the multiplier that is currently set on the MPG. As the multiplier is changed on the MPG, the lights will change to reflect the new multiplier. These buttons indicate the amount the slides will move. Each jog increment button specifies a multiplier on 0.0001" to give the amount the slides will move. The following table lists the amount the axes are moved for each of the jog increment buttons. The "x1000" button is never active as the MPG only supports a multiplier up to "x100".

Button Name	X-Axis	Z-Axis
X 1	0.0001"	0.0001"
X 10	0.0010"	0.0010"
X 100	0.0100"	0.0100"
X 1000	0.1000"	0.1000"

	WARNING
	<p>Exercise caution when the tool is close to the collet or chuck. If a high jog override multiplier is selected (e.g. 10,000) and the tool is close to the workpiece, a collision could occur when jogging the tool towards the spindle.</p>

<6> Traverse Speed Buttons

The Traverse Speed Buttons affect the speed of the slides when doing rapid traverse. Each of the traverse speed buttons specify a percentage of maximum speed at which the slides will move.

Key	Percentage Override
F0	0% of max speed (moves very slowly)
25	25% of max speed
50	50% of max speed
100	100% of max speed (normal rapid speed)

<7> Chuck Close Button

This button is really a status indicator of the collet. When the collet or chuck is closed, this button is lit.

<8> Automatic Coolant Button

The automatic coolant button let the program control coolant operation.

<9> Coolant On Button

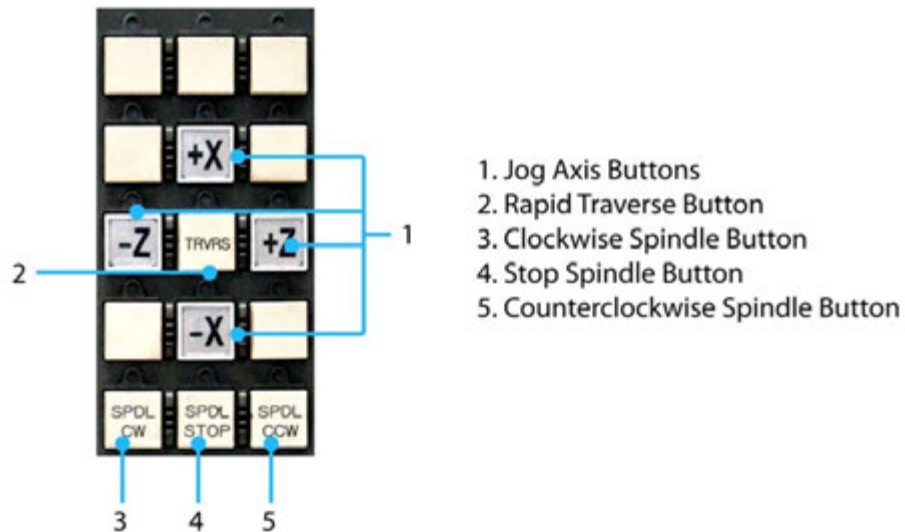
This button manually turns on the coolant overriding any coolant settings in the executing program.

Also during program execution, this button is a status indicator lighting up when the coolant is running.

<10> Coolant Off Button

This button turns off the coolant overriding any settings in the executing program. During execution mode, this button is a status indicator lighting up when the coolant is off.

(18) Button Panel 3



1. Jog Axis Buttons
2. Rapid Traverse Button
3. Clockwise Spindle Button
4. Stop Spindle Button
5. Counterclockwise Spindle Button

<1> Jog Axis Buttons

The jog axis buttons are used in jog mode to move the slides. The X+ and X- buttons on Button Panel 3 move the slide up and down towards the spindle respectively (along the X+ and X- direction). The Z- and Z+ arrow keys move the slide into and away from the spindle (along the Z- and Z+ direction).

These buttons will blink when the slides are in their home position.

<2> Rapid Traverse Button

This button is used in jog mode to move the slides as fast as possible. The rapid traverse button is used in conjunction with jog axis buttons.

Holding down the rapid traverse button along with a jog axis button will move the slides at the machine's maximum speed towards the given direction. The maximum speed during rapid traverse can be controlled by the Traverse Speed Buttons located on Button Panel 2.

<3> Clockwise Spindle Button

The Clockwise Spindle Button is used in MDI mode to begin spinning the spindle in a clockwise direction. In automatic mode, if a program is spinning the spindle clockwise, this button will be lit.

<4> Stop Spindle Button

This button will stop the spindle in MDI mode.

In automatic mode, if the spindle is stopped this button will be lit.

<5> Counterclockwise Spindle Button

The Counterclockwise Spindle Button is used in MDI mode to begin spinning the spindle in a counterclockwise direction. In automatic mode, if a program is spinning the spindle counterclockwise, this button will be lit.

(19) Manual Pulse Generator (MPG)

The handle mode allows the slides to be moved by using the manual pulse generator (MPG), also known as the handwheel. The handwheel is enabled by the Handle Button on Button Panel 2.

To turn on the handwheel, switch the axis switch from OFF to the desired axis. The increment with which to move the slides is set on the handwheel by setting the increment switch to x1, x10, or x100. The following table indicates how much each position on the handwheel increment switch will move the slides.



x1	= 0.0001"
x10	= 0.0010"
x100	= 0.0100"

The Spindle Index key is used to orient the spindle to its home position. This sort of indexing is also known as the C-axis and allows the operator to position the spindle in discrete stops across the all 360 degrees. Spindle orientation can also be performed using the M19 code and more is explained in the M Codes section.

NOTE: If the spindle will not respond to the spindle direction, start and stop keys, make sure you're in Jog mode and that a spindle speed has been programmed.

(20) Optional Foot Switch

The foot switch can be used to open and close the collet.

2. Power On & Setup

Once the MT-SERIES has been plugged into power and the circuit breaker switched on from the back, the machine must be powered up and homed. Follow these steps to do this:

1. Press the Power On Button and wait a minute for the machine to start up.
2. Check to make sure that the handwheel is turned off and that the Overtravel Button is not lit.
3. Verify that the emergency stop button is off (out). Twist the button to the right if it is pushed in.
4. Press the Position Function Key. The screen will show the current X and Z position of the slides.
5. Press and hold down the Z+ Jog Axis Key until the position screen reads 0 for the Z axis.
6. Press and hold down the X+ Jog Axis Key until the position screen reads 0 for the X axis.

The slides are now homed.

Once the machine is powered on and in its default home position, the tools must be setup.

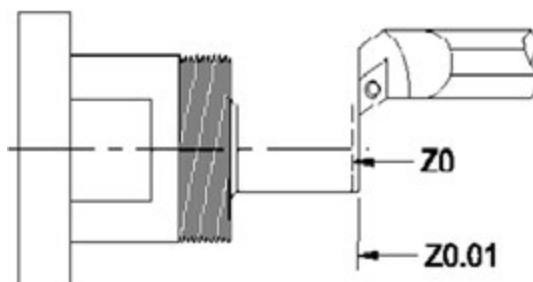
3. Setting Up Tools

Always home the X & Z axes before setting up the tools.

It is always recommended to reset any G54 thru G59 offsets to zero by using the G53 offset.

(1) Setting a Tool

All tools should be set on the tool plate prior to this step. Also, decide a number that will be assigned to each tool.



(2) Setting the Z Axis Offset

Jog the tool you want to offset in the Z direction until you touch the end of the stock.

Then without moving Z, jog off in the X+ direction until the tool is clear. In this example we are allowing 0.010" facing stock on the end of the material. Assuming you're still in the 'Tool Offset' menu, enter this information with the following key sequence:

1. OFFSET/SET Function Key
2. OFFSET Softkey
3. GEOM Softkey
4. (OPRT) Softkey
5. Cursor to the Z column
6. Z0.01
7. MEASUR Softkey

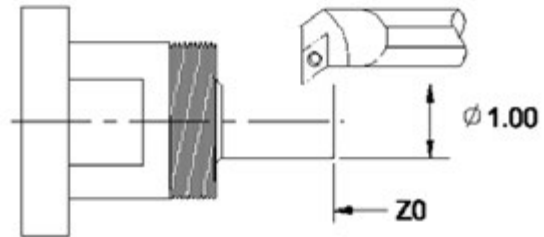
The Z value is now recorded and displayed on screen.

Notes: Always remember that X or Z can be positive or negative, enter the correct value including the negative sign (-) if the value is negative.

(3) Setting the X Axis Offset

Jog the tool you want to offset using the handwheel in the X direction to touch the outside diameter (OD) of the stock.

Then jog off the stock in the Z+ direction without moving X. In this example the material has a diameter of 1.00 and the tool is in line with that OD.



Enter this information with the following key sequence:

1. OFFSET/SET Function Key
2. OFFSET Softkey
3. GEOM Softkey
4. (OPRT) Softkey
5. Cursor to the X column of the current tool number

- 6. X1.0
- 7. MEASUR Softkey

The X value is now recorded and displayed on screen.

Repeat these steps for each tool on the plate setting the X and Z offsets for each tool.

(4) Tool Wear Offsets

You can specify a tool wear offset that is added to the geometry offsets for each tool. You can access the tool wear offset screen by using the following key sequence:

- 1. OFFSET/SET Function Key
- 2. OFFSET Softkey
- 3. WEAR Softkey

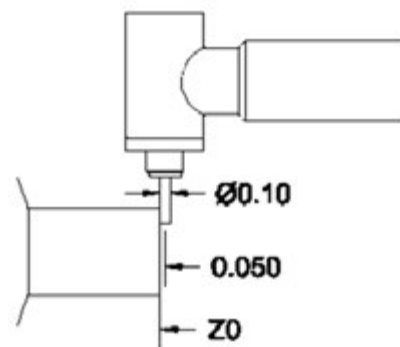
You can then either directly input or measure a wear offset for each tool just as you would tool geometry offsets.

(5) Setting Live Tools

Always set the tool centerline for end mills in an axial orientation. The C-axis won't give correct results if this setting is incorrect.

To set the drill center line in radial applications, touch off the side off the side of the tool with Z0 and specify the radius of the drill or end mill. Offsetting live tools is the same as offsetting tools on the plate. The following sequence of buttons will set a live tool with an offset of 0.05 from the Z axis:

- 1. OFFSET/SET Function Key
- 2. OFFSET Softkey
- 3. GEOM Softkey
- 4. (OPRT) Softkey
- 5. Cursor to the Z column
- 6. Z0.05
- 7. MEASUR Softkey



Once all the tools have been setup and calibrated, the MT-SERIES is ready for machining. If you already have a sample program in the machine, the

follow steps will guide you on how to begin the machining sequence.

If you don't have a sample program, check the programming section of this manual to create one.

(6) Running the Spindle in MDI Mode

To test the spindle manually, do the following key sequences:

1. **[MDI Mode]** Button
2. **[PROGRAM]** Function Key
3. **S<spindle_rpm_speed> [EOB]** (end of block)
4. **[CYCLE START]** Key

Now that the spindle speed has been programmed, the final step to turning on the spindle is setting a spindle direction (either **[clockwise]** or **[counterclockwise]**).

To set the set the spindle direction and assuming that you're still in MDI mode:

Press **[either the Clockwise]** or **[Counterclockwise]** Spindle Button.

This will start spinning the spindle at the programmed speed and direction. Alternatively, the spindle direction could also be programmed directly with the spindle speed using the M03 (spindle forward) and M04 (spindle reverse) codes.

For example: **S<spindle_rpm_speed> M03 [EOB]**

(7) Executing the Program

Follow these instructions to execute your sample program:

1. Press the **[AUTOMATIC]** Mode Button
2. Press the **[PROGRAM]** Function Key
3. Press the **[DIR]** Softkey
4. Enter **O<program_number>** (be sure to enter leading 0s)
5. Press the **[O SRH]** Softkey
6. Press the **[CYCLE START]** Key

Monitor the program closely to make sure the settings are correct. Use the feedrate override knob to control the speed of execution to the prevent crashes.

Once the program has been started, the Cycle Stop Button (red button) will pause the slides in motion and Cycle Start Button (green button) will restart the program.

NOTE: Always make sure the program is at the correct block number before pressing the Cycle Start button. You can press the Reset Key in Edit Mode to rewind the program to the beginning.

4. Drawbar Actuator & Collet Noses

(1) Drawbar Actuator

The hydraulic rotary actuator operates at speeds up to 8000 RPM. The hydraulic actuator is designed to mount to spindles. The unit will provide a push or pull force capable of opening or closing collets, fixtures, and short stroke power chucks.

(2) Changing Collets

The drawbar has a threaded collar that adapts the drawbar thread to the correct thread for a 5C collet.

To change a collet, follow these steps:

1. Remove the set screw in the collet nose.
2. Using a collet wrench, unscrew the collet from the adaptor.
3. Change the collet size and thread the collet onto the drawbar. (turn clockwise)
4. Replace the set screw.

NOTE: Always clean the threaded adaptor and the collet prior to installation.

(3) Dead Length Collets

The 5C configuration allows the use of 5C dead length collets.

A microcentric dead length collet nose is available for the main spindle.

(4) Step Collets

Oversize head step collets are available for the 5C system.

(5) Expanding Arbors

Expanding arbors are available for the 5C system.



(6) Parts Catcher

There is an optional parts catcher available with the MT that catches the part after it's been cut off and drops it into a basket. If equipped with the parts catcher, make sure that the air pressure valve is set to at least 50 psi.

If the air pressure is below 50psi, twist the air pressure valve clockwise until the gauge reads 50 psi of pressure. The parts catcher is toggled on and off using the M26 and M27 codes respectively.

CHAPTER 4: BASIC PROGRAMMING

A program can be entered in two ways: directly using the onboard editor or from an external PC transferred into the MT's memory.

1. *The Editor*

The on-board editor is very powerful and contains many features. The purpose of this section is to provide an introduction to the editor. Please refer to the Fanuc Oi Mate-TC operating manual for a full explanation of features.

With the MT's on-board editor you can:

- Create new programs.
- Create sub-programs
- Create parametric sub-programs
- Edit existing programs
- Delete existing programs
- Rename existing programs
- Copy an existing program to create a copy that can be edited without changes to the original.
- Copy all or part of a program into an existing program.

(1) Edit Mode

To get into edit mode, press the following:

1. Edit Mode Button
2. Program Function Key

This will show the programs that are currently stored in memory. A screen displaying the currently stored programs is displayed.

The edit mode is used to create new programs and to edit existing ones.

(2) Starting a New Program

To start a new program, first go into edit mode. Press the "DIR" softkey and the screen will show a listing of the existing programs and their associated program numbers. Program numbers begin with the letter "O" and are followed by a 4-digit number.

To create a new program, do the following:

1. Enter "O" followed by a unique program number up to 4 digits long (i.e. O3412)
2. Press the Insert editing key.

The screen will show the program number in the upper right hand corner. The new program has been created and is ready to be edited.

(3) LCD Screen

The LCD screen is the main interface for the MT-SERIES. This screen shows user input and displays the status of the machine. The Continuous Menu Key is used to reveal additional menu items than can be shown on one screen. The Return Menu Key move the menu up one level.

(4) Writing a Program Using Interactive Mode

The interactive program entry mode is very helpful when using canned cycle G codes. The interactive mode prompts you for G code parameters and shows a diagram of the G code. This makes is very easy to format G code.

Once a program has been created and is on screen, the interactive mode is accessed by:

1. Pressing the C.A.P softkey
2. Key in the desired G code.

This will bring up a help menu for the particular G code.

(5) Accessing and Editing an Existing Program

To open and edit an existing program, press this sequence of buttons when in editing mode:

1. O<####> (where <####> represents the program number)
2. Press the "O SRH" softkey

The screen will show the contents of the program.

You can cursor or page up or down keys to scroll through the program.

Use editing keys Insert, Alter, and Delete to modify the program.

(6) Comments

Comments describe what a particular line of code does. You can enter

comments for the MT by placing them between open and closed parenthesis (also known as control-out and control-in characters.)

```
O0005 (BEGIN PROGRAM) ;  
S120 (SPINDLE SPEED 120 RPM) ;
```

For example:

(7) Adding a New Line to an Existing Program

To add a new line to an existing program, take the following steps:

1. Cursor to the End of Block (the semicolon) of a line.
2. Enter the CNC code for the new line.
3. Press the Insert editing key.

The new line will be inserted after the currently selected line.

(8) Deleting Words

To delete one word of code, do the following once the program has been opened:

1. Highlight the word to delete using the cursor keys.
2. Press the Delete editing key

The block of code will immediately be deleted.

(9) Deleting Words

To delete one word of code, do the following once the program has been opened:

1. Highlight the word to delete using the cursor keys.
2. Press the Delete editing key

The block of code will immediately be deleted.

The Alter Key (alteration) replaces characters or symbols under the cursor with characters from the input buffer.

(10) Reset Deleting Lines

To delete a line of code, do the following once the program has been opened:

1. Cursor to the line to delete
2. Press EOB
3. Press the Delete editing key

The line will be immediately deleted.

(11) Copying Blocks to a New Program

To copy lines from one program to another, do the following:

1. Press the EDIT mode button.
2. Press the PROGRAM function key.
3. Press the (OPRT) softkey
4. Press the continuous menu key.
5. Press the EX-EDT softkey.
6. Press the COPY softkey.
7. Cursor to the first line in the range to be copied and press the CRSR~ softkey.
8. Cursor to the last line in the range to be copied and press the ~CRSR or BTTM~ softkey. The BTTM~ softkey will select the last line in the program as the last line of the range.
9. Enter a number for the new program and press the INPUT editing key.
10. Press the EXEC softkey

1. Copying from Another Program

If you want to copy another program in its entirety into the program you're currently working, do the following:

- 1) Press the EDIT mode button
- 2) Press the PROGRAM function key.
- 3) Press the (OPRT) softkey
- 4) Press the continuous menu key.
- 5) Press the EX-EDT softkey.
- 6) Press the MERGE softkey.
- 7) Cursor to the line where the program will be copied and press the ~CRSR softkey
- 8) Enter the number of the program to be inserted and press the INPUT editing key.
- 9) Press the EXEC softkey

The program will be copied at the selected cursor location.

2. Program Management

- 1) Deleting a Program
- 2) Deleting ALL Programs
- 3) Renaming an Existing Program

3. Serial Port (RS-232) Communication

- 1) Serial Port Cable Wiring
- 2) Serial Port Machine Parameters
- 3) PC Serial Port Setup & Connection
- 4) Sending & Receiving a Program from the MT
- 5) Sending a Program to the PC
- 6) Receiving a Program from the PC

4. Memory Card (PCMCIA) Communication

- 1) Transferring Programs by Memory Card
- 2) Machine Parameter for Using the Memory Card
- 3) Copying a Program to the Memory Card
- 4) Copying a Program from Memory Card to the Machine

5. Programming Basics

- 1) Setting the Program Number
- 2) Start-Up Block
- 3) Start-up Block for Constant Surface Speed
- 4) Ending a Tool Sequence
- 5) Ending a Program Sequence
- 6) Active Error Checking
- 7) Canned Cycles Preview
- 8) Long Coding versus Canned Cycles

6. Simulate & Graphics Functions

The simulation and graphics functions on the Fanuc Oi Mate-TC controller are useful for testing new programs. See the Fanuc manual for more details on these functions.

CHAPTER 5: CANNED CYCLES

1. Most Commonly Used Canned Cycles

The most commonly used canned cycles are:

- G71 Roughing in the Z axis
- G74 Hole drilling canned cycle
- G76 Threading canned cycle

Please refer to the Fanuc Oi Mate-TC programming manual for a full list of cycles.

Note: Always precede a canned cycle with a feed rate. This sets the operating feed rate for the canned cycle.

2. G71 - Roughing in the Z axis

Format:

G71 U R

G71 P Q U W F

Note: Both G71 lines are required.

(A) Parameter Definitions

First Line

U	Depth of cut per side
R	Retract amount

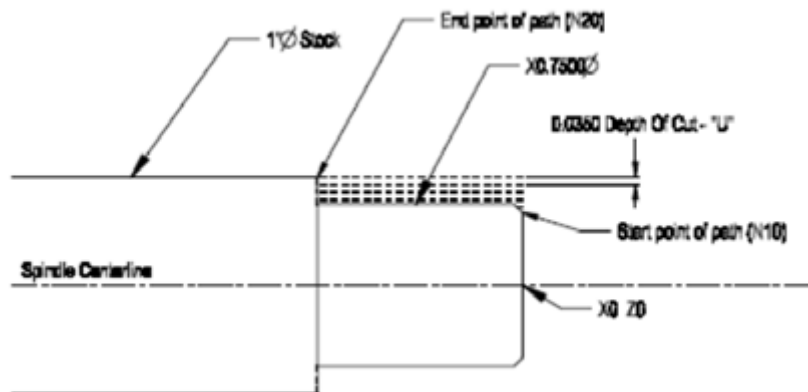
Second Line

P	First block of the profile
Q	Last block of the profile
U	Stock allowance for finishing cut on X axis
W	Stock allowance for finishing cut in Z axis
F	Roughing feed rate

Notes:

1. The profile can only use lines or arcs.
2. Always position the tool outside the material (start point), before calling G71
3. When using G41 or G42 the value of U (depth of cut per side) must be 2.1 times the tool nose radius. (in the example above, if the tool has a 0.03 radius, then U must be at least 0.062)

(B) G71 Roughing Cycle Example



```

O1234 (G71 roughing cycle example);
G00 X1.0200 Z0.0500;
G71 U0.0350 R0.0250;
G71 P10 Q20 U0.0100 W0.0020 F0.0105;
N10 G00 X0.6713 Z0.0500;
G01 Z0.0000 F0.0025;
X0.7500 Z-0.0394 F0.0020;
Z-0.9500 F0.0020;
N20 X1.0200 F0.0100;
G00 Z0.0500;
G70 P10 Q20;
G00 Z0.0500;
    
```

G70 = Finishing pass

G70 P(First block from G71) Q(Last block form G71)

3. G74 - Drilling Cycle

Format:

G74 R

G74 X Z P Q F

Note: Both G74 lines are required.

(A) Parameter Definitions

First Line

R	Rapid clearance amount
---	------------------------

Second Line

X	Diameter for drilling ("X0" in most cases)
Z	Depth of drill in Z
P	Movement amount in X ("0" for drilling)
Q	Depth of peck (No decimal)
F	Drilling feed rate (No decimal)

You cannot provide decimal values to any of the "P" and "Q" parameters.

The P and Q parameters take integer values that specify the number of increments of 0.001

For example the value 1.8 is given to the P and Q parameters as
 $1.8/0.001 = 1800$

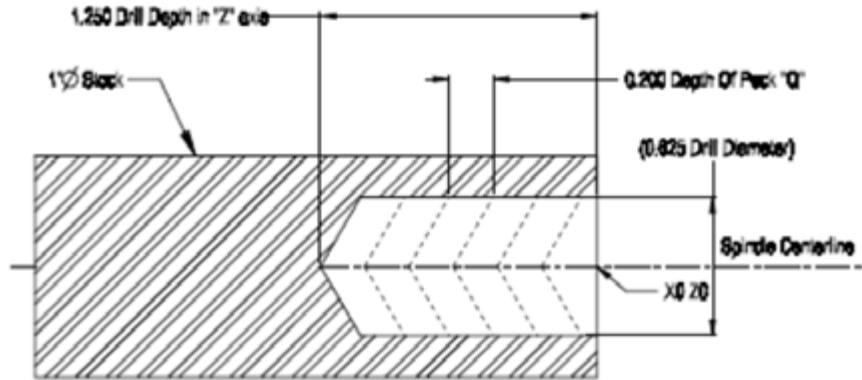
Here are a few more examples for reference:

$0.002 = Q2$

$0.0250 = P25$

The smallest value that can be provided to the P and Q parameters is 0.001 which is entered as P1 or Q1

(B) G74 Drilling Cycle Example



O1234 (G74 drilling cycle example);

G00 X0 Z0.0750;

G74 R0.0500;

G74 X0 Z-1.2500 P0 Q200 F0.0080;

G00 Z0.0750;

4. G76 - Threading Cycle

Format:

G76 Q R

G76 X Z P Q F

Note: Both G76 lines are required.

(A) Parameter Definitions

First Line

Q	Minimum cutting depth
R	Depth of last cut

Second Line

X	Finish diameter (Minor diameter for external; Minor diameter for internal)
Z	Depth of thread in Z
P	Thread height per side (Major - Minor / 2)
Q	Depth of first cut
F	Thread pitch (1 / threads_per_inch)
I	Tapered threads (optional)

You cannot provide decimal values to any of the “P” and “Q” parameters.

The P and Q parameters take integer values that specify the number of increments of 0.001

For example the value 1.8 is given to the P and Q parameters as
 $1.8/0.001 = 1800$

Here are a few more examples for reference:

0.002 = Q2

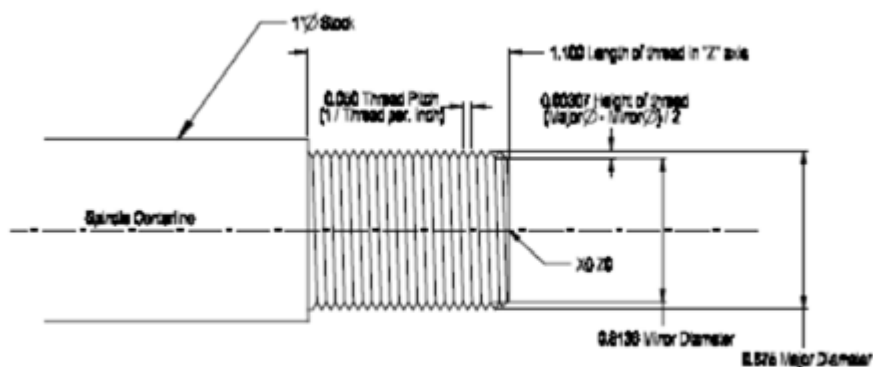
0.0250 = P25

The smallest value that can be provided to the P and Q parameters is 0.001 which is entered as P1 or Q1

Notes:

1. For straight heads use the same value as X. For tapered threads, use the I parameter.
2. P and Q parameters don't accept decimal numbers
3. The feed rate override knob does not adjust the feed rate during threading.

(B) Threading Cycle Example



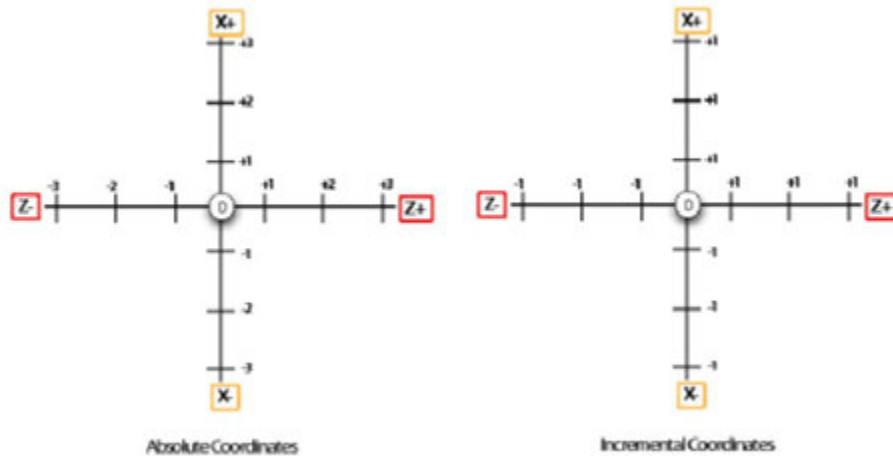
Example Size: 7/8 - 20 - 2A

Minor Diameter = $X - (Q * 2)$

```
O1234 (G76 threading cycle example);
G00 X1.0000 Z0.1500;
G76 Q2 R0.0020;
G76 X0.8136 Z-1.1000 P31 Q10 F0.0500;
G00 Z0.1500;
```

CHAPTER 6: M&G CODE

1. Coordinate Types (Cartesian)



2. Absolute Position

Absolute positioning uses a known fixed zero point as a reference for the commanded positions.

The MT using the Fanuc Oi Mate-TC controller doesn't utilize the G90 code for specifying absolute position. Instead the X, Z, and C address words are used to command an absolute position move.

A move from X0 Z0 of X1.0 Z1.0 would move the tool to the position X1.0 Z1.0. If a command of X3.0 Z-2.0 is made the tool will move to the grid position X3.0 Z-2.0. This is the normal method of programming and directly related to most machine prints.

The grid has a specific location for any point and the X and Z locations call these points.

3. *Incremental Position*

Incremental coordinates use the last position of movement to define the next move.

The MT using the Fanuc Oi Mate-TC controller doesn't utilize the G91 code for specifying absolute position. Instead the U (X-axis), W (Z-axis), and H (C-axis) address words are used to command an absolute position move.

If a move from X0 Z0 of U1.0 W1.0 is commanded, the new coordinate would be X1.0 Z1.0 If the next move was U2.0 W1.0 the new coordinates would be X3.0 Z2.0 (X2.0+U1.0)(Z1.0+W1.0)

As you can see this uses the last position as the next position zero point.

This option is very useful when programming the same feature on a part a number of times like grooves or a grid of holes when using a live tool. You would not have to define a new zero position as each call moved from the last absolute commanded position.

4. *Radius and Diameter Programming*

A program can be created using either Radius or Diameter coordinates, but cannot be switched between types in a program. The machine parameters must be set for the type of programming method required.

The most common method of programming a lathe is in diameter mode. This method is for all programs and explanations throughout this manual.

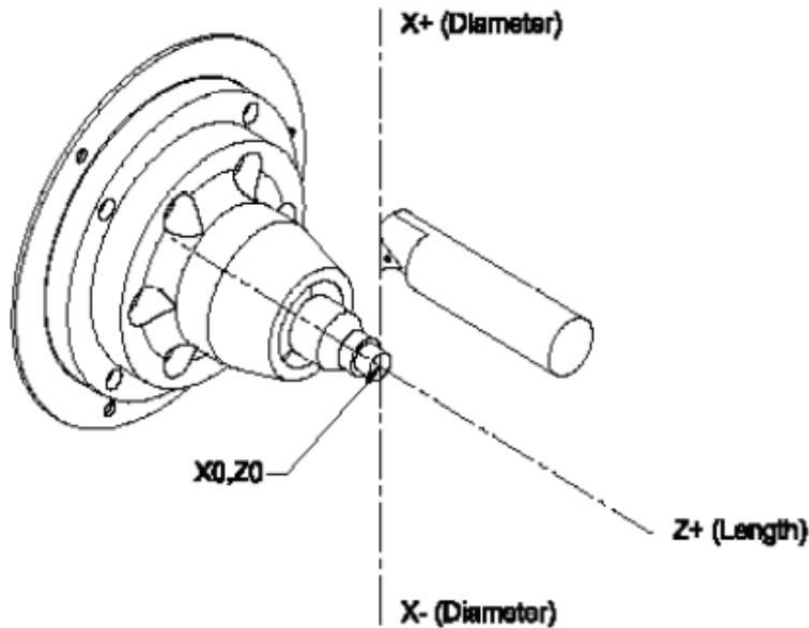
The most basic difference is that in Radius Mode, a required part diameter of 1.0" would be called out as X.5(Half the diameter as that's the definition of radius)

In Diameter Mode, the same part dimension would be called as X1.0

As you can see, this will make your programs much easier to read and troubleshoot.

In some of the canned cycles, values are requested as radius values. This is to assist in the ease of programming.

5. Machine Coordinate System



(1) Axis Direction

The axis direction on a lathe is controlled by a variety of parameters in the controller. These parameters can be set to allow different axis directions. The most often changed axis is the X direction. The Z axis direction is normally not changed.

(2) Tool Hand Directions

When using a gang tooled lathe both left and right-handed tools can be used to machine parts. The advantage of this is the ability to set tools so that fast cycles can be achieved. Most tools used on a gang tooled lathe are boring bars as this allows more tools on the tool plate and the ability to adjust the length of the tool to suit the job being machined.

The MT is setup so that a right-handed boring bar when located with the insert face up is programmed with a positive X axis coordinate. Offsetting the tool for wear is correct (X+ offset and the diameter increases)

Drilling takes place on the X axis centerline and is unaffected by the axis direction. Most live tools are setup at the top of the table and again allow positive X axis values for positive offsets.

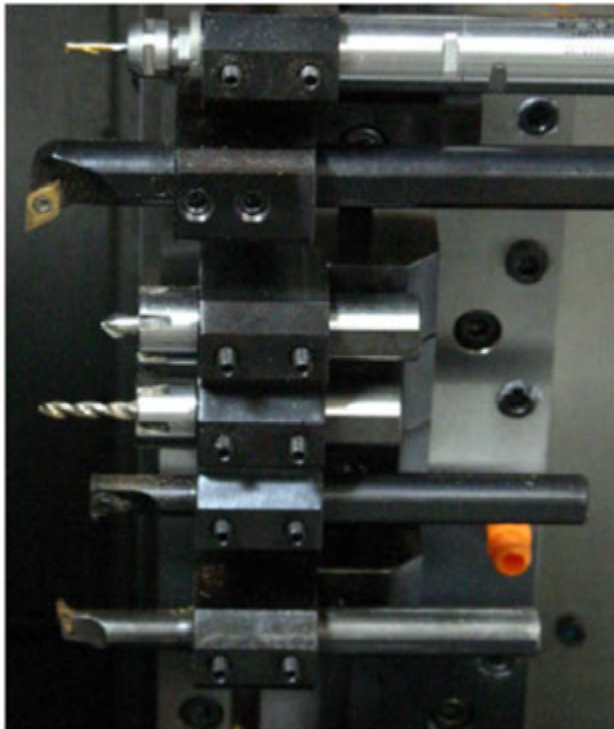
When a program example is given, it will assume this format. When threading,

any special needs will be explained with examples.

(3) Construction of the Program

After the proper coordinates have been assigned to the drawing, the programming consists of turning on or off the proper accessories and sending the tools on the path you have determined necessary to complete the part.

A rough layout of the tools on the slide or on a piece of paper should be completed before you start to write the program. Take into consideration the length of the tools and the proximity of one tool in respect to another while it is cutting the parts. Always leave room for the various tools. After a tool is clear of the workpiece, the exit move should always be in the Z+ direction to avoid crashing other tools in line of the slide. Determine the tool number for each tool that you are going to use then you are ready to start writing the program.



6. M&G Code Listing

(1) M Codes

M codes are commands that turn on or off mechanical switches. These can be relays or programmed operations.

The M code groupings are:

M00	M30 Standard M codes.
M31	M99 Optional M codes that depend on the machine
M100	up Customizable functions used for parametric programming and macros.

(2) Standard M Code List

M00	Program Hold
M01	Conditional program stop
M02	End of program
M03	Spindle forward (clockwise)
M04	Spindle reverse (counterclockwise)
M05	Spindle stop
M07	Mist coolant on
M08	Flood coolant on
M09	Coolant off
M10	Chuck/collet close
M11	Chuck/collet open
M19	Orient spindle (indexing)
M30	Program end (rewind)

(3) M Code Description

M00	Program Stop
<p>The M00 code is used to stop operation of a program. It causes the spindle and coolant to be switched off. A cycle start will resume operation of the program from the cursor position.</p>	

M01	Optional Program Stop
<p>The M01 code is used to stop the operation of a program selectively by activating the optional stop button. The spindle and coolant are not switched off. A cycle start will resume operation of the program from the cursor position.</p>	

M02	End of Program
Program end.	
M03	Spindle Forward
The M03 code will start the spindle in a clockwise direction at the programmed spindle speed. The machine waits until the spindle reaches a percentage of the programmed speed. Direction of the spindle is determined by standing at the left side of the machine and looking towards the tool slide M03 is considered clockwise.	
M04	Spindle Reverse
The M04 code will start the spindle in a counterclockwise direction at the programmed spindle speed. The machine waits until the spindle reaches a percentage of the programmed speed. Direction of the spindle is determined by standing at the left side of the machine and looking towards the tool slide M04 is considered clockwise.	
M05	Spindle Stop
The M05 code stops the spindle.	
M07	Mist Coolant On
Starts mist coolant pump.	
M08	Flood Coolant On
Start flood coolant pump. This code is activated at the end of a block. The coolant switch, if OFF, overrides the coolant pump activate.	
M09	Coolant Off
Switch off the main coolant pump.	
M10	Chuck/Collet Close
Retract the draw bar into the spindle (close the collet or chuck).	
M11	Chuck/Collet Open
Push out the draw bar from the spindle (opens the collet or chuck).	

M19	Spindle Orientation
------------	----------------------------

Enable the spindle for indexing.

M30	Program Rewind
------------	-----------------------

End the current program. It will stop the spindle and turn off the coolant. The cursor will be reset to the beginning of the program.

(5) G Codes

This is a full list of available G codes for the Fanuc Oi Mate-TC. The MT utilizes version “A” of the Fanuc G code system.

Not all G codes are explained in this manual. See the Fanuc manual for more details.

G Code	Description
G00	Positioning (Rapid traverse)
G01	Linear Interpolation (Cutting feed)
G02	Circular Interpolation CW
G03	Circular Interpolation CCW
G04	Dwell
G07.1 (G107)	Cylindrical interpolation
G10	Programmable data input
G11	Programmable data input cancel
G12.1 (G112)	Polar coordinate interpolation mode
G13.1 (G113)	Polar coordinate interpolation cancel mode
G18	ZpXp plane selection
G20	Input in inch
G21	Input in mm
G22	Stored stroke check function on
G23	Stored stroke check function off
G27	Reference position return check
G28	Return to reference position
G30	2nd, 3rd and 4th reference position return
G31	Skip function
G32	Thread cutting
G34	Variable-lead thread cutting
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G50	Coordinate system setting or max. spindle speed setting
G50.3	Workpiece coordinate system preset
G52	Local coordinate system setting
G53	Machine coordinate system setting
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection

G Code	Description
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G65	Macro calling
G66	Macro modal call
G67	Macro modal call cancel
G70	Finishing cycle
G71	Stock removal in turning
G72	Stock removal in facing
G73	Pattern repeating
G74	End face peck drilling
G75	Outer diameter/internal diameter drilling
G76	Multiple threading cycle
G80	Canned cycle for drilling cancel
G83	Cycle for face drilling
G84	Cycle for face tapping
G86	Cycle for face boring
G87	Cycle for side drilling
G88	Cycle for side tapping
G89	Cycle for side boring
G90	Outer diameter/internal diameter cutting cycle
G92	Thread cutting cycle
G94	Endface turning cycle
G96	Constant surface speed control
G97	Constant surface speed control cancel
G98	Per minute feed
G99	Per revolution feed

(6) Preparatory Functions

G codes are the basic codes that command a CNC controller to move, or prepare the control for a type of move, or set the control in a mode to be able to interpret a method of operation.

(7) Modal Operation

The advantage of modal G codes is that you do not have to type or enter a repeated code on every line. In this example, the modality of the G01 code is shown.

Code 1 (Non-Modal)	Code 2 (Modal)
G00 X1. Z.1	G00 X1. Z.1
G01 Z0	G01 Z0
G01 X1.5 Z-.5	X1.5 Z-.5
G01 Z-2.	Z-2.
G00 X3. Z1.	G00 X3. Z1.

In the modal version, the G01 code is only needed on the first line. The subsequent lines automatically use the G01 setting until a new a setting is made.

(8) G Code Format

The format for a G code can be G00 or G0. Both will work with the controls.

(9) G00 Rapid Travel

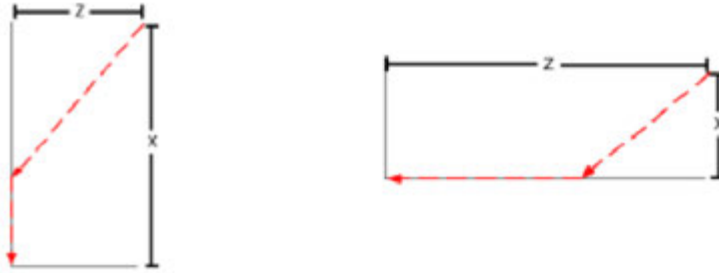
This code causes the machine motion to occur at the maximum traverse rate of the machine tool.

When only one axis is programmed the move will be a straight line. When both axis move simultaneously, each axis moves at its maximum speed, until one axis end point arrives at the programmed location. In the case of an X and Z rapid move, the axes would move along a 45 degree angle until either axis reached its programmed location. The other axis would continue to its programmed location.

In the figure below, the left example shows a rapid move when the X axis move is greater than the Z axis move.

The figure on the right shows a rapid move when the Z axis move is greater

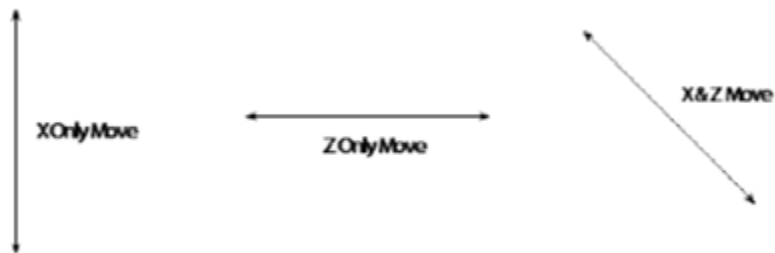
than the X axis move.



(10) G01 Linear Interpolation

Linear interpolation is a mode of movement in which one or two axes move simultaneously to the programmed end point. The programmed feed rate in moves with X or Z motion are maintained along the line determined by the X and Z dimension words.

A single axis move moves the machine in one direction only. Either X or Z. These moves are considered axis parallel. When a taper is to be programmed both axes must be commanded.



(11) G02 Circular Interpolation Arc Clockwise

A clockwise arc generated by the coordinated motion of two axes. The arc is defined as G2 when the tool motion, relative to the work piece, is clockwise. An arc is defined with the start point, the end point, the incremental distance from the start point to the center of the arc (I and K), and the direction of the arc, or an R value equal to the required radius.

In the example to the left either an R value or an I value both equal to the radius would be programmed, K is zero, and it is recommended that it is included in the line.

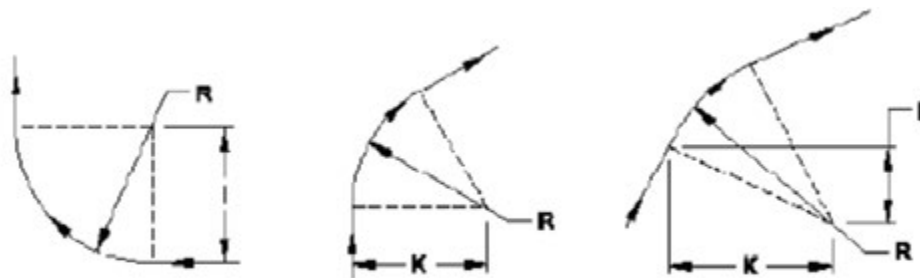
In this example, in the middle either an R value or a K value both equal to

the radius would be programmed. I is zero and it is recommended that it is included in the line.

In the example to the right either an R value or both I & K values are required.

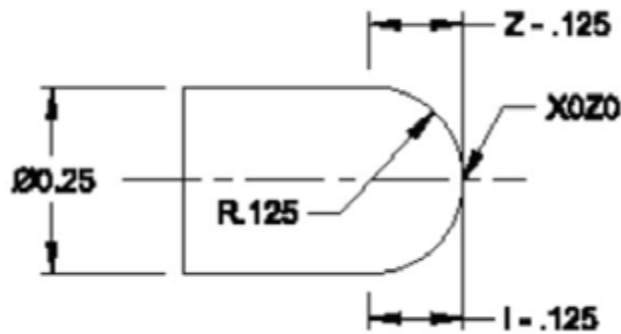
Remember I & K values are distances from the start position to the center of the radius and can be positive or negative.

In G68 roughing cycles, arcs must be programmed using I & K.



Example of a 1/8" radius from the X0 Z0 centerline:

```
N30 G2 X.25 Z-.125 R125
OR N30 G2 X.25 Z-.125 10 K-.125
```

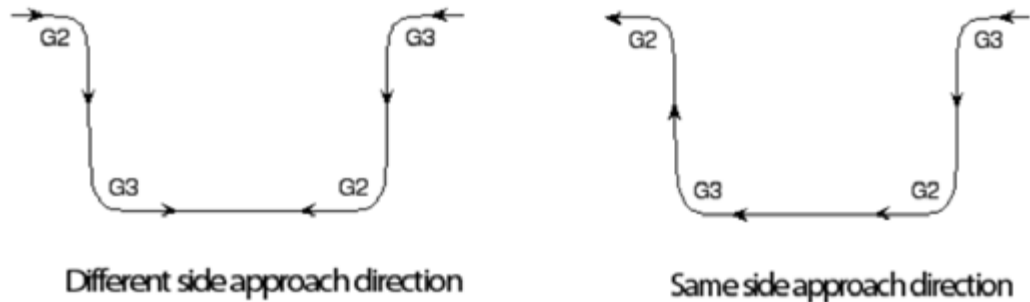


(12) G03 Circular Interpolation Arc Counterclockwise

A counterclockwise arc is generated by the coordinated motion of two axes. The arc is defined as G3 when the tool motion, relative to the work piece is counterclockwise. An arc is defined with the start point, the end point, the distance from the start point to the center of the arc (I & K), and the direction of the arc, or an R value equal to the required radius. The examples in G2 can be used to explain the I, K and R values.

(13) Directions When Programming Arcs

When programming external or internal arcs, the basic rule of clockwise and counterclockwise always applies. The same arc can be programmed with either a G02 or G03 depending on the direction of the tool motion. Grooving is good example of this



(14) G04 Dwell

A timed delay of programmed duration. The time is specified using the P word whose value is in 0.001 seconds. You cannot provide a decimal point value to the P word. See the following examples on how to specify dwell values:

CODE	DESCRIPTION
G04 P2000	Dwell 2 seconds
G04 P500	Dwell 0.5 seconds
G04 P2	Dwell 0.002 seconds

The smallest time increment that you can specify is 0.001 seconds and is given by the code G04 P1

(15) G20 Inch Programming

Programming in inches. The machine is setup in inch mode. This code is not required.

(16) G21 Metric Programming

If you wish to program in millimeters, use G21 at the beginning of the program.

Please refer to the parameter tables and change the machine coordinate system to metric if you wish to have the machine display change also.

(17) G28 Home Search

This code sends the programmed axis to its home position.

Example: G28 X0 Z0;

This example will move the slides to their home positions on the X and Z axis.

(18) G40/G41/G42 Tool Nose Radius Compensation

These G codes allow the actual part profile to be entered as the program. Editing the tool table for radius rather than changing the program can accommodate any variation in tool nose radius.

The main rules are that the proper tool location codes are installed on the tool-offset page and that the entry and exit moves are at least double the radius value entered in the table.

(19) G54-G59 Workpiece Coordinate Systems

These are position offsets for moving the tool offset table by a fixed amount.

They can be used for cutting multiple parts.

If a part is to be turned around in the collet and have a 2nd operation completed, the G55 offset could be used to provide a 2nd Z zero location for the 2nd operation. Without changing the tool offsets used for the 1st operation.

A positive offset moves the tool away from the spindle.

A negative offset moves the tool towards the spindle.

(20) G70 - G76 Repetitive Canned Cycles.

See the Fanuc Oi-Mate TC manual for details.

(21) G80 - G89 Drilling Canned Cycles

See the Fanuc Oi-Mate TC manual for details.

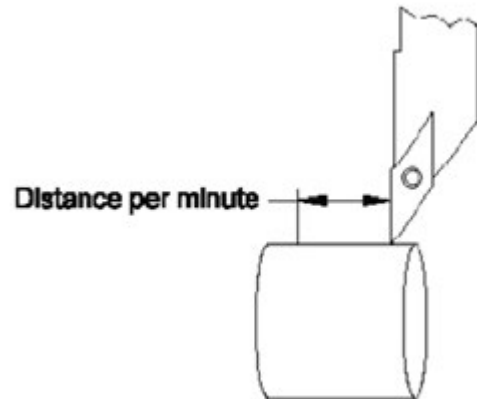
(22) G98 Feed Rate Specification IPM, MPPM

This mode allows the feed rate to be specified by millimeters per minute or inches per minute. Normally this is used when no spindle speed is active and machine motion has to be controllable (not rapid). One example of this is bar feeding.

This method of programming is the recommended method when using a live tool. No decimal point is allowed.

F80 = 8 inches per minute

F800 = 80 inches per minute



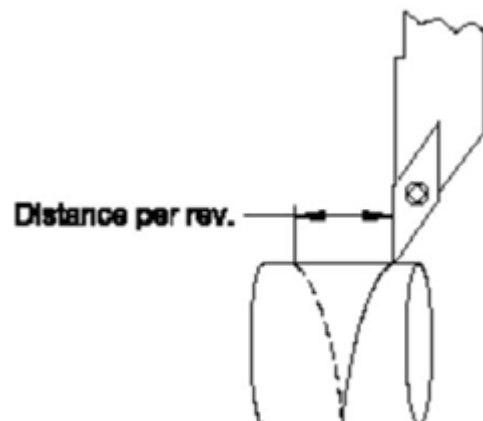
(23) G99 Feed Rate Specification IPR, MMPR

This mode allows the federate to be controlled in relation to the commanded spindle speed.

The feedrate is based on a feed per revolution (FPR)

In inch mode, the federate would be programmed as:

F.002(0.002" per rev)



Or in metric:

F.02(0.02mm per rev)

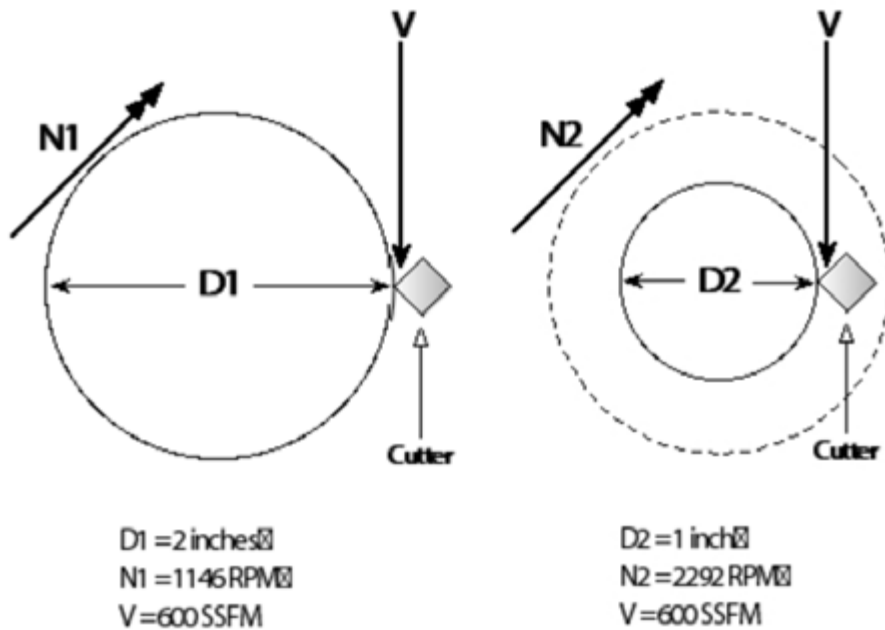
In order to program finer federates than the minimum allowed by G99, it may be necessary to switch to G98 and specify a small feed in inch per minute to achieve a similar slow feed rate.

Example: 3000rpm x 0.15" per minute = 0.00001" per rev.

(24) G96 Constant Surface Speed Mode (CSS)

This code controls cutting speeds based on surface speed and diameter being cut. A given surface speed will produce a variable RPM depending on the diameter programmed.

The following example demonstrates this:



$$\text{RPM}(N) = \text{SSFM}(V) * 3.82 / \text{Work Diameter } (D)$$

Given the same SSFM(V) 2 different diameters will give you 2 different speeds in RPM:

600 SSFM at a 2" diameter = 1146 RPM

600 SSFM at a 1" diameter = 2292 RPM

If a 2" part was to be faced to a 1" diameter using CSS the machine would increase RPM from 1146 RPM to 2292 RPM gradually as the cut decreased in diameter.

If the same calculation is used on a .1 diameter you can see that the RPM exceeds the spindle capabilities of the machine.

$$\text{RPM} = \text{SSFM} * 3.82 / \text{Work Diameter}$$

$$600 \text{ SSFM at a .1" diameter} = 22,920 \text{ RPM}$$

This is the reason why a G50 (max spindle speed) code is required when using CSS.

Example:

G50 S4500 ; Max speed preset

G96 S250 M3 ; SSFM and spindle on

(25) G97 Constant Surface Speed Cancel

This code cancels constant surface speed mode that was set by the G96.

7. Spindle Orientation

To activate the spindle as a vector spindle capable of indexing, issue a M19 to the control for each index you require. An S value is also required to specify where to index.

CODE	DESCRIPTION
M19 S0	This will index to the marker pulse location.
M19 S90	This will index clockwise to the 90° position.
M19 S-90	This will index counterclockwise to the 90° position.

In this example, 3 holes are to be drilled at 120 degrees to each other.

The drilling moves are located in a sub-program at N1000 to N1025

G98 feed in inches per minute is programmed as no spindle speed is being used.

```

%00101,MX,
N01 T7 M5 M8           ; 0.068 Cross Drill
G00 X0.6
Z0.1
M19 S60                ; 1st spindle position
G01 Z-1.0 F250         ; Index to 1st position
(RPT N1000,N1001)     ; Drill 1st hole
M19 S180               ; 2nd spindle position
(RPT N1000,N1001)     ; Drill 2nd hole
M19 S300               ; 3rd spindle position
(RPT N1000,N1001)     ; Drill 3rd hole
G00 Z2.0 M5
M30
N1000 G01 X0.1         ; Cross drill sub
G01 X0 F20.
G00 X0.1
X0
G01 X0.1
N1001 G00 X0.6 F250.   ; Safe index position
  
```

To program a spindle movement with an axis movement requires a full C axis spindle.

8. Other Addresses

(1) D - Address

The D word is used for tool offsetting. Please refer to tool offsetting.

(2) F - Feed Rate Address

This code is used to control how fast or slow the machine executes an axis move. Depending on the G code selected, this code can be in inches per revolution or in inches per minute.

Feedrates are modal commands and once an amount is commanded, that feed rate is active until a new feedrate is programmed. This number can be up to 4 decimal places (F.0001)

Normally in lathe programming the setting is for inches per revolution.

Example:

```
N1 G1 X.5 Z-.1 F.002 = .002"/rev
```

```
N2 G1 X.5 Z-.1 F.010 = .010"/rev
```


Line N1 commands a feed of .002 inches per revolution.

Line N2 commands a feed of .010 inches per revolution. This feed rate is 5 times faster than line N1.

(3) I & K - Incremental Value for Position of Radius Center Point from Start Point

See the explanation of I and K in G2 circular interpolation.

(4) N - Block Numbers

Block numbering is optional. The 8055iTC control has an auto-numbering feature that will add numbering to new lines that are input.

(5) P - Address

The P word is used to specify a time delay in execution. The value to the P word is an integer specifying the delay amount in hundredths of a second. For example: P100 = 1 second; P30 = 0.3 seconds

(6) A - Address

The A word is used for angular programming. Please refer to the Advanced Programming section for more details.

(7) R - Address

The R word is used for programming radius values in G2 and G3.
Example: X1.0 R-.05

(8) S - Spindle Speed Address

This is a 4-digit number that is used to specify the spindle speed or surface speed required.

See G96 and G97 for specific details.

(9) T - Tool Number Address

This is a number between T0 and T100 that is used to specify the tool required.

The format for this number is T###. The digits specify the tool offset.

The tool table sets an offset value to the tool number using a D address. This address is linked to the tool number. T1 calls offset D1 etc, through

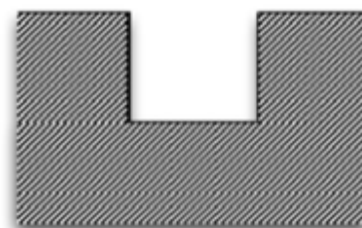
T100

(10) Using Dissimilar Tool Offset Numbers and D Offsets

These numbers do not have to correspond. On most machines, 100 offsets are provided. This allows you to assign the extra offset values to multiple tool offsets.

Here's an example where you need to provide 2 offsets to a groove tool.

T5 (D5) would be the left corner and T5(D15) would be the right corner. This allows the operator to adjust the width of a groove without changing the program.



(11) X, Z, & C - Absolute Addresses

These addresses command axis movement in absolute mode. This is the recommended style of programming. X is a diameter designation and Z is linear.

(12) U, W, & H - Incremental Addresses

These addresses are used to command axes moves in incremental mode. The U, W, & H addresses correspond to the X, Z & C axes.

CHAPTER 7: TOOL NOSE COMPENSATION

1. G40, G41, G42 Compensation Codes

The G40, G41, and G42 codes define how the machine calculates tool nose radius compensation (TNRC).

2. G40 Compensation Code

G40 cancels the active G41 or G42 mode.

3. G41 & G42 Compensation Codes

G41 provides right side tool nose radius compensation. G42 provides left side tool nose radius compensation.

4. R and F Compensation Registers

In the tool offset pages there are registers for X and Z, I and K, and R and F.

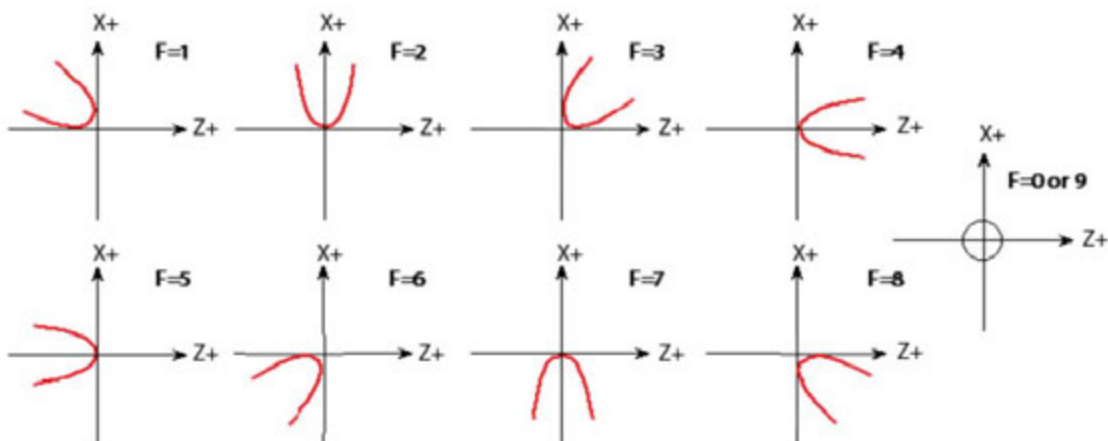
The R and F registers are critical to the correct function of TNRC.

R is the register for the radius of the tool tip. All commercially available tools have a radius. It may be as small as 0.004" or as large as 0.500". This tool nose radius is the reason for TNRC.

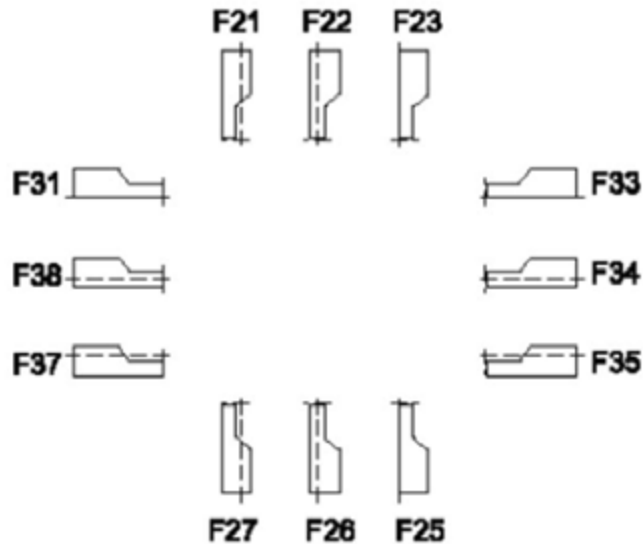
The F register is used to help the control decide how to compensate for the radius. If the control knows how to apply the direction, the program will produce the correct shape.

(1) Defining F Register Values

There are 9 initial values for the F number 0-9. (0-9 mean the same value, tool radius center).



These codes are defined by the X axis positive direction. If this is reversed the F code values change.



As this figure shows, the numbers correspond to a position around an XZ reference point.

In general use the tool codes to represent the following tool types:

1. F1 would be the right side of a grooving tool or a back turning tool on X+
2. F3 would be an outside turning tool on X+ or a boring bar on X-
3. F4 would be a drill or facing tool with the center on the middle of the radius.
4. F5 would be a boring bar on X+ or an outside turning tool on X-
5. F0 or F9 are used when programming the tool radius center.

NOTE:

When threading you would not use TNRC and therefore do not need an F code for the tool tip. It is a common mistake to apply an F2 or F6 to a threading tool tip.

(2) Defining F Register Values for Groove Tools

The software for the Oi Mate-TC controller allows the designation of groove tool edges.

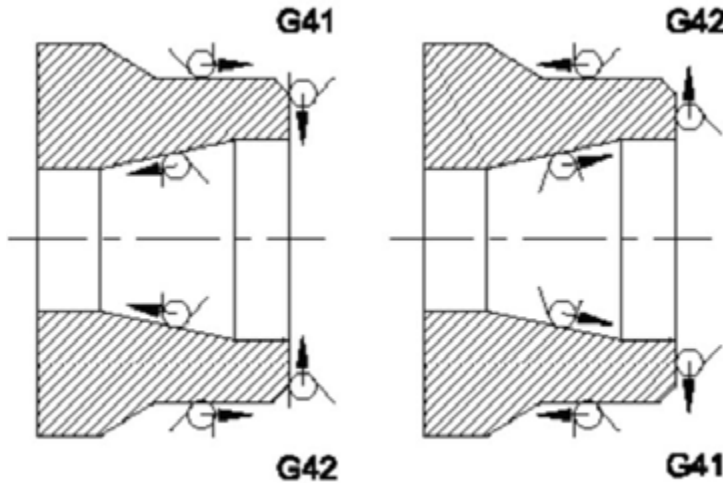
These codes are intended for better visual aid in proofing programs and for correct application in grooving and roughing canned cycles.

The controller is capable of compensating for both width and radius in

roughing cycles.

5. Compensation Directions

When machining a profile to the actual drawing dimensions, the control has to calculate the amount of change that is required to compensate for TNRC. It also needs to know on which side of the profile it has to compensate.



As seen in this figure, profiling towards the chuck on the X+ side would require a G42 code with a value of F3. While boring on the X+ side would require a G41 code with a value of F5.

If a profile was generated on the Xside, the G41 would be used for an outside profile but would require a value of F5 while a boring bar would require a G42 with a value of F3.

6. Entrance and Exit Amounts for TNRC

For TNRC to be active, the controller requires that the approach distance be at least 2 times the radius value.

For a complete explanation of limits with TNRC, please refer to the Fanuc Oi Mate-TC programming manual.

CHAPTER 8: MAINTENANCE

1. *Daily Checkup and Maintenance*

This chapter describes the maintenance operations required to keep the MT running safely and accurately. Please observe these rules and perform the “Startup” and “Closing” checks daily to insure operator safety and minimize the chances of failure.

If a part needs to be replaced, make sure you restore the other parts to their normal operating order as well.

	WARNING
	When adjusting or restoring each part of the machine be sure to turn off the main disconnect switch beforehand. Never turn on the power with the safety guards or protective covers removed.

2. *Forbidden Rules (CAUTION)*

- Don't supply deteriorated lubricants and hydraulic oil.
- Don't use other lubricants and hydraulic oil than specified.
- Don't leave the cutting tools loosely tightened.
- Don't overhang the cutting tools unnecessarily.
- Don't apply excessive force to tighten bolts, etc.
- Don't remove the safety devices without proper reasons.
- Don't use a spanner as a hammer.
- Don't hit workpiece burrs with a file.
- Don't operate switches and pushbuttons with wet or oily hands.
- Don't forget to turn off the main disconnect switch in the event of a power failure.
- Don't forget to check the center alignment of all equipment after an earthquake.
- Don't overhaul each part unnecessarily.
- Don't neglect to clean each part and accessory.
- Don't use accessories from other machines.
- Don't start operations with the safety guards removed or fitted temporarily.


- Don't forget to check the pneumatic and hydraulic pressure with pressure gauges.
- Don't start operation without understanding the characteristics and operating cycles of each equipment.
- Don't neglect to carry out a non-load running test to check actuation of each part before automatic operation.
- Don't neglect to perform warm-up operation.
- Don't start operation without confirming the setting positions of the select switches.
- Don't neglect to take care of lubrication.
- Don't try to load and unload workpiece beyond specifications.
- Don't try to carry out maintenance around hazardous areas of equipment while machining is operating.
- Don't put your hands inside the safety guards or protective covers while machine is operating.
- Don't turn off the main power without performing the reference point or home position returns on each equipment.
- Don't turn off the main power with workpiece clamped on the spindle chuck (except when a collet chuck is used)

3. Start-up Check

Perform the following operations EVERYDAY before starting up the machine and during the warm-up operation:

- Check that the mounting bolts of the spindle chuck and jaws, attaching bolts of the tool holders and cutting tools, etc are not loose.
- Check that the cutting chips do not remain on the spindle chuck, table, etc inside the machine.
- Check that an excessive amount of cutting chips do not accumulate in the chip pan (if chip conveyor is not used)
- Check the lubricating oil level in the lubrication pump tank.
- Check the hydraulic oil level in the hydraulic pump tank through its level gauge.
- Check the amount of coolant in the coolant tank through its level gauge.
- Check that the oil level and oil dropping rate of the air lubricator on the air control unit are proper.
- Check that air filter on the air control unit is not filled with water.
- Check that the air source pressure is within the specified range.
- Check that the hydraulic pressure for the hydraulic chuck is regulated correctly.
- Check that all safety guards are fitted securely.

- Check that the surroundings of the machine and the machine itself are cleaned and put in order.
- During the warm-up operation, check that the machine does not produce any abnormal noise from the CNC feed units, headstock and other parts.
- During the warm-up operation, check that the machine does not vibration at the CNC feed units, headstock and other parts.
- After completing the warm-up operation, check all CNC axes are returned to their respective reference points.
- Make sure all items of start-up checkup and setup are completed before starting production run.

	<p style="text-align: center;">WARNING</p> <p>When a trouble or malfunction is found during the start-up checkup, immediately halt the machine operation and take the proper remedial measure. The machine operation must not be restarted until the cause of the trouble is completely eliminated.</p>
---	---

4. *Closing Check*

When closing for the day, carry out the following maintenance and checks around the machine:

- Brush cutting chips off the spindle, table etc.
- Scrape out cutting chips from the chip pan (if chip conveyor is not used)
- Check that no workpiece is clamped by the spindle chuck (except when a collet chuck is used).

If the power is turned off while a workpiece remains clamped to the spindle chuck, the workpiece may fall off soon as the machine loses hydraulic pressure.

5. *Lubrication*

All slideways, rotating and sliding parts, and other moving sections of the machine **MUST** be constantly lubricated with the correct amounts of clean lubricants. Hydraulic oil pressurized by the hydraulic pump unit is used to actuate the hydraulic collet/chuck unit.

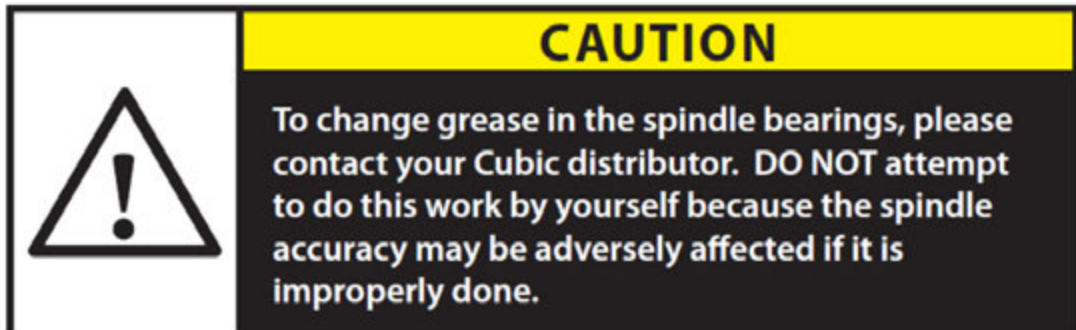
When conducting the start-up check, be sure to measure the oil level in the lubrication pump tank and hydraulic pump tank as well as other lubricant filling sections, and supply the recommended lubricants and hydraulic oil as required.

Additionally, changing the total amount of oil in the hydraulic pump tank is recommended since hydraulic oil gradually deteriorates with the progress of machine operation.

(1) Headstock

The headstock adopts the spindle drive system with pulley and belts instead of gears so that only the spindle bearings need to be lubricated. In fact, the spindle bearings are pre-charged with an appropriate amount of high quality grease which has been selected after careful examination of grease life, heat on the bearings, and other properties.

Because the life expectancy of this grease generally extends to about 5000 or 8000 hours of operation (though this period may depend on the running conditions of the machine, the ambient temperature, etc) a daily check of the headstock lubrication is not necessary.



(2) Slideways

The X-axis and Z-axis slideways are automatically lubricated with oil supplied by the lubrication pump through the oil feed piping. This lubrication pump is activated when the machine powers on and thereafter it operates intermittently to feed a constant amount of oil. Check the oil level in the lubrication pump tank daily, and refill the recommended oil as necessary through the oil filter port.

Note: Clean the outlet port filter and suction filter of the lubrication pump every three months.


(3) Other Elements

Be sure to oil or grease the air lubricator and hydraulic pump tank.

(4) Recommended Oils and Filler Ports

This is a list of the recommended oils and lubricants for each oiling section. Check the oil level daily and supply oil up to the proper level as required. Lubricating and hydraulic oils need to be replaced at the specified intervals for stable and accurate machining.

Oiling section	Oil supply/change needed	Oil amount/capacity	Recommended oil
Lubrication Pump	Supply oil as required	2 liter	MOBIL Vactra Oil No. 2 or equivalent
Hydraulic Pump Unit Tank	Supply oil as required. Complete Change out every 6 months.	3 gallon	MOBIL DTE 24 or equivalent
Air Lubricator (if equipped)	Supply oil as required.	0.15 liter	MOBIL DTE 24 or equivalent
Chuck Master Jaws (if equipped)	Supply grease once a day	Proper amount	Grease as specified by manufacturer
Chip Conveyor (if equipped)	Supply grease once a month	Proper amount	Grease as specified by manufacturer

	CAUTION
	<p>Using lubricants or hydraulic oil not meeting the specifications above may cause a chemical reaction with coolant resulting in corrosion of the mechanical parts and/or clogging in the oil feed piping.</p>

(5) Lubrication Pump



(6) Oil Level Check

When performing the start-up check, check the oil level in the lubrication pump tank with the level gauge. If the oil level is low, remove the oil filter cap and pour the appropriate amount of oil into the pump tank.

Note: When the level of the remaining oil in the pump tank goes below 1/3 of the level gauge, be sure to refill the tank.

(7) Adjustment of Oil Discharge

To adjust the lubrication pump unit, change the feed pressure setting. By default the pump pressure is set at 2 kg/cm², this can be adjusted by turning the knob on top the lubrication pump unit clockwise or counterclockwise.

(8) Cleaning of Oil Inlet Filter

1. Remove the cap of the inlet filter.
2. Take out a filter inside the inlet filter, wash it with clean machining oil and blow compressed air through the filter.
3. Set the filter and secure the filter cap.

Note: Clean the filter every 3 months.

(9) Cleaning of Suction Filter

1. Remove the upper cover of the tank and take out the suction filter assembly.
2. Detach the clip, remove the suction filter and wash it with clean machine oil.
3. Set the suction filter and secure the clip.
4. Replace the upper cover

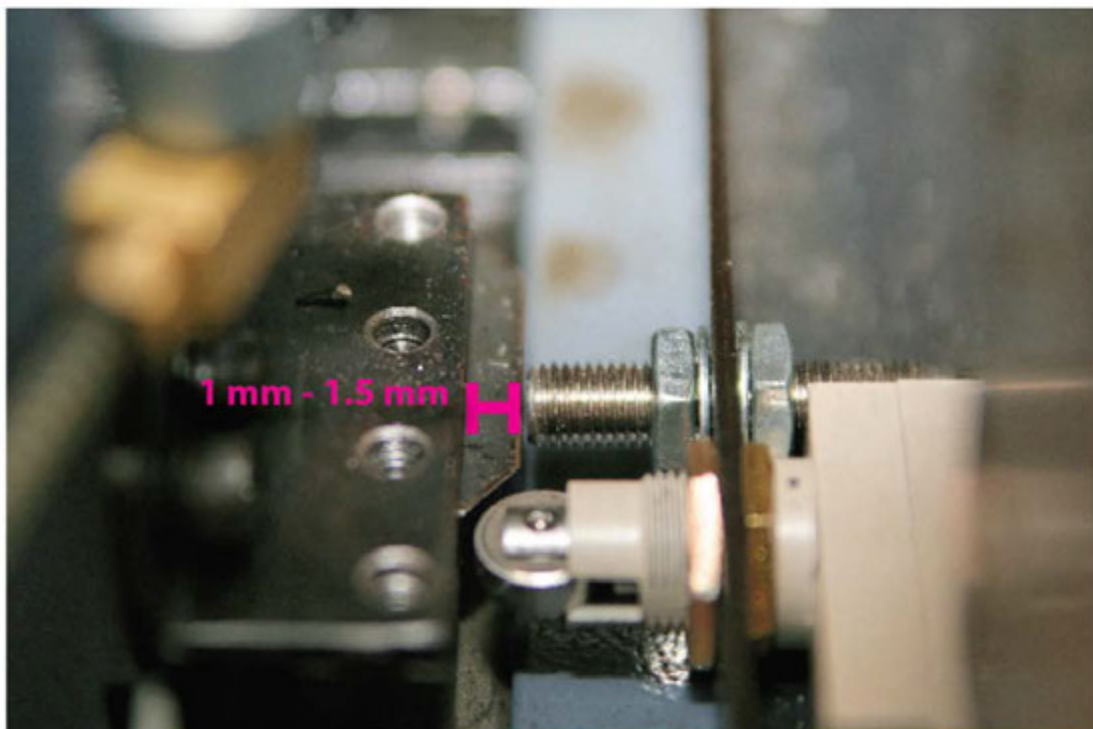
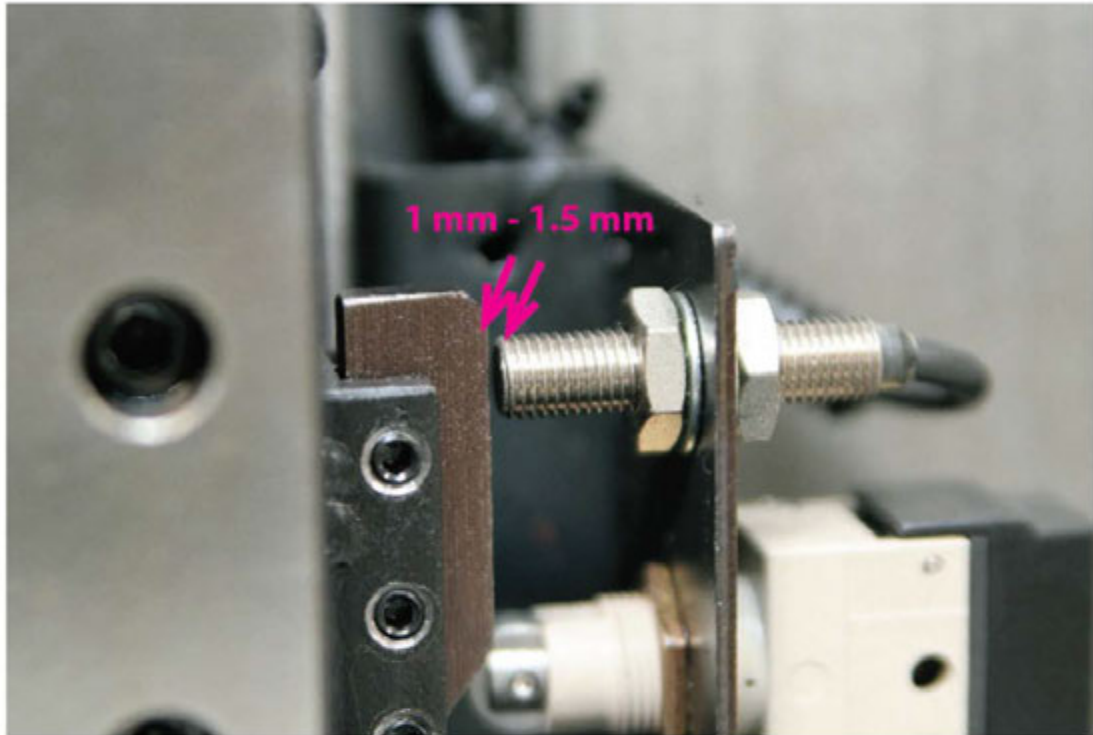
Note: Clean the suction filter every 3 months.

(10) Reference List of Lubricants and Hydraulic Oil

TYPE	MOBIL	SHELL
Gear ubricating Oil	MOBIL DTE LITE	SHELL TETRA OIL 32
	MOBIL DTE OIL HEAVYMEDIUM	SHELL TETRA OIL 68
	MOBIL DTE OIL EXTRA HEAVY	SHELL TELLUS OIL C150
	MOBIL GEAR 629	SHELL OMARA OIL 150
	MOBIL GEAR 632	SHELL OMARA OIL 320
	MOBIL GEAR 634	SHELL OMARA OIL 460
Bearing Lubricating Oil	MOBIL VELOCITE OIL No 3	SHELL TETRA OIL 2
	MOBIL VELOCITE OIL No 6	SHELL TETRA OIL 10
	MOBIL VELOCITE OIL No 10	SHELL TETRA OIL 22
Slideways Lubricating Oil	MOBIL VACTRA OIL NO 2	SHELL TONA OIL T68
	MOBIL VACTRA OIL NO 4	SHELL TONA OIL T220
Hydraulic Oil	MOBIL DTE OIL LITE	SHELL TETRA OIL 32
	MOBIL DTE OIL HEAVY MEDIUM	SHELL TETRA OIL 68
	MOBIL DTE 24	SHELL TETRA OIL 32
	MOBIL DTE 26	SHELL TETRA OIL 68
	MOBIL VACUOLINE OIL 1405	SHELL TETRA OIL 32
	MOBIL VACUOLINE OIL 1409	SHELL TETRA OIL 68
Grease	MOBILAX EP 0, 1	SHELL ALVANIA EP GREASE R0,1
	MOBILAX EP 2	SHELL ALVANIA EP GREASE R2

6. Proximity Switch Clearance

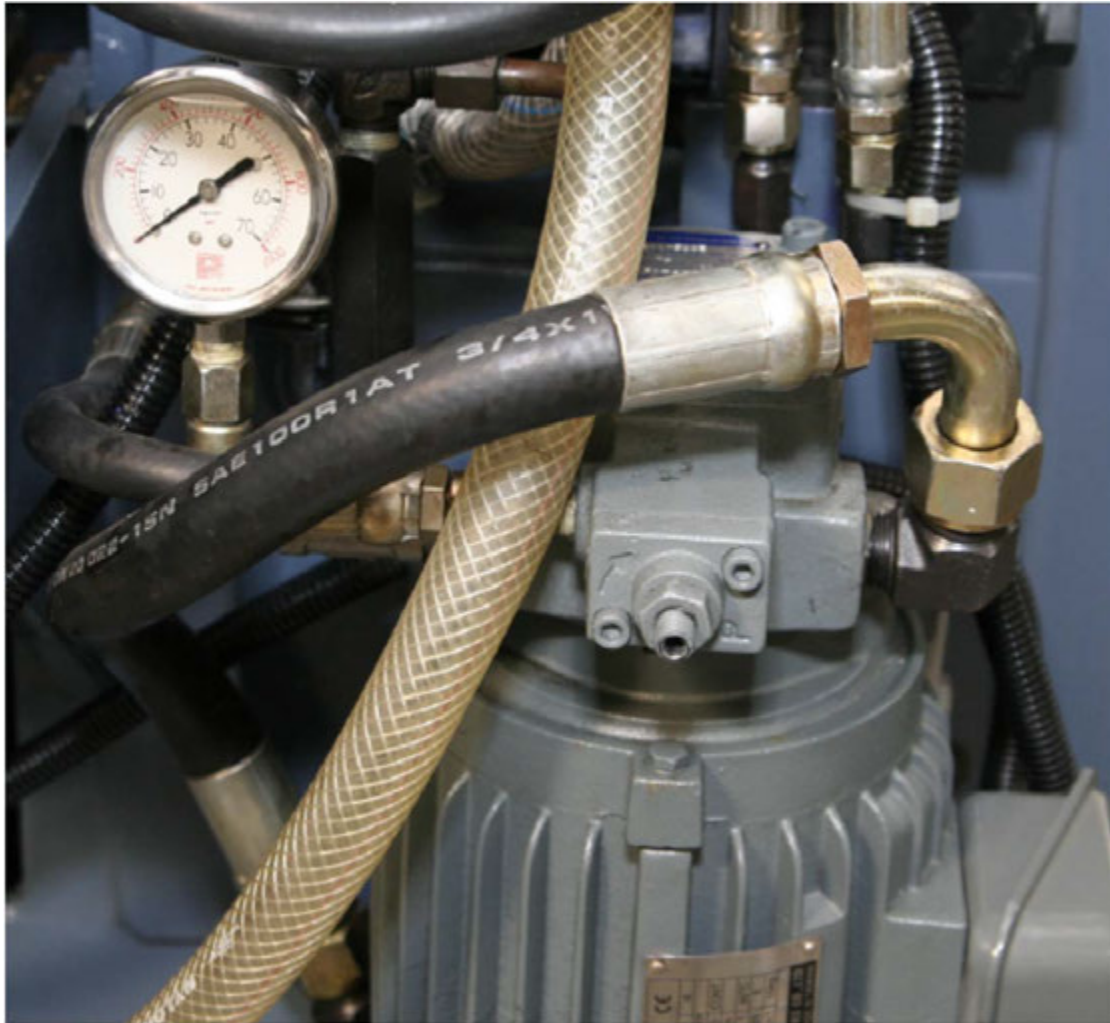
The clearance between the proximity switch and sensor block must be between 0.0393" (1mm) and 0.0590" (1.5mm). If out of this range, the machine may not be able to perform a reference search routing properly. Adjust the screw on the proximity switch if necessary to maintain correct clearance.



7. Hydraulic System

(1) Hydraulic Unit Specification

Item	Specification
Tank Capacity	3 gallon
Hydraulic Oil	MOBIL DTE 24
Oil Temperature	10° to 60° C




(2) Adjustment of clamping force

The work clamping force of the chuck can be adjusted by increasing or decreasing the delivery of hydraulic pressure with the pressure regulating valve on the hydraulic pump unit.

The hydraulic pressure supplied to the hydraulic cylinder can be checked with its pressure gauge.

NOTE: The hydraulic working pressure to be set for the chucking device varies depending on the adapted chuck and cylinder types. Therefore, refer to the chuck manufacturer’s manual for the allowable pressure ranges.

Consult with chuck manufacturer to find out the proper chucking pressure.

	<p style="text-align: center; background-color: yellow; font-weight: bold; margin: 0;">CAUTION</p> <p>Always use the chuck within the allowable pressure range. The clamping force of these chucks greatly varies depending on the position and the size of the attached jaws, cutting conditions, etc. The utmost care must be taken when deciding the proper chucking pressure. Otherwise, the chucking device may be damaged.</p>
---	---

(3) Switching Between ID/OD Chucking

The keep relay parameter K0.1 is used to switch between ID and OD chucking.

K0.1 = 0	OD Chucking
K0.1 = 1	ID Chucking

To change this parameter, do the following:

1. Press the SYSTEM function key
2. Press the PMC softkey
3. Press the PMCPRM softkey
4. Press the KEEPRL softkey
5. Cursor to parameter K00
6. Cursor to bit 1 (2nd bit from the right)
7. Input 0 or 1 value, then press the INPUT editing key

8. Coolant Equipment

The coolant equipment is intended to cool the chucked workpiece and cutting tools. The coolant also helps to flush chips away from the workpiece. Thus, the use of coolant affects the machining accuracy, machinability, and tool life.

The coolant system consists of a pump, tank, and piping with nozzles.

The coolant tank is integrated into the machine base and designed to accommodate a chip pan for easy disposal of cutting chips.

(4) Coolant Equipment Specification


Item	Specifications
Coolant Tank Capacity	27 gallons
Rated Pump Output	¼ HP

(5) Recommended Coolant

There are a variety of types of coolant available on the market. KEEN CHIEF does not specify any one type of coolant. Rather, choose a coolant suitable for your applications by consulting the supplier and taking the following requirements into consideration:

- The coolant must be free of adverse health effects to humans.
- The coolant must not deteriorate during storage.
- The coolant must not cause corrosion of the machine.
- The coolant must not peel the coating off the machine.
- The coolant must not cause swelling of rubber parts.
- The coolant must not cause deterioration of accuracy.
- DO NOT use low flash point cutting fluid.

NOTE: KEEN CHIEF can't be held responsible for any trouble arising from coolant use.


	WARNING
	<p>When using the cutting oil that may catch fire, be sure to equip the machine with an automatic fire extinguishing appliance.</p>

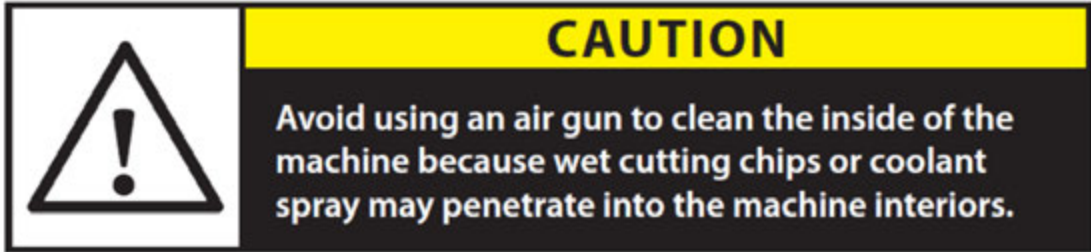
If the coolant viscosity is high, the thermal overload relay for the coolant pump will trip frequently and cause the machining errors. As the coolant viscosity is apt to change with the ambient temperature, select a coolant according to the following environmental temperature conditions:

- When the machine is used in an environment where the low atmosphere temperature reaches 20 C, select a coolant having an ISO viscosity grade VG16 or lower.
- When the machine is used in an environment where the low atmospheric temperature lowers reaches 10C, select a coolant having an ISO viscosity grade VG11 or lower.
- In the case of a high-viscosity coolant, the thermal overload relay may trip frequently when the coolant flow rate is reduced by the nozzle faucet. If you can't use a coolant with an ISO viscosity grade higher than noted above, consult your local KEEN CHIEF distributor.

Notes for coolant equipment:

- Check the amount of coolant in the tank through the level gauge daily. Refill the tank as necessary or when the coolant level reaches below 1/3 of the level gauge.
- Clean the coolant pump filter in the tank on a daily basis.
- Remove cutting chips accumulated in the chip pan on the coolant tank whenever necessary.
- Cutting chips may enter the coolant tank through the coolant recovery mesh. Therefore, flush the coolant every 3 months and clean the tank on the same occasion. Your cleaning cycle may depend on the type of coolant used and the material of the workpiece being machined.
- When cleaning the coolant tank, check the inside of the machine. If presence of cutting chips in the machine is evident, clean chips off the inside of the machine.

	WARNING
	Before changing the coolant and cleaning the coolant tank and/or after the inside of the machine, be sure to turn off the main disconnect switch to ensure safety.



9. Spindle Drive Belts

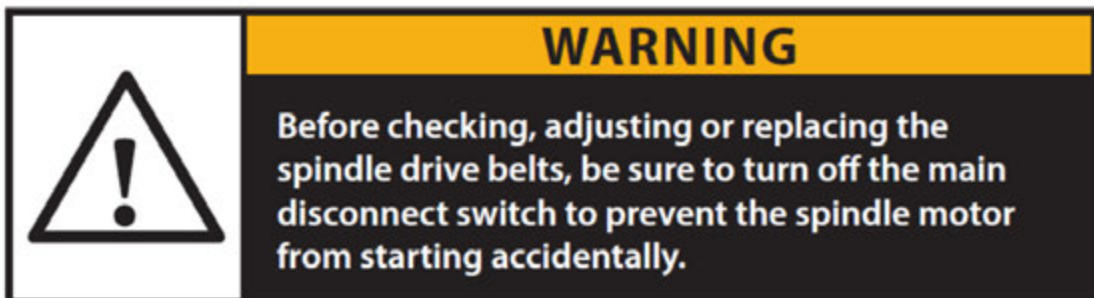
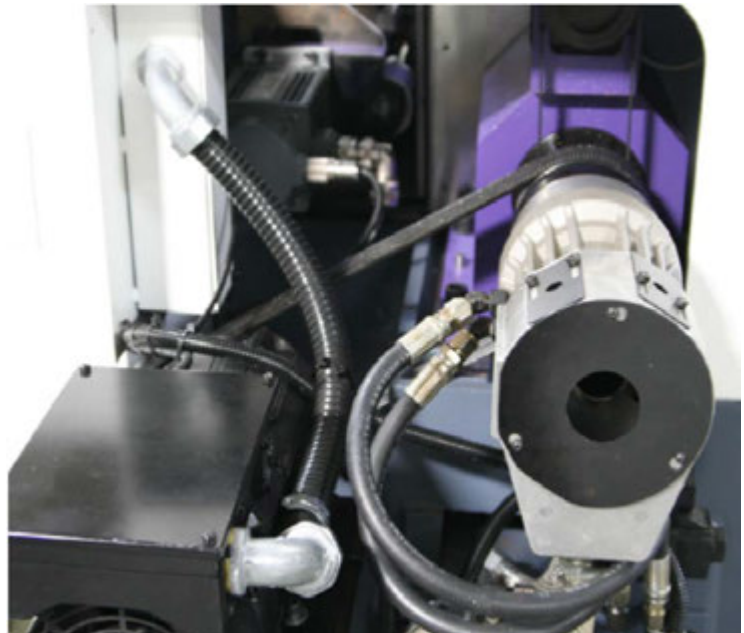
Always run the machine with the spindle drive belts properly stretched between the spindle and motor pulleys.

If the drive belts are stretched too tight, they will put excessive force on the bearings causing them to fail early. An overly tight belt also shortens belt life and reduces machining accuracy.

If these drive belts are too slack, they will slip on the pulleys and the required motor torque will not be transmitted effectively.

Moreover, the loose drive belts will generate heat making the belts less durable and producing unusual noise and vibration.

Check the tension of each drive belt at least once every 6 months and to readjust the tension or replace the belts as necessary.



10. Cooling Fan Filter

Clean the mesh filter on the heat exchanger of the electrical cabinet once a month or as necessary. To clean the filter:

1. Pull the tab on the bottom of the filter UP and then OUT to remove it from the fan cover.
2. Blow it clean with compressed air.
3. Reinstall onto the fan. Slide it up and then push down to fit it into the grooves.



11. Front Cover Window Glass

The front cover window glass is durable and won't break when hit by chips generated during machining.

There are instances (due to improper operation) when this glass has been broken by a workpiece, soft jaws, or a fixture. In such cases, the customer will be charged for replacing the broken glass.

Appendix A: Specification

1. Machine Specification

	GTV-27	GTV-42
X Axis Travel	11.8" (300mm)	
Z Axis Travel	7" (165mm)	
Spindle RPM	60-6000rpm	
Axes Rapid Travel (X,Z)	787"/ min (20m/min)	
Axes Maximum Feed Rate	400"/ min (10.15m/min)	
Axes Power Rating	42 in/lb (5.2 Nm)	
Positional Repeatability	0.00005" (0.00127mm)	
Positional Accuracy	0.00005" (0.00127mm)	
Spindle Runout	0.00005" (0.00127mm)	
Spindle Motor Power(continuous)	5 HP (3.7 kilowatt)	
Bar Capacity	1.062" (27mm)	1.654" (42mm)
Spindle Length with Collet Closer and Chuck	30" (762mm)	31" (787mm)
Collet Type	5C	16C,42CR
Chuck Size	5.3" (135mm)	6" (152mm)
Spindle Configuration	A2-4	A2-5
Chuck / Collet Clamping System	Hydraulic	
Simultaneous Axes	X-Z-C (3 Axes), X-C & Z-C Flat Plane Selection	
CNC Controllers	FANUC 0iMate-TC	
Tooling Center Height	1.653" (0.5"& 0.75" Opt.) [42mm (12.7mm & 19.0mm Opt)]	
Coolant Capacity	27 gallons (102 liters)	
Coolant Pump Power	½ hp (0.4 kilowatt)	
Chip Conveyor Type	Reversible Direction, CNC Controllable (Opt)	
Weight	3750 lb (1704 kg)	3800 lb (1724 kg)
Dimensions w/o options (L x D x H)	60" x 48" x 72" (1524mm x 1220mm x 1828mm)	

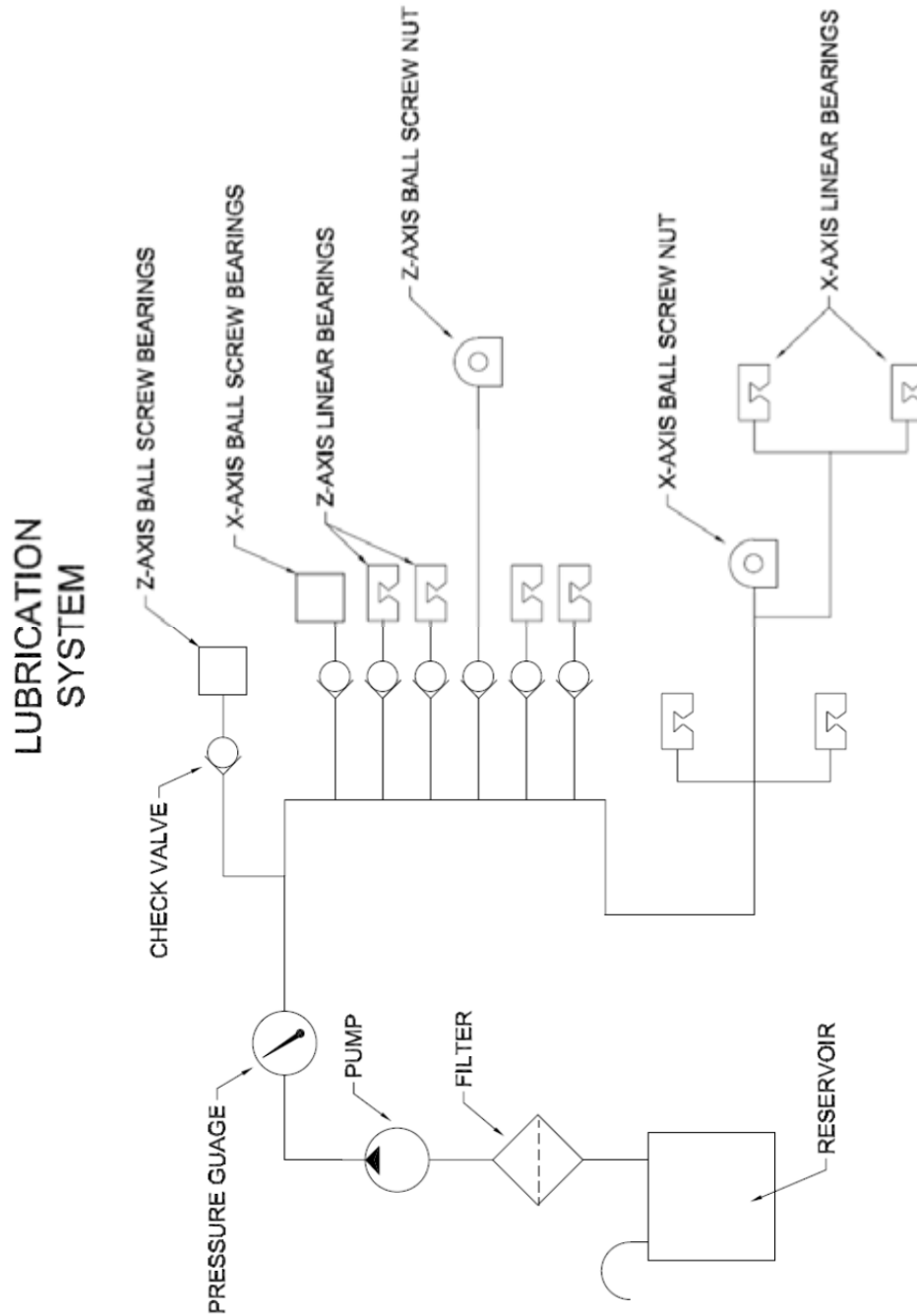
2. *Electrical Specification*

The electric power size required for MT is 12 KVA.

Voltage	AC220V, 3-phase (+10%, -15%)
Frequency	50/60 Hz
Momentary Power Failure	Less than 10 msec.
Voltage Impulse	Peak value is 200% or less of the effective value (rms value) of the line voltage with pulse duration of 1.5 msec.
Waveform Distortion	7% or less
Line Voltage Imbalance	5% or less

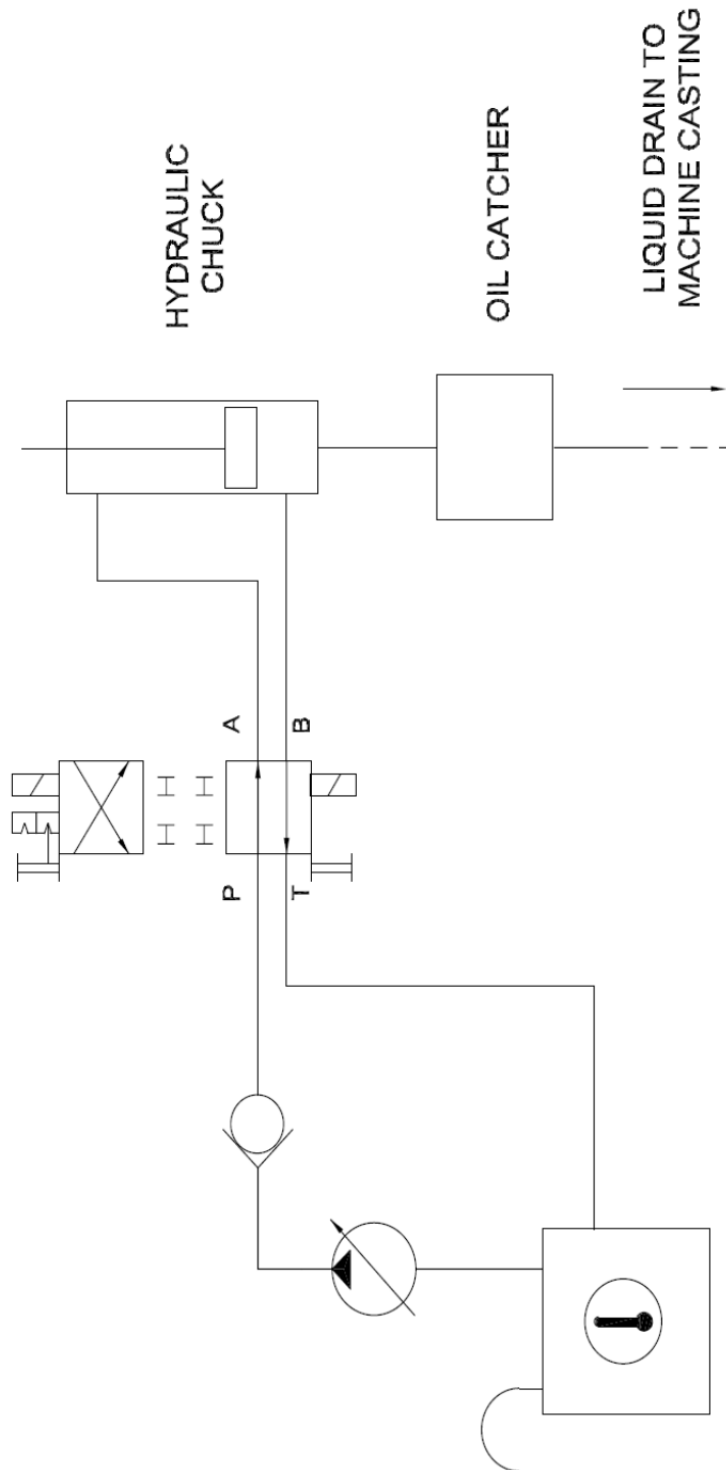
Appendix B: Schematics

1. Lubrication Schematic



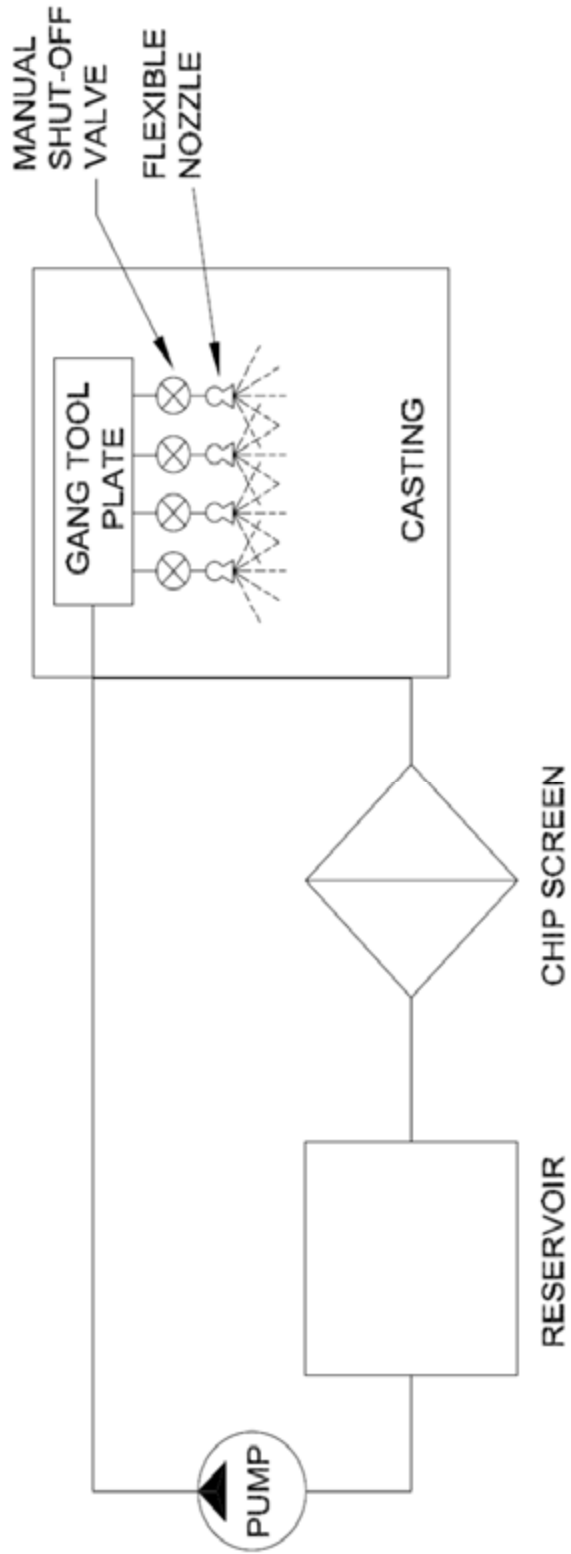
2. Hydraulic Schematic

HYDRAULIC SYSTEM



3. Coolant Schematic

COOLANT SYSTEM



Appendix C: Parts Lists

Hydraulic:

(KC20130814155517) Autostrong CR42 Collet chuck

(KC20140314113429) A2-5 16C Traub Collet

(KC20141230120519) Autostrong Hydraulic cylinder M1246(3/4") + CM12(1")

(KC2015098142218) ASHUN 9UC047613102401 Hydraulic Unit (1HP4P VP15 K101 25L container 3Φ 220V/440V 50 HZ)

Belt:

(KC20140311143158) BENDO 11M-1060 * 3

bearing:

(KC2013072141352) NSK Ball bearing 17TAC47BSUC10PN7B

linear guide:

(KC20130617152225) NSK Z axis linear guide RA250500ANC2V02K63

(KC20130617152322) NSK X axis linear guide RA250620ANC2B10K63

Ball Screw:

(KC20130617152518) NSK X axis Ball Screw W2004Z-1170PSS-C3-01

(KC20130617152429) NSK Z axis Ball Screw W2003Z-1417PSS-C5-01

Controllers and Servo motor:

(KC20130620121219) FANUC Controllers Oi-TF3(Bii6/12000)

(KC20130620122637) FANUC AMP Servo Amplifier

(KC20130620122001) FANUC Spindle Servo Motor

(KC20130620122854) X Y axis Servo Motor

Others:

(KC2013072110814) Heat exchanger DHE-25AF 1Ø 220V

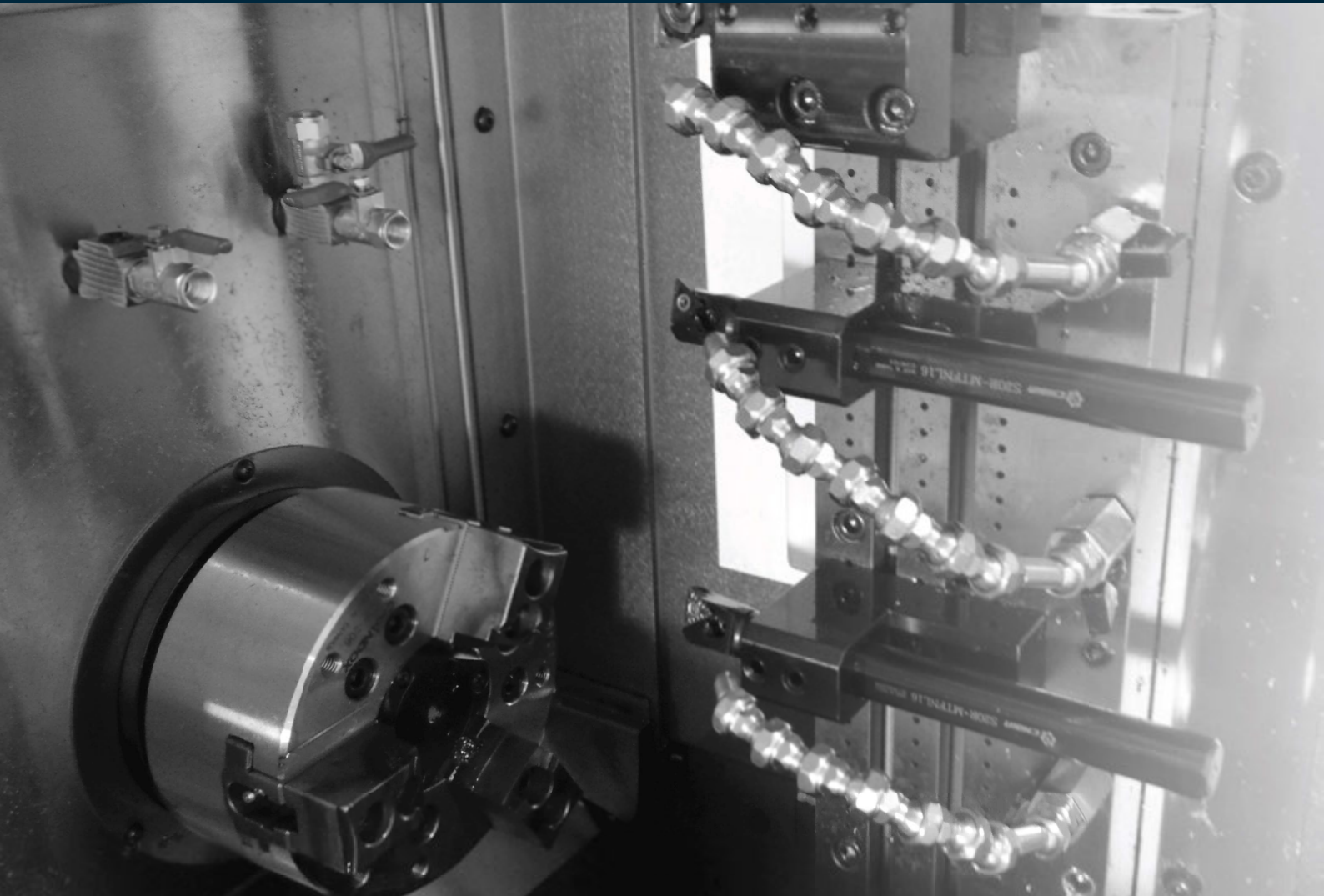
(KC20130620135244) Position coder A860-2109-T302

(KC20130620123400) REACTOR Voltage Dependent Resistor

(KC1507862955) 3Ø insulating transformer 6KVA I/P : 380V-440V(9.1A) O/P : 220V(15.7A)



www.keenchief.com



KEEN CHIEF
光鉛企業股份有限公司

TEL: +886-4-2312 9409 FAX: +886-4-2335 7071

Office: No.39, Henan E. 1st St., Situn Dist., Taichung City 407, Taiwan (R.O.C.)

Factory: No.43, Aly. 166, Chengfeng Ln., Taiming Rd., Wuri Dist., Taichung City 414, Taiwan (R.O.C.)

Email: info@keenchief.com