Automatic Tool Changer (ATC) for the VMC-5000

A Supplement to the VMC-5000 User's Guide

The information contained in this supplement (34-7208-0000) was accurate at the time of its printing. Light Machines Corporation reserves the right to change specifications and operational performance of its products without notice.

Printed in U.S.A.

Contents

Introduction

Welcome to the Automatic Tool Changer Supplement for the VMC-5000. This supplement is provided with machines equipped with the Automatic Tool Changer (ATC) Option. The purpose of this guide is to provide the user with the necessary information for the operation of the ATC. Also included in this guide is a "Quick Start" section. This section is a tutorial designed to familiarize the operator with the set up and operation of a multiple tool program using the Automatic Tool Changer. Please read through this manual before attempting to machine any multiple tool parts.

Warning

Improper setup or use of the Automatic Tool Changer can result in tool crashes which may cause damage to the Machining Center.

Installation

Hardware

Follow the general hardware installation procedures in your VMC-5000 User's Guide. Then perform the additional procedures outlined below.

Connect the ATC to an Air Supply

The Automatic Tool Changer requires a regulated shop air supply of 90 psi. Connect the air supply to the Filter Pressure Regulator Unit at the back of the Machining Center. Adjust the ATC air regulator to 80-85psi. If the pressure drops below 75psi., the controller will display an error message and the ATC will not work.

Install the Tools

Tools used with the ATC are inserted into the four tool holders supplied with the ATC. To insert the tools into the holders, follow these instructions.

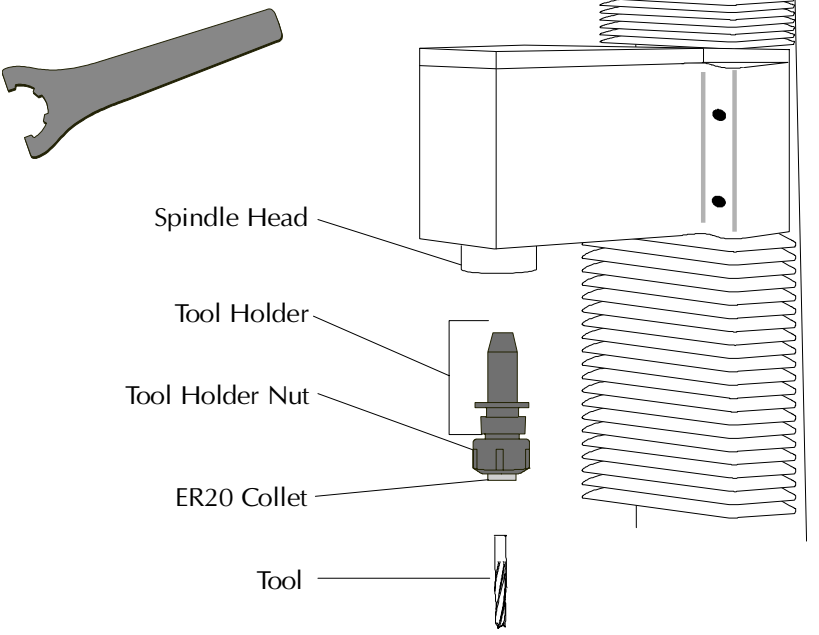
Note: The ER20 collet, ER20 wrench and collet chuck are included in the ATC Accessory Kit.

1. Insert an ER20 collet into the tool holder.
   a. Remove the nut from the tool holder.
   b. Insert the ER20 collet. It should snap in.
   c. Replace the nut on the tool holder, but do not tighten it.

Note: The tool holder nut and tool holder are balanced as an assembly. You should not exchange components between different tool holder assemblies. You may install any ER-20 collet in the tool holder assembly.

2. Unclamp the draw bar by pressing the "Tool Unclamp" button on the Pendant. The LED above the button lights, indicating the draw bar is unclamped.

Use the wrench supplied in the ATC Accessory Kit to tighten the Tool Holder Nut.



3. Insert the tool holder into the VMC-5000 spindle head and clamp the draw bar by pressing the "Tool Unclamp" button again. The LED will turns off, indicating the Draw bar is now clamped.
4. Insert the spindle locking pin into the side of the spindle head.
5. Insert a tool into the tool holder.
6. Tighten the tool holder nut using the supplied wrench to secure the tool.

CAUTION:

Do not overtighten the nut. Maximum torque should be no more than 35 ft.-lbs.

7. Remove the spindle locking pin, and remove the tool holder from the spindle by pressing the Tool Unclamp button on the pendant.
8. Hang the tool holder on a tool station of the ATC. Press the tool holder, with the tool end hanging down, into the tool station until you feel a slight snap as the holder passes the detent on the station.

Important!

The tools are assigned numbers one through four starting at the front. The tool you mount closest to the front of the Machining Center is regarded as tool #1. The tool you mount furthest from the front of the Machining Center is regarded as tool #4.

When the Machine is Not in Use

Always leave the spindle empty when the Machining Center is not in use. Before shutting down the machining center, enter the MDI mode and run the M30 command.

The Pneumatic Draw Bar

The pneumatic draw bar reverts to a normally closed state if air pressure is lost. If you lose shop air while a tool holder is inserted in the draw bar, you will not be able to remove the holder. If you lose shop air with no tool inserted, you will not be able to insert a tool until air is restored. Refer to the section on *Recovering from Interrupted Tool Changes* in this supplement for more information.

System Sensors

Low Air Pressure Sensor

This sensor monitors the pressure of the shop air supply to the ATC. If the air pressure drops below 75 psi, an Air Pressure Failure message is displayed on the screen. The machine is taken out of cycle and machine power is dropped when a failure is detected.

Draw Bar Position Sensor

This sensor is mounted in the Draw Bar Assembly on the Spindle Head, and monitors the presence or absence of a tool holder in the spindle.

Tool In Position Sensor

This sensor is mounted in the bottom of the spindle head, and monitors the height of the tool holders. A small post is mounted on each tool station on the ATC. When the tool station inserts a tool holder into the spindle, the small post is seen by the sensor. If the sensor does not see the post, the tool change will be aborted.

To correct this situation, you must reinitialize the ATC. Make sure that the top of the tool holders are centered and just touching the plug during initialization.

Mechanism Align

In order for the VMC-5000 to operate properly with the ATC, it must complete a Mechanism Alignment. This is a series of position and sensor checks to determine the location of the ATC in relation to the spindle, whether there is a tool in the spindle, and the orientation of the tool holders on the ATC (are they up or down?). This Mechanism Alignment, or Mech Align, must be performed after the initial power up of the machine, and any time the Emergency Stop button is pressed during machining operations. See page 22 for information on how to perform the Mechanism Alignment.

Programming Tool Changes on the ATC

Programming Tool Changes

Here is an example of tool changing code using the 4-digit method:

M06T02; Change to tool in Station #2

M06 is the Tool change command, T02 is the Tool ID number, or tool reference number.

Tool changes are easily entered in the Programming mode. Select the Programming Mode button on the touch screen, and enter the changes using the keyboard.

The MDI mode may also be used to program tool changes, but it is not recommended that you use the MDI mode for anything other than short programs or setup activities. The Programming mode offers much more flexibility.

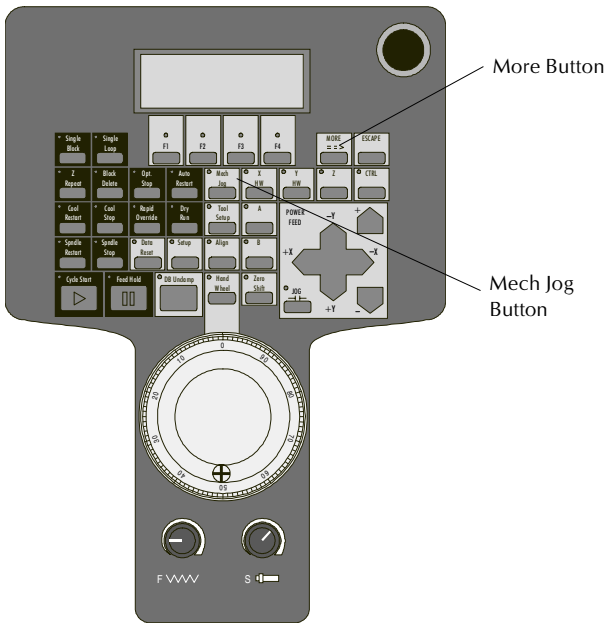
Programming for More Than Four Tools

If you are using more than 4 tools for a particular program, the additional tools must be set up as Manual Load tools. During the procedure to establish tool offsets and define the additional tools (see Quick Start section, page 32), under the Tool Load Type field, select "Manual Load." When the program requires the additional tools, the machine stops. A message for an operator requested tool change is displayed.

Recovering from an Emergency Stop

Any time an Emergency Stop is executed, the Mechanism must be re-aligned. If the Emergency Stop is pressed when the ATC is in the middle of a tool change and the tool is not secure in the spindle, then the tool holder must be released and the ATC must be jogged away from the spindle head.

1. Use the Unclamp Tool button on the pendant to release the tool holder. If there is a tool holder in the spindle, be prepared so the tool doesn't fall onto the cross slide.
2. Using the Mech Jog button on the pendant, lower the tool station.
   a. Press the Mech Jog button on the pendant. The LED above the button lights and the LCD display changes.
   b. Press the More button to the right of the display until "Mech Jog 5" through "Mech Jog 8" are shown.



c. To raise or lower the appropriate tool station, press the corresponding F-Key. The F1 key is under Mech Jog 5, and raises and lower tool station 1. F2 / Mech Jog 6 is used for tool station 2, and so on.

d. When the tool stations are in the down position exit the Mech Jog mode by pressing the Mech Jog button again. The light goes off.

3. Make sure there is no tool in the spindle. Place the tool manually into the tool station.
4. Slowly jog the spindle head away from the tool station, then press Data Reset.
5. Pressing the Unclamp Tool button on the pendant again clamps the draw bar.
6. The Mechanism must now be realigned.
7. Locate and press the Align key on the Pendant. When the Align option is activated, the LCD displays choices above the F keys. The message Mech--Align is above the F2 key.
8. Press the F2 key. The machine automatically makes a series of position and sensor checks. During these checks a message appears on the controller screen indicating the mechanism is being aligned. Once the checks are complete, the message "Mechanism Aligned" appears on the screen.
9. Press the Escape key on the pendant to exit the alignment mode.

Tool Parameters Used on the A2100 Control

The A2100 Control contains a large number of tool parameters. This is a brief explanation of each tool parameter and its use.

Accum Count: The Accumulated Count indicates the number of times the tool has been used. When the Accumulated Count reaches the Limit Count, and the Count Monitor is enabled, the tool is no longer valid for use.

Accumulated Time: The Accumulated Time is the length of time a tool has been in service. When the Accumulated Time reaches the Limit Time, the tool is no longer valid for use. The tool IS NOT automatically unloaded and an alternate tool loaded. The tool is not allowed to be reloaded into the spindle after removal.

Alternate ID: The A2100 Control has the ability to monitor tool usage and categorize tools in several different ways. When a tool change is encountered, the system checks to ensure that the specified tool is still usable. In the event that a worn or broken tool is specified, an alternate tool may be substituted. The alternate ID specifies what the substitute tool ID number is.

Count Monitor: The Count Monitor determines whether or not tool usage is updated based upon the number of times a tool has been used. This may be set to either Enabled or Disabled, and is used in conjunction with Limit Count and Accumulated Count.

Dia Offset: The Diameter Offset is the amount of deviation from the actual cutter diameter to the programmed cutter diameter. This parameter is very important when using tool diameter cutter compensation. For example, a 0.125" diameter end mill may be specified for a particular setup. When loading the tools, the diameter of the end mill may actually measure out to 0.1248". The deviation in diameter is the Diameter Offset.

Flute Length: The length of the flute on the cutter.

Feed Ovrd: Feed rate Override limits the maximum percentage of feed rate override on a per tool basis that can be set by the pendant.

Length: In multiple tool programming, the Tool Length is the Tool Length Offset. Tool Length can be set relative to a reference tool, relative to a touch probe, or relative to the machine home position. It is important that the Tool Length be set properly to ensure that tool changes are performed correctly.

Limit Count: This integer sets the maximum number of times that a tool may be used when the Count Monitor is enabled.

Limit Time: This number specifies how long a tool may remain in service.

Load Method: This option specifies how a tool is to be loaded into the machine. Options include:

- Manual Load
- Auto Load

The VMC-5000 Machining Center without ATC supports only Manual load tools. In this case, the spindle motor is shut off, the machine is taken out of cycle and the operator is requested to load the next tool. After installing the tool, the operator must press the cycle start button on the pendant to restart the machining process.

Both Auto Load and Manual Load are available on the VMC-5000 with ATC. An Auto Load tool is automatically loaded from and unloaded to the ATC.

Material: Defines what type of material the tool should be manufactured from. Options include:

- Unknown
- High Speed Steel
- Tin Coated HS Steel
- Carbide Insert
- Carbide Coated
- Carbide Solid

Max Feed/Tooth: Controls the maximum feed rate based on spindle speed and number of teeth on the cutter.

Max Spn RPM: Maximum spindle speed the cutter can run.

Nom Diameter: Nominal Diameter is used by the operator during machine setup. This is the nominal diameter for the selected tool.

Pocket: Used to identify where on a tool changer a tool is loaded. The VMC-5000 with the ATC recognizes pocket numbers 1-4 as valid entries.

Record Number: The Record Number is used by the system to keep track of tools. This field can not be changed by the user. If a tool change is commanded and a valid tool ID is not recognized, then the system will use the record number in place of the tool number.

Serial Number: This is a user defined reference number. The control does not use this number. It is supplied to assist in machine setup. Frequently it is linked to some type of tool suite.

Size: The Size parameter is used to define how many tool pockets must be vacant adjacent to an Auto Load tool. For example, a shell mill may have a fairly large diameter. Depending upon the configuration of the tool handling equipment on the machine, tool pockets next to the shell mill may not be able to hold tools because of the size of the shell mill. The size parameter defines how many pockets both before and after the shell mill must remain vacant to avoid interference.

Spindle Direction: Determines what spindle rotation directions are permissible for this tool. Options include:

- No Rotation
- CW Rotation
- CCW Rotation
- Either Direction

If the NC program attempts to rotate a cutter in a direction for which it is not authorized, an alarm will be posted and program execution terminated.

Spindle Ovrd: Spindle Override limits the maximum percentage of spindle override on a per tool basis that can be set by the pendant.

 # Teeth: Defines the number of teeth on the cutter. The A2100 Control has the ability to limit feed rates based on advance per tooth. If this feature is used, the number of teeth must be set properly to ensure that feed rates are properly limited.

Thread Lead: Lead (pitch) on a tap. Used for tapping applications.

Time Monitor: Monitors tool usage based on the amount of time the tool has been in service. Settings are either Enabled or Disabled. Time Monitor is used in conjunction with Limit Time and Accumulated Time.

Tool ID: This is the tool number used in an NC program. For example, when programming M06T01, T01 references the Tool ID. If a valid Tool ID is not found in the tool table, the system will check for a valid tool reference number and use that tool.

Tool Status: Monitors whether a tool is New, Good, Worn, or Broken based on usage. The A2100 supports automatic tool monitoring. In the event that a tool is worn or broken, the system has the ability to select an alternate in place of the bad tool.

Tip Angle: Defines the tip angle for the tool. For most applications, Tip Angle is not necessary. However, for certain canned drilling cycles, the tip angle must be properly specified to ensure that holes are drilled to the correct depth, taking the tool tip into account.

Type: Identifies what type of tool the operator should use. Some of the options available are:

- Unknown
- Rough End Mill
- Finish End Mill
- Ball End Mill
- Face Mill
- Shell Mill

Quick Start

This section is intended as a tutorial for starting and operating the VMC-5000 Machining Center with the Automatic Tool Changer and the Acramatic 2100 Control. The emphasis of this exercise is to learn how to set up and machine a multiple tool program on the VMC-5000 Machining Center. It is strongly suggested that the project MILLONE.NC from the VMC-5000 User's Guide be completed before attempting MILLTWO.NC. Basic machine operations are covered in the MILLONE.NC program. This project more involved, but is designed so that an inexperienced operator will be able to set up and run this program, provided that all steps are carefully followed.

For this exercise, the Automatic Tool Changer will be used to run the multiple tool program. Other items required:

- 1/8" End Mill
- 1/4" End Mill
- 3" x 2" x 1.5" Machinable Wax
- Vise or Low Profile Clamping Kit
- 3.5" Floppy Disk with MILLTWO.NC

Start Up

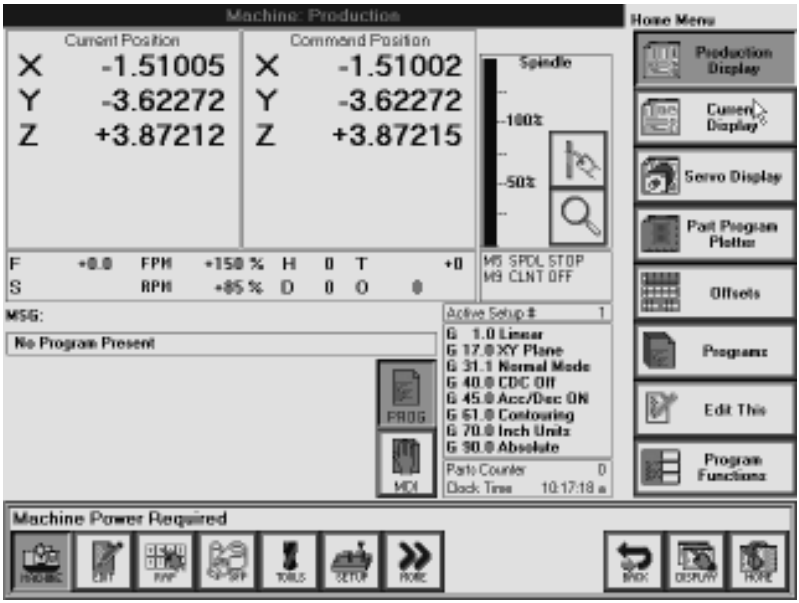
Several steps are involved in the start up procedure of the equipment. These steps must be followed each time the machine is switched on.

1). Turn on the Acramatic 2100 Controller by pressing and releasing the green power switch located on the left hand side of the Control Panel, just above the Emergency Stop button.

The start up procedure takes several minutes to complete. The system goes through a series of internal checks and loads various software applications.

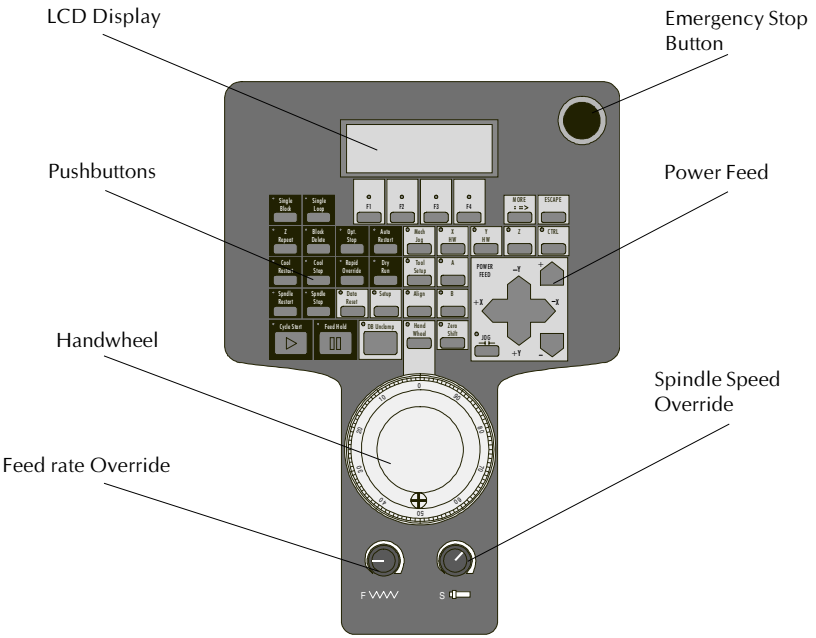
2). Once the software is running, Press the Home button located at the lower right hand corner of the touch screen. This will bring the software back to the Home screen.

Note: As you navigate through the software, you may find that you do not know how to get back to where you started. Simply press the Home button and you will automatically be returned to the starting location.



After the controller is running, you will be able to power up the Machining Center. As a safety precaution, the machining center is not automatically powered up when the controller is switched on, and needs to be powered up separately.

1. Pull out the Emergency Stop button on the Pendant. The Emergency Stop button is the red button located at the top right hand corner of the pendant.
2. Pull out the Emergency Stop button on the Controller. The Emergency Stop button on the controller must be deactivated by pushing the Emergency Stop button inward, rotating counterclockwise, and then releasing. The Emergency Stop button will then be released.
3. Press in and hold the green power switch for two seconds to power up the machining center. The machining center is now powered up and ready for use.



Align the Machining Center

Once the Machine is powered up, the "Machine Unaligned" message is displayed on the Controller. You need to align the machine before you can use it. Aligning the machine establishes a point of origin at the ends of travel on the X, Y and Z axes. The machining center uses this point as a reference for all machine coordinate movements.

1. Locate and Press the "Align" key on the pendant. When the align option is activated, the LCD display on the pendant will now read "Axes - Align" above the F1 key.
2. Press and hold the F1 key. The machine will automatically align itself to the +X, +Y and +Z axes limit switches. During the alignment sequence a message is displayed on the controller indicating that the machine is performing the alignment cycle. Once the machine has completed the cycle, the message "Axes are Aligned" is displayed on the screen, above the menu buttons on the bottom left corner. This message indicates that the machine has successfully aligned itself and is now ready for use.

3. Press the Escape key on the pendant to bring the pendant out of the alignment mode. The LED on the ALIGN key stays lit, indicating that the machine is aligned.

Mechanism Align

In order for the VMC-5000 to operate properly with the ATC, it must complete a Mechanism Alignment. This is a series of position and sensor checks to determine the location of the ATC in relation to the spindle, whether there is a tool in the spindle, and the orientation of the tool holders on the ATC (are they up or down?). This Mechanism Alignment, or Mech Align, must be performed after the initial power up of the machine, and any time the Emergency Stop button is pressed during machining operations.

To perform the Mech Align:

1. Locate and press the Align key on the Pendant. When the Align option is activated, the LCD displays choices above the F keys. The message Mech--Align appears above the F2 key.
2. Press the F2 key. The machine automatically makes a series of position and sensor checks. During these checks a message appears on the controller screen indicating the mechanism is being aligned. Once the checks are complete, the message Mechanism Aligned appears on the screen.
3. Press the Escape key on the pendant to exit the alignment mode.

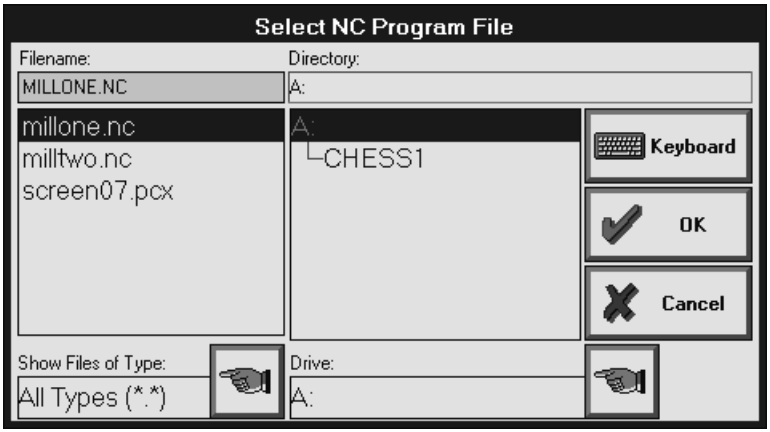
Load the NC Program

The Acramatic 2100 Control keeps track of an active NC file set for production applications. Additional storage space is available on the hard drive for storing NC files. Use the following procedure to load a new NC file onto the hard drive and to transfer it into the active NC file area.

1. Insert the disk containing the desired NC program into the floppy drive.
2. Press the “Programs” button on the Home Menu of the Controller. This button is located on the right hand side of the screen. The NC Part Program dialog box appears.

3. Select “Transfer Program” to transfer a new NC file into the active NC program area. The Transfer Program sub-menu appears.
4. Select “Transfer Program In.” The Transfer To Program Store dialog box appears.
5. Select the “Floppy Drive” button.
6. Select the “Browse” button to view the contents of the floppy drive. The Select NC Program File dialog box appears listing the available NC programs on the floppy drive.

7. Select the Hand Icon to narrow down the number of files listed. Select the Hand Icon adjacent to the “Show Files of Type” dialog box. A pop out listing of the most common file types is displayed.
8. Select EIA274 (*.NC) from the file type pop out menu. The filename window now only shows files with the NC extension.
9. Select MILLTWO.NC from the file listing, then select OK to return to the Transfer to Program Store dialog box.
10. Select START TRANSFER to transfer MILLTWO.NC to the active program store area.



11. MILLTWO.NC is now registered in the Active NC Store Area and is shown on the NC Programs listing. If it is not currently selected, choose MILLTWO.NC from the list and then select RUN THIS PROGRAM. This will load the program into the memory buffer, but won't start the program.
12. MILLTWO.NC is now the active NC program for the machining center. The codes for MILLTWO.NC appear in the program window on the left hand side of the screen.

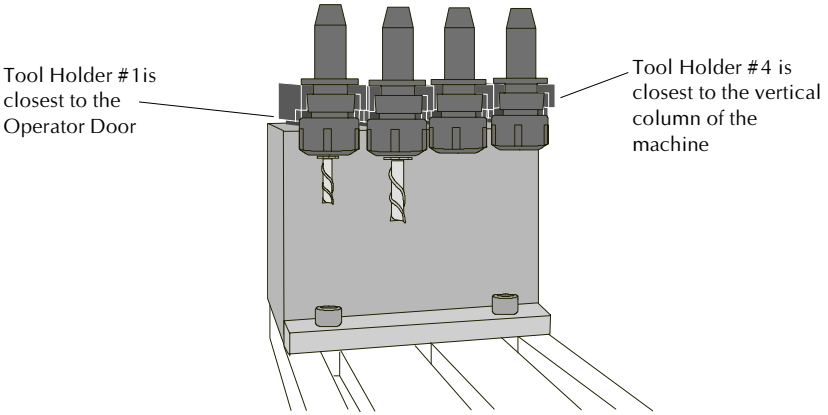
Mount the Workpiece

A vise and workpiece need to be mounted on the cross slide at this time. Use the Power Feed keypad on the pendant to jog the machine to a location convenient for mounting the vise and workpiece on the Cross Slide. To move an axis, select the appropriate axis direction key from the Power Feed keypad. Press and hold the key to move the axis. Release the key to halt motion.

1. Using the Power Feed keypad, jog the cross slide to a spot convenient for mounting the vise.
2. Press in the Emergency Stop button on the Controller, and open the safety shield.
3. Mount the vise and workpiece. The MILLTWO.NC program uses a 3" x 2" x 1.5" piece of machinable wax.
4. Close the safety shield and pull out the Emergency Stop button on the Controller.
5. Any time the Emergency Stop button is pressed on the machining center, power is completely shut off to the machine. The machine will have to be manually started by pressing in and holding the green power button on the controller for two seconds. This will start the machining center.

Tool Set Up

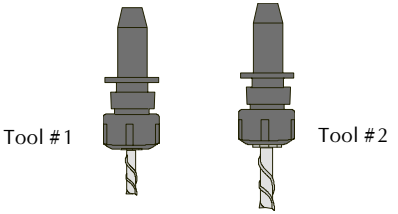
1. Install the 1/8" End Mill into the tool holder for Tool #1. See page 6 of this manual for the tool installation procedure.
2. Install the 1/4" End Mill into the tool holder for Tool #2.
3. Install the two tool holders onto the tool changer. Tool #1 should be installed in tool fork #1, and Tool #2 should be installed in tool fork #2. Tool fork #1 is the tool fork closest to the operator door on the machine. Tool fork #4 is closest to the vertical column on the machine.



Establish Tool Length Offsets

For this project, Tool #1 will be used as a reference tool. Tool #2 will be either longer or shorter than Tool #1. This difference in tool lengths is known as a Tool Length Offset. When the machine runs any NC code that uses multiple tools, it will have to offset the Z axis by the tool length offset to ensure that all cuts are made to the correct depth.

For first time multiple tool users, this method of specifying tool length offsets is easy to understand. After becoming proficient in setting tool length offsets, you may wish to modify this procedure. The following figure should help visualize tool length offsets.



As you can see from the diagram, tool #1 is shorter than tool #2 (your setup may vary).

When the NC program is run, the spindle head will have to be offset by the difference in tool lengths to ensure that the tools cut to the proper depth relative to the stock. In the case of the tools depicted in the diagram, the spindle head will have to be raised 0.25" higher (relative to the stock) when Tool #2 is loaded than when Tool #1 is loaded. This is the Tool Length Offset.

For the MILLTWO.NC program we will be specifying tool length offsets measured relative to Tool #1. To measure the offset, we will use the Tool Length Offset Sensor.

The electronic Tool Length Offset Sensor is supplied with most Light Machines' machining centers. The sensor contains a battery which uses the continuity of the machine to complete a circuit. When the sensor is placed on the cross slide, the tool is jogged down so it just touches the top of the sensor. When it touches, the circuit is complete, and the three LED's light. The sensor will NOT function if it is placed on a non conductive surface such as a block of machinable wax.

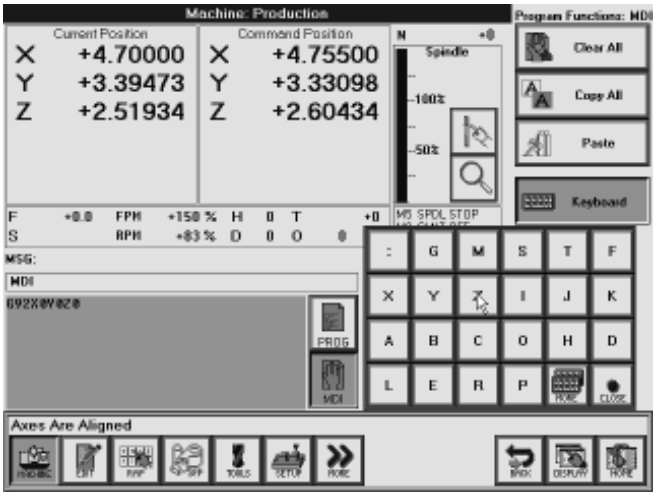
Establish Reference Plane

In this exercise, the top of the sensor will be used as a reference point from which tool lengths will be measured. Once the reference point has been established, all other tools will be loaded into the spindle and brought to sensor allowing the tool length offset to be measured.

1. Set all tool lengths to Zero.
   a. Select the Tools button
   b. Set all lengths to 0.
   c. Return to Home Screen

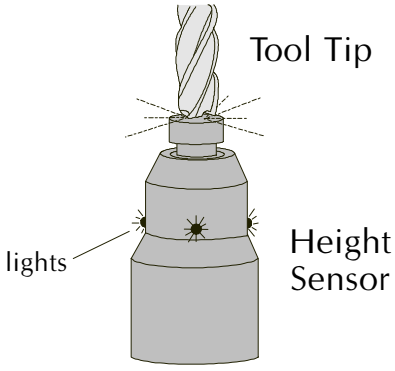
2. Install Tool #1 in the spindle. To do this, a short program must be written and executed on the controller.
3. Press the MDI button. The MDI button is located to the right of the NC Code listing window, above the menu buttons on the bottom of the screen. The NC code will disappear from the window when in MDI mode.
4. Select the Program Functions button from the Home Menu. The Program Functions Menu allows us to open an onscreen keyboard. The keyboard will be used along with the numeric keypad on the control to create a one-line NC program. The Program Functions MDI Menu appears.
5. Open the G+M code keyboard by selecting Keyboard. A small G+M Code specific keyboard appears. This keyboard has all of the letters used in common NC programming. (A QWERTY keyboard is available by pressing the More button on the keyboard.)
6. Using a combination of the keyboard and the numeric keypad, type M06T01. The NC code will appear in the NC code window as you type.
7. Close the keyboard by pressing the Close button on the bottom right corner of the keyboard.

8). Press the Cycle Start button on the pendant to run the program. The machine will automatically install Tool 1 into the spindle.



9. Place the Tool Length Offset Sensor on the machine cross slide.
10. Using the Power Feed keypad on the pendant, jog the tool close to the sensor. Do not attempt to use the Power Feed keypad to get the tool to actually touch the sensor. The Power Feed does not allow for fine positioning of the tool and should only be used for gross adjustments of tool position. A handwheel is provided to allow for precise positioning of the tool.
11. Once the tool is close to the sensor, use the Jog Handwheel to jog the tool tip so it just touches the top of the sensor. To use the handwheel, press the Handwheel button on the pendant. Select the Z axis motion key from the motion keys above the Power Feed keys to set the Z axis as the current axis for motion.
12. The LCD display on the pendant now displays “Handwheel Multiplier Select” options (X1 X10 X50 X100) selected by the function keys (F1 through F4 respectively).

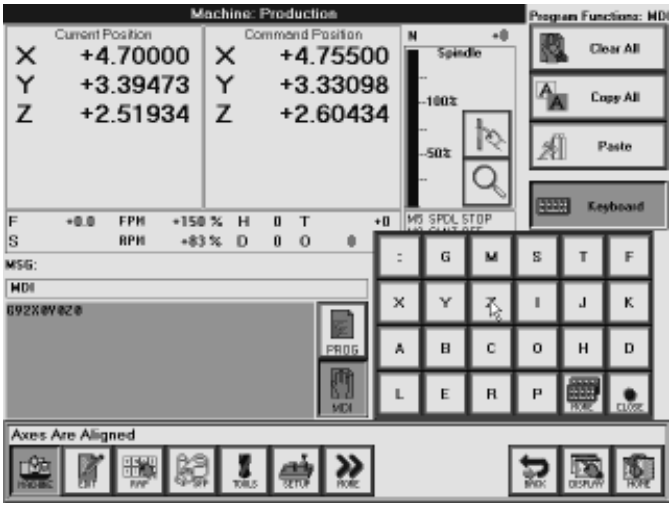
    The Handwheel Multiplier options determine how far the axis moves for each increment on the handwheel. With X100 selected, the axis will move .010" each time the handwheel is incremented. With X50 selected, the axis will move 0.005", X10 will move the axis 0.001" and X1 will move the axis 0.0001. Select a Handwheel Multiplier by pressing the corresponding F Key. For now select X50 by pressing the F3 key.
13. Rotate the handwheel to jog the Z axis. Each increment on the handwheel will cause the Z axis to move 0.005". Rotating the handwheel counterclockwise will cause the axis to move downward, and clockwise rotation will move the axis up.
14. Jog the tip of the tool so it just touches the tool height offset sensor. The LED’s on the sensor light, indicating contact has been made.



Set Z Axis Coordinate

Once Tool #1 has been positioned at the sensor, you must set the machine's Z axis coordinate to 0.0. This establishes the reference plane (the top of the sensor) from which all tool length offsets are measured. In order to set the Z axis position to 0.0, a short NC program will need to be written and run on the machine. Follow these steps to set the Z axis coordinate to 0.0.

1. Leaving the tool on the sensor, press the HOME button to return the control to the home screen.
2. Press the MDI button. The MDI button is located to the right of the NC Code listing window, above the menu buttons on the bottom of the screen. The NC code will disappear from the window when in MDI mode.
3. Select the Program Functions button from the Home Menu. The program functions menu allows us to open an on screen keyboard. The keyboard will be used along with the numeric keypad on the control to create a one line NC program. The Program Functions MDI menu appears.



4. Open the G+M Code keyboard by selecting Keyboard. A small G+M Code specific keyboard appears. This keyboard has all of the letters used in common NC programming. (A QWERTY keyboard is available by pressing the More button on the keyboard.)

5. Using a combination of the keyboard and the numeric keypad, type G92Z0.0. The NC code will appear in the NC code window as you type.
6. Close the keyboard by pressing the Close button on the bottom right corner of the keyboard.
7. Press the Cycle Start button on the pendant to run the program. The machine's position will be updated on the display. Note that the Z axis position now reads 0.0000.

Establish Offset for Tool #2

Tool #2 will now be installed in the spindle and jogged to the top of the tool height offset sensor. Because Tool #2 is a different length than Tool #1, the spindle head will be at a different location when Tool #2 is in contact with the sensor. This difference is spindle head location is the tool length offset. The tool length offset can then be taken directly off of the control.

1. Jog Tool #1 away from the workpiece and install Tool #2 (see procedure on page 28).
2. Jog Tool #2 to the sensor following the same procedure as Tool #1. Tool #2 should just touch the top of the sensor and the LED's on the sensor light.

Note that the Z axis coordinate on the control unit now displays a different value than the value for Tool #1. If Tool #2 is longer than Tool #1, the Z axis position is a positive value. If Tool #2 is shorter than Tool #1, the Z axis position is a negative value. This value is the Tool Length Offset for Tool #2. Write this number on a sheet of paper.

Define the Tool

In order to record the Tool Length Offsets, we must set some of the tool parameters. For a brief summary of all the tool parameters, see the Tool Parameters Section of this guide.

Select TOOLS from the General Mode Button Bar at the bottom of the screen. The tool setup dialog box is displayed on the screen. There are four parameters of interest for this tutorial, Record Number, Tool ID, Load Method and Length. Note that your display may appear slightly different from the screens pictured in this manual.

Record Number: The Record Number Field is an internal register maintained by the system to keep track of tools. It cannot be changed by the user.

Tool ID: The Tool ID number is the number that defines the tool in an NC program. When a tool change is commanded, the Tool ID number is the number specified by the "T" code. For example, M06T3 specifies a tool change to Tool ID #3.

Load Method: This option specifies how a tool is to be loaded into the machine. Options include:

- Manual
- Auto Load

Both Auto Load and Manual Load are available on the VMC-5000 with the ATC. When Auto Load is selected, tools are automatically loaded from and unloaded to the ATC. If a tool is loaded into the ATC, then Auto Load is selected as the load type.

Length: The length is the Tool Length Offset. In the setup for this tutorial, Tool ID #1 has a length of zero. (Tool #1 is the reference tool from which other tools are measured) Tool #2 has a length that was determined during the steps above.

Define Tool #1

We need to create a tool definition for Tool #1. Tool #1 is the 1/8" end mill. Because it is also our reference tool, the Tool Length for Tool #1 is set to zero. The tool ID should be set to 01. Enter these parameters in the Tool Data table. Set the Load Method to Auto Load.

Define Tool #2

Create a tool definition for Tool #2. Tool #2 is the 1/4" end mill. The value found earlier in this exercise (Tool Length Offset) is the Tool Length. The Tool ID for Tool #2 is 02. Enter these parameters in the Tool Data Table. Set the Load Method to Auto Load.

Additional Tools

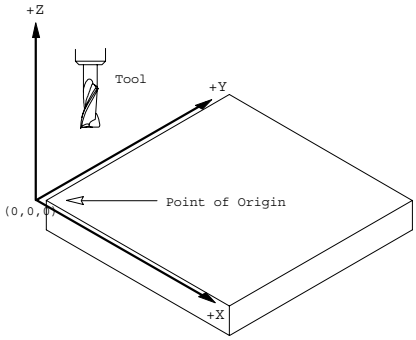
If this program used additional tools, you would repeat the above procedure to determine the offsets for the additional tools. Keep in mind that the tool offset would be relative to Tool #1.

Data Reset

Press Data Reset to save the new tooling information to the Control.

Set Workpiece Origin

The MILLTWO.NC program uses the top left front corner of the workpiece as the Origin, or 0,0,0 location for the machine, relative to Tool #1. In preparation for running the NC program, Tool #1 needs to be manually jogged to this location and the machine position set. The Power Feed and the Jog Handwheel are used to position Tool #1 over the workpiece origin.



1. Use the MDI mode to create a short program to reinstall Tool #1 in the spindle. Tool #1 is the reference tool, so we will be using this tool to set the workpiece origin.
2. Use the Power Feed keypad to jog the tool to the approximate 0,0,0 location on the workpiece. Once the tool is in the general area of the origin, use the jog handwheel to fine position the tool.
3. Once the tool is close to the corner of the workpiece, use the jog handwheel to jog to the machine 0,0,0 location. To use the Handwheel, press the Handwheel button on the pendant. Select the appropriate axis motion key from the motion keys above the Power Feed keys. For now, select the Z key to set the Z axis as the current axis for motion.
4. The LCD display on the pendant now displays “Handwheel Multiplier Select” options (X1 X10 X50 X100) selectable by the functions keys (F1 through F4 respectively).

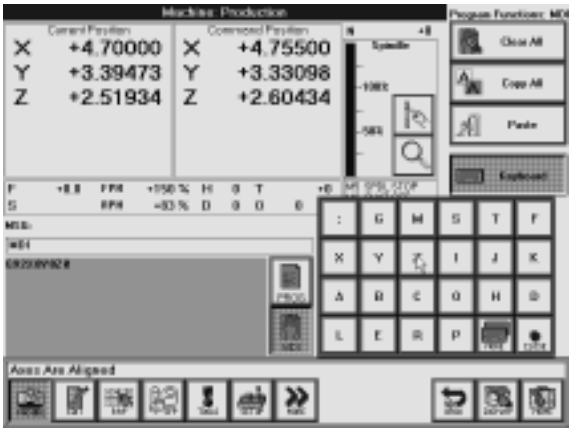
   The Handwheel Multiplier options determine how far the axis moves for each increment on the handwheel. With X100 selected, the axis will move .010" each time the handwheel is incremented. With X50 selected, the axis will move 0.005", X10 will move the axis 0.001" and X1 will move the axis 0.0001. Select a Handwheel Multiplier by pressing the corresponding F Key. For now, select X50 by pressing the F3 key.

5. Rotate the handwheel to jog the Z axis. Each increment on the handwheel will cause the Z axis to move 0.005". Rotating the handwheel counterclockwise will cause the axis to move downward. Clockwise rotation will cause the axis to move upward.
6. Use the Jog Handwheel with the X, Y, and Z axes keys as well as the Handwheel Multiplier options to jog the tool to the 0,0,0 location.

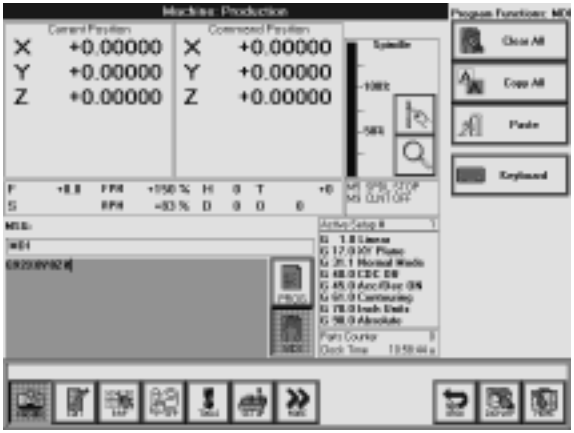
Set the Machine Coordinate

Once the tool has been positioned over the origin of the workpiece, you must set the machine's coordinate system to 0,0,0 before cutting the part. This establishes the reference point from which all tool motions are programmed.

1. Without moving the tool, press the MDI button on the screen. The MDI button is located to the right of the NC Code listing window, above the menu buttons on the bottom of the screen. The NC code will disappear from the window when in MDI mode.
2. The machine position is reset by running a short NC program that resets the current machine position to 0,0,0 using G+M coding. Select the Program Functions button from the Home Menu. The Program Functions MDI menu appears.
3. Open the G+M code keyboard by selecting Keyboard. A small G+M Code specific keyboard appears. This keyboard has all of the letters used in common NC programming. (A QWERTY keyboard is available by pressing the More button on the keyboard.)



4. Using a combination of the keyboard and the numeric keypad, type G92X0Y0Z0. The NC code appears in the NC code window as you type.
5. Close the keyboard by pressing the Close button on the bottom right corner of the keyboard.
6. Run the NC program to set the machine's working coordinates to the current tool location. Press the Cycle Start button on the pendant to run the program. The machine's position will be updated on the display. Note that the X, Y, and Z axes positions now read 0.0000



7. Return to MILLTWO.NC by pressing the PROG button above the MDI button. MILLTWO.NC is now returned as the active NC program.

Run MILLTWO.NC

You are now ready to machine the MILLTWO.NC program. Review the Safety Checklist at this time to ensure that all safety considerations are being met. It is also a good habit to jog the tool slightly above the workpiece before running the program. Once the tool is positioned somewhere above the workpiece, you may run the NC file.

1. Jog the tool above the workpiece. You may use either the Power Feed keypad or the Jog Handwheel to jog the tool.
2. Run the NC program. Start the Machining Process by pressing the CYCLE START button on the pendant. The machine will automatically run your NC program, including the tool changes.
3. Once your NC file is finished, use either the Power Feed or Jog Handwheel to move the tool to a safe location before removing the workpiece.
4. Press in the Emergency Stop button on the controller before opening the Safety Shield and removing the workpiece.

Congratulations, you have just machined your first multiple tool part on the VMC-5000 Machining Center.

Establishing Tool Length Offsets

This is an alternative method of setting tool length offsets. In this procedure a tool gauge plug is used as a reference from all tool lengths are measured.

1. Start the system.
   a. Make sure that all the tools are properly mounted on the ATC.
   b. Start the computer and Machining Center.
   c. Follow the directions on page 21 of this guide to *align* the Machining Center.
2. Install the tool gauge plug.
   a. Select the Tool Unclamp button on the pendant to open the draw bar.
   b. Place the tool gauge plug in the spindle.
   c. Selecting the Tool Unclamp button on the pendant again closes the draw bar.
   d. Place a tool height offset sensor, or other reference surface, on the machine cross slide.
3. Establish the Z zero point. Jog the plug down until it just touches the tip of the tool height offset sensor. The LED on the sensor lights.
4. Set the Z axis coordinate to 0.0. To do this, an NC program needs to be written and executed on the machine. Follow these steps to set the Z axis coordinate to 0.0.
   a. Press the HOME button to return to the Home screen.
   b. Press the MDI button. The MDI button is located to the right of the NC Code listing window, above the menu buttons on the bottom of the screen. The NC code will disappear from the window when in MDI mode.
   c. Select the Program Functions button on the Home Menu. The Program Functions menu allows us to open an on screen keyboard. The keyboard will be used with the numeric keypad on the control to create a one line NC program. The Program Functions MDI menu appears.

d. Open the G+M code keyboard by selecting Keyboard. A small G+M Code specific keyboard appears. This keyboard has all of the letters used in common NC programming.

e. Using a combination of the keyboard and the numeric keypad, type G92Z0.0. The NC code appears in the NC code window.

f. Close the keyboard by pressing the Close button on the bottom right corner of the keyboard.

g. Press the Cycle Start button on the pendant to run the program. The machine's position will be updated on the display. Note that the Z axis position now reads 0.0000

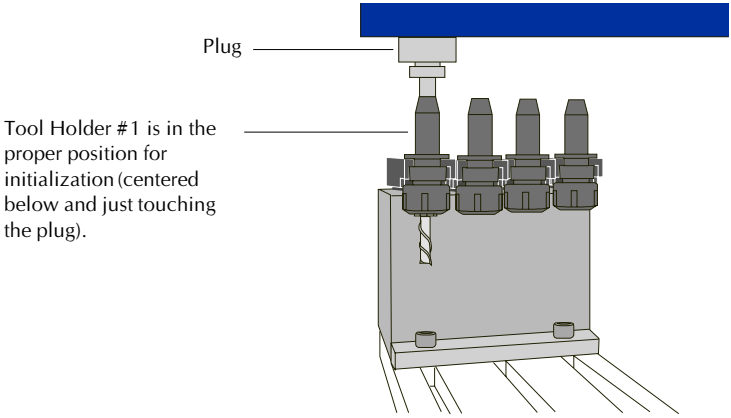
5. Remove the tool gauge plug.
   a. Jog the spindle up, away from the tool height offset sensor.
   b. Select the Tool Unclamp button on the pendant to unclamp the draw bar.
   c. Remove the plug from the spindle.
   d. Selecting the Tool Unclamp button on the pendant again will clamp the draw bar.
6. Set the offset for Tool #1.
   a. Load Tool #1 into the spindle.
   b. Jog Tool #1 down until the tip just touches the tip of the tool height offset sensor. The LEDs on the sensor lights.
      The Z axis coordinate on the control unit now reads a number different from the tool gauge. This value is the tool length offset for Tool #1. Write this number on a sheet of paper.
   c. Select TOOLS from the General Mode Button Bar at the bottom of the screen. The tool setup dialog box will be displayed on the screen. For Tool 1, enter the Z axis value in the Length box.
   d. Jog the tool up, away from the tool height offset sensor.
7. Repeat steps 6a through 6d for each of the remaining tools.

ATC Initialization

The ATC for the VMC-5000 is installed and initialized at the factory. The tool change station locations have been entered into the control system. When a tool change is commanded, the machine recognizes the ATC, knows where it is located, and performs the tool changes. The only configuration the user will need to perform is establish tool length offsets and tool definitions. However, if the ATC is ever removed from the machine and then reinstalled, the initialization procedure must be performed. It is important that the X, Y and Z positions for each of the four tool holders be set properly in order to avoid tool crashes.

Should you ever need to initialize the ATC, use the following procedure. To initialize the tool holders, a tool gauge plug (supplied) is inserted into the draw bar, and each tool is jogged to the plug to set the position.

1. Make sure the tool holders are mounted in the stations on the ATC.
2. Turn on the controller.
3. Turn on the Machining Center.
4. Select the Unclamp Tool button on the pendant to open the draw bar.
5. Insert the tool gauge plug, smallest end down, into the draw bar. Make sure the plug is inserted all the way into the spindle.
6. Selecting the Unclamp Tool button on the pendant again will close the draw bar.
7. Using the Power Feed and the Handwheel, jog the spindle until the tool gauge plug is centered and just touching the top of the tool holder (see diagram on the next page). If it does not touch during the setup, you may get a "Tool in Position Sensor Not Activated" message when automatic tool changes are programmed.
8. Choose the More button from the icons at the bottom of the screen, and then select the Service Password level.
9. Choose the System Configuration button and then the Machine Application button.



10. From the file cards select Spindle/Tool Changer Config.
11. Select Enable Tool Changer and then the Enable Auxiliary Mechanism Menu (if not already selected).
12. Select the Tool Change Positions file card.
    a. Set the Z axis Manual TC Position = 9.00
    b. Set the Z axis Auto TC Position 1 = 9.00
    c. Set the Z axis Auto TC Position 2 so that a tool held in the tool fork is fully inserted into the spindle when the tool fork is fully raised. To do this:
        1. Jog Z axis fully up.
        2. Unclamp Draw bar and remove plug.
        3. Use Mech Jog key to raise tool fork.
        4. Use the handwheel to jog the Z axis down until the tool holder is fully inserted into the spindle.
        5. Read the Z axis value from the pendant and enter this in TC Position 2.
13. Go to the User Defined Positions (select the More Selections arrows in the menu bar at the top of the screen).
    a. Set the X and Y data for Positions 1 through 4 for tool load sta tions. Use the tool gauge plug for this operation.

b. Set X and Y data for positions 5 through 8 for tool unload.

   X Position = X load position + 1.5

   Y Position = Y load position.

   e.g., P1x = 1.700 P5x = 1.7 + 1.5 = 3.2

   P1y = .797 P5y = .797

14. Under Axes Configuration, set X axis low limit. Take the lowest of P1x - P4x and subtract 0.1
15. Set P9. The average of P1x through P4x.
16. Set P10. The average of P5x through P8x.
17. Verify initialization.

    a. Enter the MDI mode and create the following program:

       [Start]

       M06T01

       M06T02

       M06T03

       M06T04

       (GOTO [START])

    b. On the pendant, select Rapid Override (the LED should be lit) and turn the Feed Rate Override to slow. This way, the machine will execute the tool changes at a very slow rate. If things are not set up correctly you will be able to see potential tool crashes and stop the machine before they happen.

    c. Select Cycle start to run the MDI program. Verify that the tool changes take place and no errors are encountered.

    d. Exit MDI mode and return to running the machine.

Error Messages

Following are some of the more important error messages you may see on the the controller while opereating the VMC-5000.

Message:

<Activity> has failed to execute

Cause:

The programmed activity has failed.

Remedy:

Correct problem and retry. Check for other alarm messages that may give more specific information. Consult Technical Support if the problem persists.

Message:

Air Pressure Failure

Cause:

The air pressure has fallen below 75 psi.

Remedy:

Check shop air supply or lines to the VMC-5000.

Message:

Tool Drawbar Unclamped

Cause:

An attempt was made to run the machine with the drawbar unclamped.

Remedy:

Clamp Spindle Drawbar.

Message:

Unexpected Tool Present in Spindle

Cause:

A tool was sensed in the spindle when no tool should have been there.

Remedy:

Clear all tools from spindle and ATC and realign the mechanism.

Message:

Tool Absent

Cause:

No tool was sensed in the spindle when one should have been there.

Remedy:

Clear all tools from spindle and ATC and realign the mechanism.

Message:

Load manual tool into spindle

Cause:

Manual tool change must be performed by operator.

Remedy:

Press cycle start to continue.

Message:

Unload manual tool from Spindle

Cause:

Manual tool change must be performed by operator.

Remedy:

Press cycle start to continue.

Message:

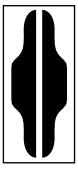
Tool Draw Bar Unclamped

Cause:

The draw bar failed to unclamp.

Remedy:

Check drawbar for proper operation, as well as limit switches and solenoids.



LIGHT MACHINES
C O R P O R A T I O N
444 East Industrial Park Drive
Manchester, New Hampshire 03109-5317
(603)625-8600 FAX 603.625.2137 800/221-2763

34-7708-0000
June 1996