VMC-5000
Machining Center
User's Guide

VMC-5000
Machining Center
User's Guide

Manchester, New Hampshire, U. S. A.   June, 1996

Contents

Section 1: Introduction

Section 2: Installation

Section 3: Quick Start

Section 4: Supported G+M Codes

Section 5: Optional Machining Capabilities

Section 6: Maintenance

Section 7: Dismantling and Storing

Section 8: Working with spectraCAM

Appendices

WARNING

The operation of rotating machinery should only be attempted by experienced,knowledgeable individuals!

Read the entire contents of this guide before running NC programs on the VMC-5000 Machining Center.

To avoid possible injury *always* observe the safety precautions described in this User's Guide.

Section 1: Introduction

What is the VMC-5000 Machining Center?

Operating Conditions

Machining Center Options

Using This Guide

Other Documentation Resources

What is the VMC-5000 Machining Center?

The VMC-5000 Machining Center is a three-axis tabletop milling machine with the Acramatic 2100 Computer Numerical Control. This state-of-the-art machining center provides high-speed, high-efficiency part processing.

The machining center can machine a large variety of parts in plastics, aluminum and steel. You can use the VMC-5000 Light Machines' spectraCAM software to design and machine parts all from one operator station.

The VMC-5000 Machining Center includes:

❏ A VMC-5000 Machine

❏ An Acramatic 2100 Control Operator Station

❏ A Machine Pendant

The VMC-5000 Machining Center is shipped fully assembled, and is easy to install. Complete instructions for installation of the machining center, as well as software installation instructions, are included in Section 2.

Features

Some of the VMC-5000 Machining Center's most notable features include:

- ❏ A one-horsepower permanent magnet spindle motor

- ❏ An R8 industry-standard spindle taper for the VMC-5000 and an EX16 spindle taper for the VMC-5500.

- ❏ Spindle speeds from 200 to 5,000 RPM (500 to 10,000 RPM optional) for the VMC-5000, and from 200 to 42,000 RPM for the VMC-5500.

- ❏ EIA RS-274D standard G&M code programming

- ❏ Multiple tool programming

- ❏ Feed rate and spindle speed override functions

- ❏ A built-in full-screen NC program editor

- ❏ spectraCAM CAD/CAM software

- ❏ An operator station featuring a touchscreen and keypad

- ❏ A machine pendant

Intended Use of the Machining Center

The intended use of the VMC-5000 Machining Center is as a conventional computer numerical controlled (CNC) vertical mill used in industrial environments.

The Process

A trained operator affixes a workpiece to the machine's table and a cutting tool into the machine's spindle. The cutting tool rotates as a computer controls the machine's table and spindle movements. These motions result in the cutting tool being brought into contact with the workpiece and the cutting tool removing material from the workpiece.

The operator specifies appropriate feeds, speeds, and cutting depths for the type of tooling and workpiece material being used so as to not overload the tool or the machine.

While the machine is in use, the operator is out of the danger zone.

Cutting Tools and Fixtures

The cutting tool may be any off-the-shelf tool designed for vertical milling operations. The tool's shank is restricted to 3/4" diameter to fit a standard R8 collet. Typically, the diameter of the cutter is no more than 1/2".

You can use standard off-the-shelf fixtures to affix a workpiece to the machine's table.

Workpiece Materials

The workpiece may be ferrous or nonferrous, including, but not limited to steel, aluminum, plastic, and wax.

Misuse of the Machine

Do NOT use the machine for tasks it was not designed to perform. Improper use or modification of the machine may damage the machine and may result in serious injury or death.

Misuse of the machine may result from:

❏ Improperly affixing a nonstandard or oversized cutting tool to the machine.

❏ Improperly affixing the cutting tool to the machine.

❏ Manually holding or feeding the workpiece into the machine.

❏ Improperly affixing the workpiece to the machine.

❏ Improperly defining feeds, speeds, and depth of cut while machining.

❏ Entering the danger zone during use.

❏ Introducing hazardous materials when machining.

❏ Removing the safety shields or making any other unauthorized modifications to the machine.

❏ Cutting unacceptable materials.

❏ Using damaged or dull tooling.

All of these misuses could result in the cutting tool, workpiece, and fixtures being ejected from the machine.

WARNING:

Use the machine only for tasks it was designed to perform. Improper use of the machine may result in serious personal injury.

The Machine Components

There are, of course, more components on the machine than those shown here. But to begin, you need only be concerned with the depicted major components.

The Safety Shield encloses the machine to help protect the operator from flying chips. A magnetic Shield Interlock Switch prevents the machine from operating with the shield open.

The X, Y and Z motion of the machine is performed by closed-loop DC Servo Drive Motors on each axis. There are also Limit Switches (beneath the way covers, next to the drive motor on each axis) to prevent the machine from traveling beyond its limits on each axis.

The Spindle Head supports a 1hp permanent magnet spindle motor.
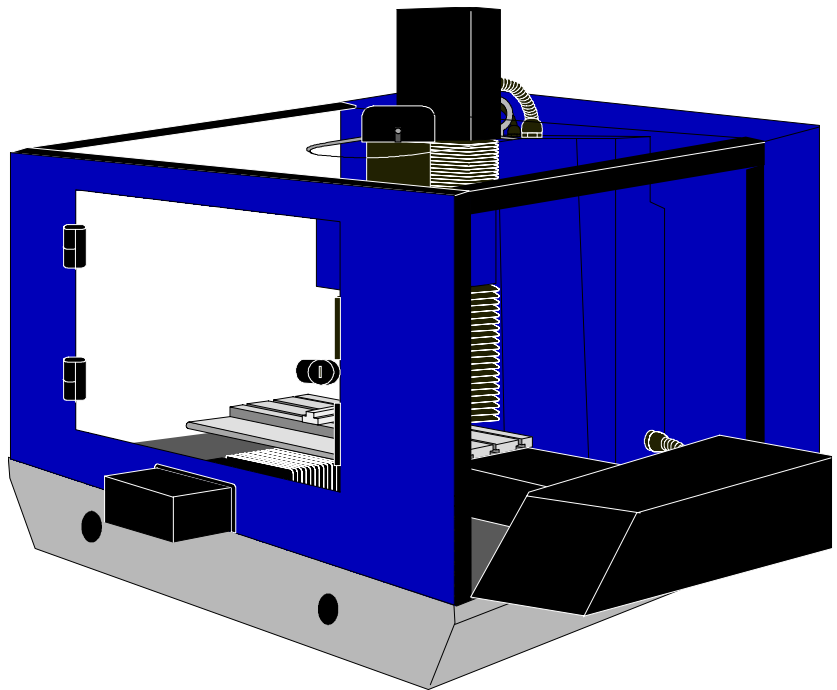
The VMC-5000 machine.

The Rear Panel

The Rear Panel houses the power controls and circuit breakers for the machine.

The Key Lock switch keeps unauthorized persons from turning on the machine. When the Key Lock is in the unlocked position, you can turn on power to the machine using the green power button on the operator station. The Main Power light illuminates when you turn on the power to the machine. The Drive Power light illuminates when you turn on power to the servo amplifiers.

The machine has five sets of circuit breakers for main power, the operator station, the servo motors, auxiliary outputs, and the spindle.

When a circuit breaker blows, the circuit breaker switch pops out, revealing white at the top of the switch.

To reset the circuit breaker, push the switch up and back into the circuit breaker until it locks in place.

The Operator Station Components

The operator station completely houses the Acramatic 2100 Control and spectraCAM and provides the operator interaction with the control.

The Control

The Acramatic 2100 consists of two 486DX processors, each with 20MB RAM and 540MB hard drives. The control runs under Windows-NT, a 32-bit operating system. The control features a PLC compatible I/O, and is expandable with serial and parallel ports.

Apart from the control software and spectraCAM inside, there are important components of the operator station on both the front and back panels.

The Front Panel

The touchscreen, keypad, floppy drive, and keyboard connector are all located on the front panel.

Use the touchscreen and keypad to interact with the control software. You can adjust the touchscreen when you change sitting positions. The keypad is helpful for entering NC codes into the control software.

Touchscreen

Keypad

Floppy Drive

Keyboard Connector
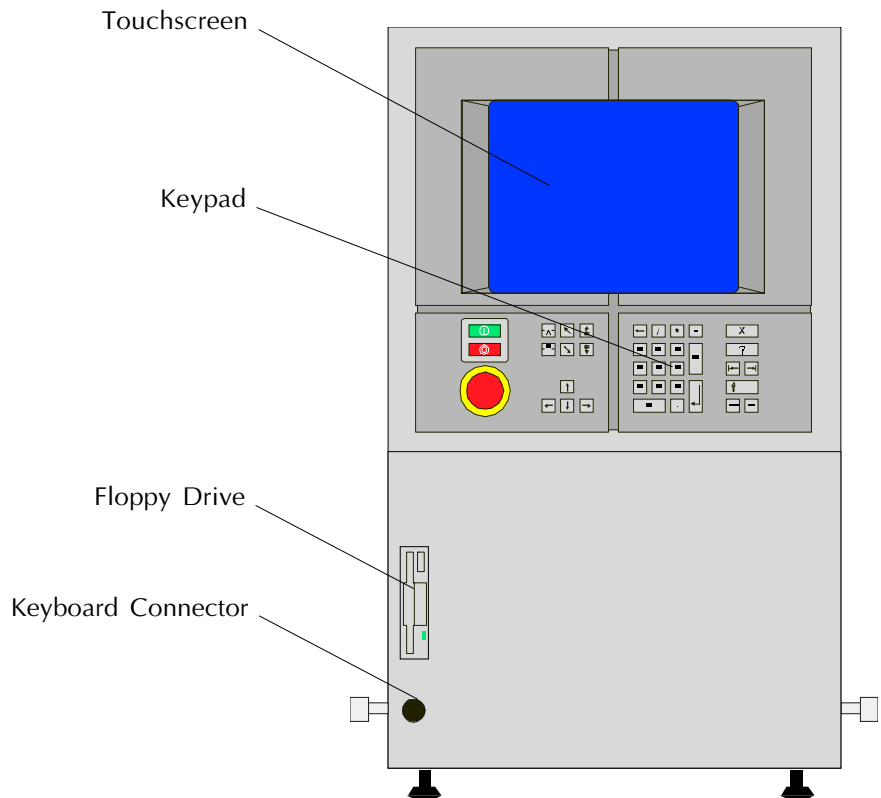
Use the floppy drive to load files into the operator station. These can be NC files for the control software, or DXF or SCM files for spectraCAD/CAM.

You can connect a computer keyboard to the operator station through the keyboard connector. Although you do not need an external keyboard to run the machining center, it is helpful when you are using spectraCAD/CAM.

The Back Panel

The I/O connector, parallel port, serial port, LED diagnostic lights, and interface cable are all located on the rear panel.

Use the I/O connector to connect the control with external devices such as robots.

Use the parallel port on the back panel to connect a printer or other device requiring a parallel port to the control. Such devices include modems, video equipment, CD-ROMs and LAN adapters.

You can connect a mouse through the serial port. Although you do not need a mouse to run the machining center, it is helpful when you are using spectraCAM.
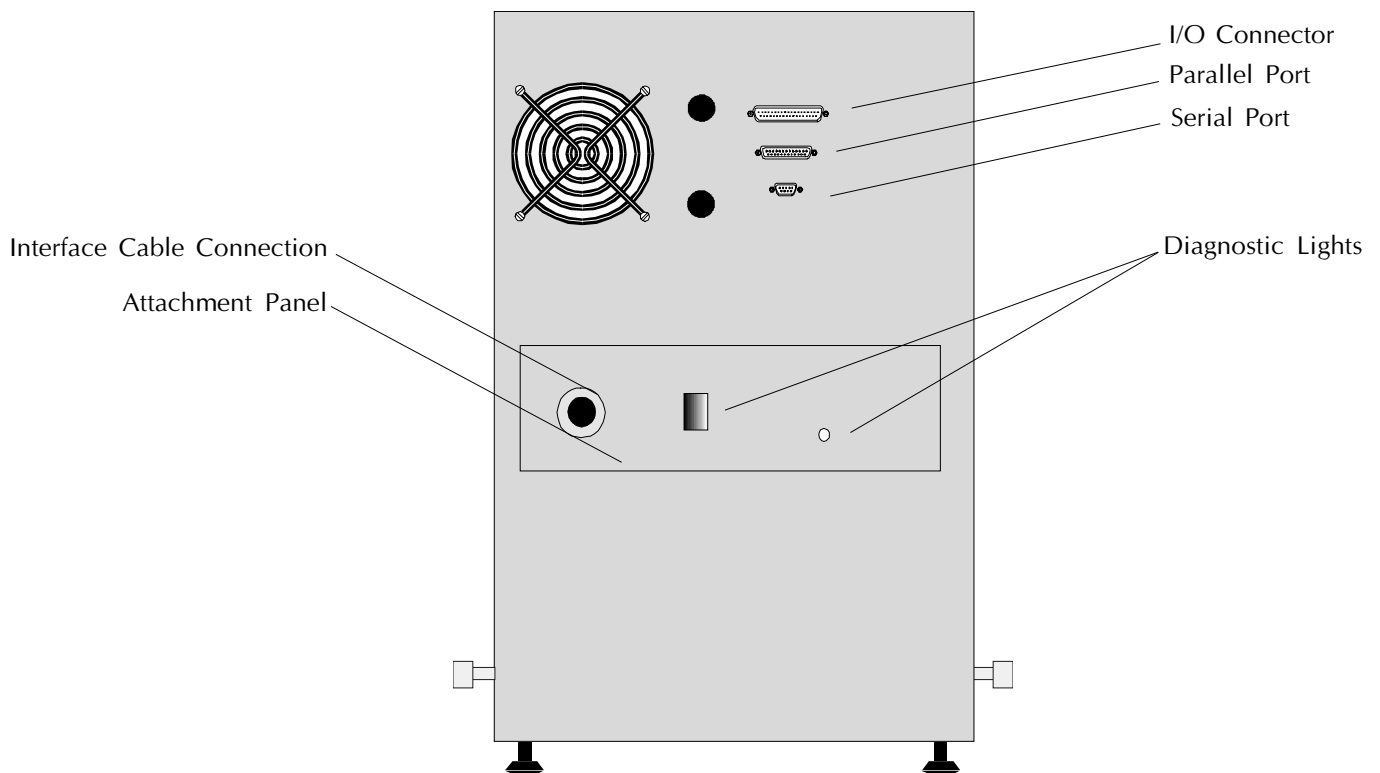
The Machine Pendant

The machine pendant is an important component of the machining center. On the pendant are; the handwheel, feedrate and spindle speed override pots, pushbuttons, and powerfeed controls. The pendant allows you to have access to the workpiece and setup functions without having to move from the machine to the control. The LCD display on the pendant provides NC program coordinates for the selected axis and supports additional functions with the menu keys.

The pendant is the only way to move the machine in either a manual mode or to start an NC file. Use the Cycle Start button to move the machine.
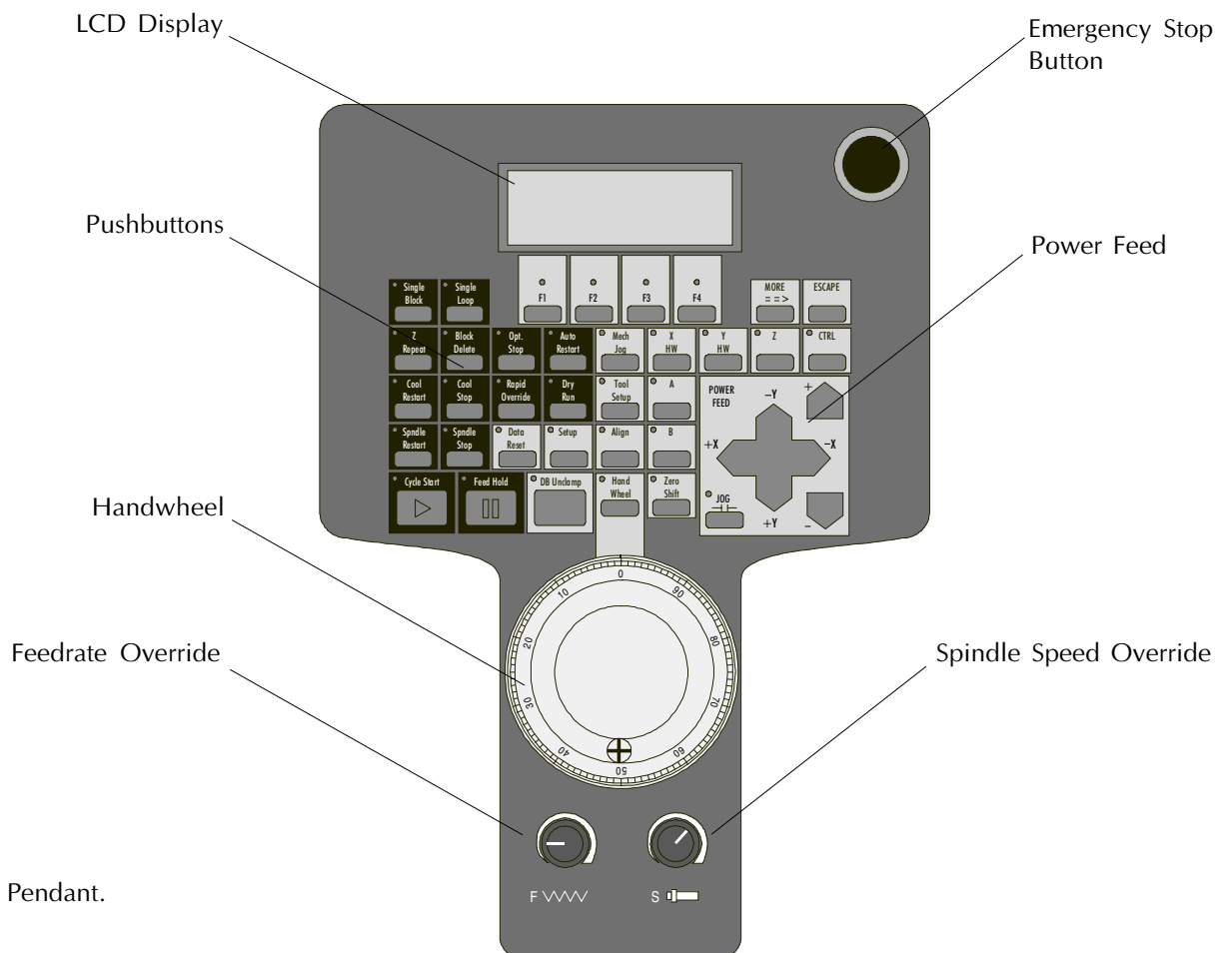
The Machine Pendant.

The Accessory Kit

The Accessory Kit that comes with the machining center contains all the tools and hardware necessary for installing and maintaining the mill. It also contains a collet and tools to get you started; other tool holding devices and tools are available as options.

Operating Conditions

You must meet all work area, power, grounding, and environmental conditions before using the machine.

Work Area

You must allow at least 12.5 square feet (1.16 square meters) of work space area to operate the machine.

Main AC Power Supply

AC power input must satisfy the following requirements:

Voltage

The voltage depends on the model you are using.

= 115 VAC + 10% - 15% or

= 230 VAC + 5% - 10%

Frequency

= 47-63 Hz single phase, 3 wire

Power Rating

Operator Station and Pendant: = 450 VA

Machining Center: 1800 VA

Grounding

See Chapter 2 of the Control Reference Manual (34-7704-0000) for more information on grounding the control.

A positive earth grounding system for the machine and the control is required for proper and safe operation. The ground path must have a DC resistance of 3.5 Ohms or less to a true earth ground.

Ground the machining center in conformance to all applicable electrical codes.

Environmental Conditions

The machining center is specified to operate within the following environmental limits:

Temperature

Operating: 45 to 85 deg. F (7 to 29 deg. C)

Storage: -40 to 167 deg. F (-40 to 75 deg. C)

Transition: 10 deg c/hr

Humidity

5% to 80% without condensation

Altitude

Operating: 0 to 10,000 feet (3,048 meters)

Non-operating: 0 to 40,000 feet (12,192 meters)

Shock

Non-operating: 10g in 11 milliseconds

Vibration

Operating: 5Hz to 500Hz, 0.5G acceleration

Non-operating: 10Hz to 500Hz, 1.0G acceleration

Machining Center Options

The following options are available for the VMC-5000 Machining Center from Light Machines Corporation.

❏ **Machining Center Machinist Kit (ACC-5110)** The Machinist Kit includes a 3" milling vise with hold-down clamps, a 4-piece R8 collet set, a 9-piece high-speed end mill set (including 2 ball end mills), a 52-piece hold-down set, a Jacob's chuck, an R8 to 2JT arbor, and a 3/8" x 12" cushion-grip T-handle hex key. *Not available for the VMC-5500.*

❏ **End Mill Package (ACC-5120)** The ACC-5120 End Mill Package consists of the following end mills, all with 3/8" shanks: two 1/8" HSS center cutting 4 flute end mills, two 3/16" HSS center cutting 4 flute end mills, two 1/4" HSS center cutting 4 flute end mills, two 5/16" HSS center cutting 4 flute end mills, two 3/8" HSS center cutting 4 flute end mills, two 1/8" HSS ball 2 flute end mills, and two 1/4" HSS ball 2 flute end mills. *Not available for the VMC-5500.*

❏ **Jacob's J6 Drill Chuck with R8 to J6 Arbor (ACC-5130)** The ACC-5130 consists of a Jacob's chuck that opens from 0" to 1/2". The chuck comes with a key and R8 to J6 arbor. *Not available for the VMC-5500.*

❏ **Quick-Change Tooling (ACC-5141)** Quick-Change Tooling provides an easy way of integrating multiple tools within one NC program. It comes with the tool body, five tool holders, a tool holder stand, three allen keys and an electronic tool height offset sensor. The Quick-Change Tooling Kit also has optional positive drive tool holders. *Not available for the VMC-5500.*

❏ **Low Profile Clamping Kit (ACC-5180)** This kit includes a custom rail and stop with four clamps for work holding.

❏ **52-Piece Hold-Down Set and Vise (ACC-5185)** The ACC-5185 includes 52 pieces for holding piece to the workpiece and a 3" vise with the capacity to open up to 4".

❏ **4th Axis 5C Rotary Positioner (ACC-5223)** The ACC-5223 is a complete rotary positioning device that mounts horizontally on the VMC-5000 Machining Center and provides simultaneous 4-axis machining capability. The rotary positioner incorporates a 5C collet system with manual collet closer. When installed: Available X axis travel is 7", Z axis travel is 5-3/4".

❏ **5C Collet Set for 5C Rotary Positioner (ACC-5370)** The ACC-5370 includes 1/4", 1/2", 3/4", and 1" 5C collets.

❏ **5C 3" 3-Jaw Self-Centering Chuck for 5C Rotary Positioner (ACC-5372)** The ACC-5372 is an industrial grade chuck for use with the 4th Axis 5C Rotary Positioner. Use this chuck when collets are not available for the working diameter. The chuck has a runout of 0.003".

❏ **Mobile Workstation with Storage Cabinets (ACC-5590)** The ACC-5590 is designed for use with all VMC-5000 systems. The dove-gray plastic laminated top provides a comfortable 60" wide by 30" deep work surface. The classic blue storage cabinets have keyed alike locks and include one cabinet with a door and one cabinet with three drawers. The cabinets are mounted on a mobile base with four 6" diameter heavy-duty ball bearing casters. The overall height of the workstation is 31-1/4".

❏ **Engraver Tool Set (ACC-5710)** The ACC-5710 includes one carbide conical nose 60° end mill with 1/8" shank, one carbide 4 flute end mill 1/16" with 1/8" shank, one carbide 4 flute end mill 1/32" with 1/8" shank.

❏ **Low Voltage Light (ACC-5720)** The ACC-5720 provides light to the machine's work area. This is a factory installed option.

❏ **Vacuum System (ACC-5730)** The ACC-5730 Vacuum System is designed for use with the VMC-5000 Series machining centers. It includes a vacuum with noise reduction features, a GFT hose, a 1-1/2" diameter nozzle mounting assembly, and an electrical relay. *Not available for the VMC-5500.*

❏ **Machinable Wax Milling Stock, Large Blocks (PKG-9110)**

❏ **Machinable Wax Milling Stock, Small Blocks (PKG-9120)**

❏ **Air Vise (PNU-4115)** The Air Vise has jaws that open to a maximum of 3.0 inches. The vise comes with a solenoid valve and all the tubing required to interface with 1/4" pipe fittings from filtered and regulated shop air (50-125 psi).

❏ **Portable Air Compressor (PNU-4535)** The Portable Air Compressor is an intermittent duty 3/4 HP oil-less air compressor with a built-in color coded regulator and pressure selection chart. It delivers 2.1 CFM at 90 PSI.

Using This Guide

This guide is intended for experienced personnel only. It is not intended for inexperienced operators or maintenance personnel, nor is the information intended to constitute a training program or provide sufficient background for electrical maintenance or setup.

In addition to this introduction, this guide is divided into eight sections.

Section 2
Follow the procedures in Section 2 to assemble and connect the machining center and to install accessories.

Section 3
Section 3 is a quick start that takes you step-by-step through a sample NC program. It introduces you to machining center components and operations. Note, however, you should be familiar with all safety regulations before you begin operating the machine.

Section 4
The VMC-5000 Machining Center supports standard EIA RS-274D G&M NC programming codes, along with an array of other codes. All of the supported codes are reviewed in Section 4.

Section 5
Section 5 explains the VMC-5000 Machining Center's optional machining capabilities: multiple tool programming, quick change tooling, and machining with the 4th Axis Rotary Positioner. If you plan to use any of these options you should read this section.

Section 6
Good maintenance practices assure a longer life for the VMC-5000 Machining Center. Section 6 shows you how to care for the machining center and Operator Station.

Section 7
Section 7 explains how to dismantle, package, store, and transport the VMC-5000 Machining Center.

Section 8
Section 8 explains how to use spectraCAM from the Operator Station.

Other Documentation Resources

Apart from this user's guide, the VMC-5000 documentation set also includes other documents that provide additional information on the Acramatic 2100 Control and Light Machines' spectraCAM software.

❏ The *Control Reference Manual* (34-7704-0000) provides maintenance information, hardware functional descriptions, system configuration information, service procedures, and other related reference information.

❏ The *Operation Guide* (34-7705-0000) provides information on using your operator station and its components and provides procedures for completing various tasks using the control.

❏ The *Programming Manual* (34-7706-0000) provides information on the G and M codes the control uses.

❏ The *spectraCAM Milling User's Guide* (34-6741-0000) provides information on using spectraCAM. This publication includes a quick start section for teaching you the basics of spectraCAM.

❏ The *spectraCAD User's Guide* (34-7791-0000) provides information on using spectraCAD. This publication includes a quick start section for teaching you the basics of spectraCAD.

This user's guide refers to these other publications when applicable.

Section 2: Installation

Getting Ready

Unpacking the Machining Center

Connecting the Components

Getting Ready

Check Your Shipment

The first thing you should do after receiving your VMC-5000 Machining Center is locate the packing slip in the Accessory Kit. This slip lists all of the items you should have received with your machining center. Check all of the items on the list.

Inspect the machining center for signs of visible damage. If you notice any damage, or if you find any discrepancies between the packing slips and the items you have received, call Light Machines' Customer Service Department (800/221/2763) or 603-625-8600.

Register Your Machining Center

You'll find a registration card in the small box with the documentation and software disks. It's important that you clearly print all the requested information and return this card to Light Machines.

You need to know the serial numbers of your machining center when you call for service. Both the operator station and the machine have serial numbers. The serial numbers are located on the back of each component.

Prepare Your Work Place

The work place should be clean and uncluttered, with enough room to open and assemble the machining center and its components. You need 12.5 square feet (1.16 square meters) to operate the machining center.

You should have a sturdy bench on which to place the machine and the operator station. The machining center weighs approximately 650 lbs (295 kg). Your table must be able to safely support this weight.

Keep in mind that, when troubleshooting, you may need to check the circuit breakers on the rear panel of the machine. Place the operator station in an area unexposed to metal chips or cutting fluid.

Be sure you place the machining center near an appropriate electrical supply.

Finally, be sure you have met all of the required operating conditions listed in Section 1.

Unpacking the Machining Center

The VMC-5000 Machining Center comes to you carefully packed in three shipping containers:

❑ The machine on a pallet with a cardboard carton stapled to the pallet.

❑ The operator station in a cardboard carton.

❑ The accessory kit, including manuals, in a cardboard carton.

The component weights of the machining center are as follows:

❑ Machine: 500 lbs (227 kg)

❑ Operator station: 130 lbs (59 kg)

❑ Pendant: 4 lbs (2 kg)

If the shipping containers are not correctly oriented, reposition each carton correctly using proper lifting techniques.

Unpacking the Machine

1. Position the pallet on which the machine is attached near the table on which you'll set the machine. Place the table against a wall for maximum support.

IMPORTANT: Be sure to keep the pallet and all of the original cartons in which the VMC-5000 Machining Center was shipped. Should you need to return any components to the factory, repack them exactly as you received them.

Light Machines will not be responsible for any damage caused during shipping when components are not returned in their original cartons.

2. Remove the staples that attach the cardboard carton to the pallet.

3. Lift the cardboard carton off the pallet.

4. Inspect the machining center chassis for signs of visible damage such as a broken shield, a dent in the chassis or damaged cables.

   If any damage is noted, or if you find any discrepancies between the packing slip and the items received, call Light Machines' Customer Service Department (800/221/2763 or 603-625-8600).

5. Remove the four bolts holding the machining center base to the pallet. Keep the bolts and other packaging materials.

Lifting the Machine off the Pallet

The machine is equipped with a lifting ring. Attach an appropriate lifting device to this ring to move the machine. Contact supervision if you are unsure about the proper procedures for lifting the machine.

CAUTION

Do not lift the machine by the safety shield or the X axis extension box. Lifting the machine by methods other than the lifting ring will damage the machine.

---

1. Lift the machine off of the pallet using the lifting ring.

   When you lift the machine, it tilts forward 20°.

2. Guide the machine to its workplace.
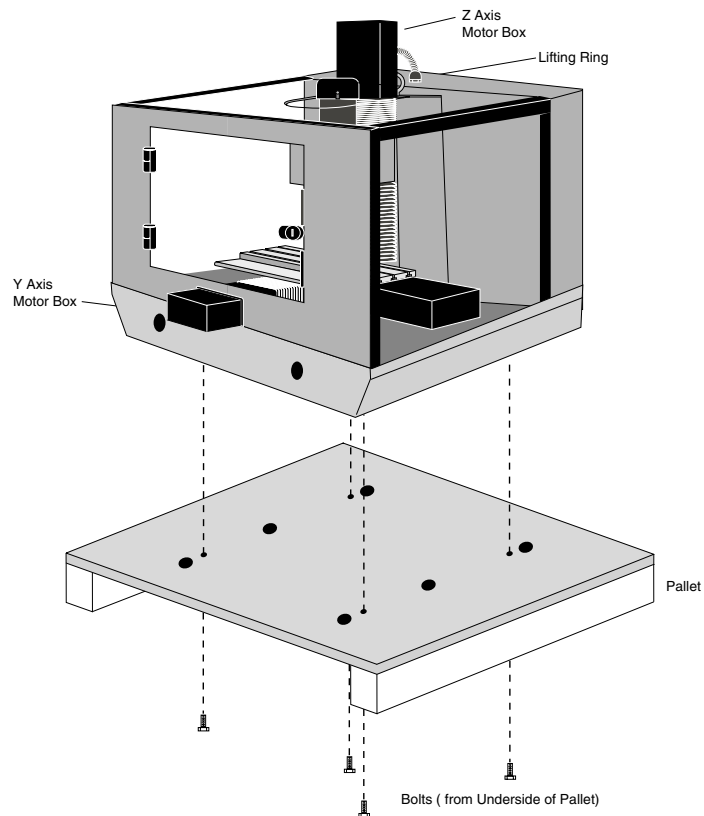
3. Remove the protective paper from the safety shield.

WARNING:

Failure to follow
instructions on this page
may result in serious
personal injury.

Unpacking the Operator Station

Now unpack the operator station.

1. Position the carton containing the operator station near the
    table on which you'll set the machining center. Place the table
    against a wall for maximum support.

2. Open the carton and remove packing materials from the operator
    station.

3. Inspect the operator station chassis for signs of visual damage.

    If you notice any damage, or if you find any discrepancies between
    the packing slip and the items received, call Light Machines'
    Customer Service Department (800/221/2763 or 603-625-8600).

Lifting the Operator Station off the Pallet

Now lift the operator station off the pallet. The operator station
weighs 57 lbs (25.86 kg). You will need two people to lift it. Con-
tact supervision if you are unsure about the proper procedures for
lifting and carrying.

With two people, lift the operator station using the handles on
both sides. Place the operator station close enough to the machine
so that you are able to connect the two.

The pendant is pre-wired into the operator station and is packaged
in a small box with the operator station. Remove the pendant from
the box in which it is packaged.

Connecting the Components

The following paragraphs explain the procedures for interconnecting the operator station with the machine.

Positioning the Machine and Operator Station

The interface cable between the machine and the operator station is ten feet long. This allows you to position the operator station up to ten feet away from the machine. Make sure to place all components on a stable, flat surface.

Installing the X Axis Servo Extension Box

You need to install the protective box that covers the opening for the X axis servo extension. This piece and the necessary hardware are packaged in a small carton within the machine carton.

If your machine is equipped with the Flood coolant option, you will find a tube of sealant in with the hardware. Use this sealant to form a bead on the inside edge of the extension box. This will seal the cover and prevent leakage of coolant.

To install the X axis servo extension box:

1. Remove the extension box from the carton.

   a. If you are installing the box on a flood coolant machine, lay the extension box on its side.

   b. Using the tube of sealant, form a bead on the edge of the box that will mate with the opening in the enclosure.

2. Locate the ten phillips head screws and flat washers in the carton.

3. Remove the plastic covering from the safety shield on the side of the machine with the X axis extension opening.

4. Locate the ten screw holes for attaching the extension box. From the inside of the machine's safety shield, insert a screw through one of the screw holes.

5. Place a washer on the screw.

6. Hold the extension box up to the extension opening and insert the screw into the extension box.

Installing the X Axis servo extension box.

Safety Shield

X Axis Extension Box
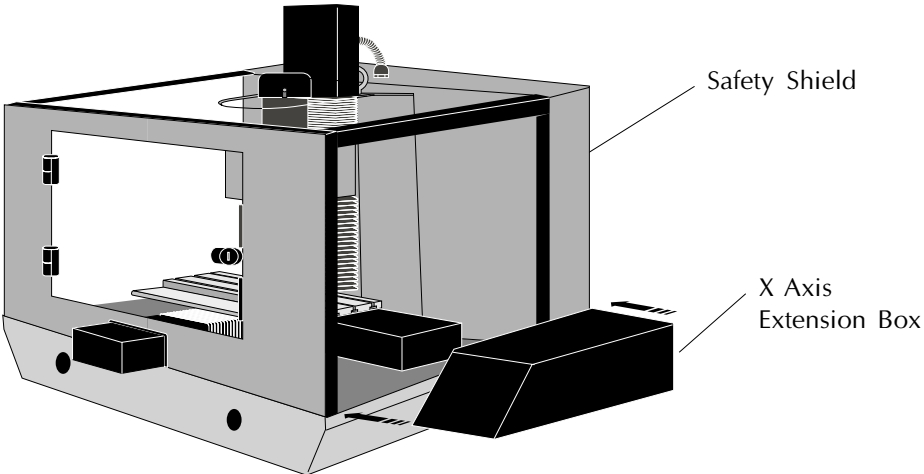
7. Insert the remaining screws and washers to attach the extension box to the machine.

Installing the Air Filter Regulator (5500 Only)

You need to install the air filter regulator assembly on the left side panel towards the back of the machine. This piece and the necessary hardware are packaged in a small carton within the X axis extension box carton. To install the air filter regulator assembly:

1. Hold the assembly up to the machine on the left side panel towards the back of the machine. You can see the air fitting coming out of the machine.

Back of the machine

Installing the air filter regulator assembly.

Air Filter Regulator Assembly
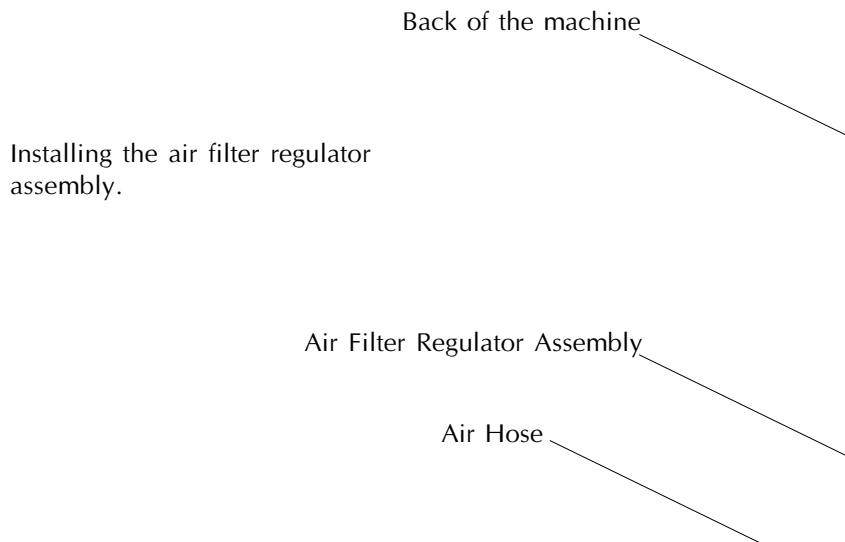
Air Hose

2. Push the air fitting marked OUT on the assembly into the air fitting on the machine.

3. Use the two phillips head screws provided to attach the assembly to the machine.

4. Plug the air hose provided into the air fitting on the back of the machine.

5. Plug the other end of the air hose into the air fitting marked IN on the assembly.

Connecting the Operator Station to the Machine

The interface cable, which is attached to the machine, contains all of the connections between the operator station and the machine. At the end of the interface cable is the attachment panel. The connectors from the interface cable come out the back side of the attachment panel.

There are four connectors that come from the interface cable:

❑ A male 25-pin connector

❑ A female 25-pin connector

❑ A male 50-pin J1 connector

❑ An AC power connector

These connectors correspond to the connectors on the back panel of the operator station. Each connector from the interface cable fits only one connector on the operator station.

25-pin connector (female)

25-pin connector (male)　　　　　　　　　　　　AC Power Connector

The operator station connector panel.

LED Display

50-pin J1 connector (male)
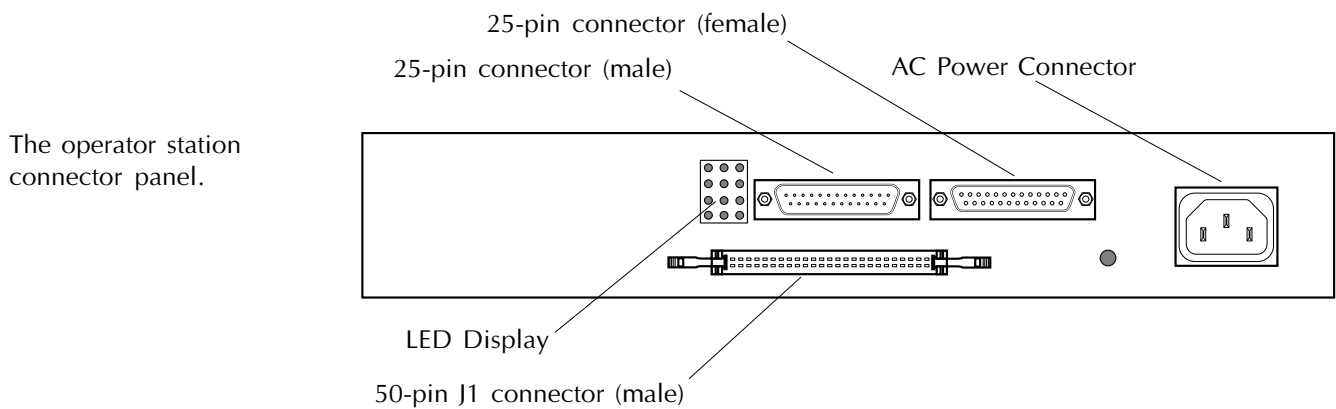
1. Hold the attachment panel up to connector panel on the rear of the operator station.

2. Connect the female 25-pin connector to the male 25-pin connector on the operator station connector panel. Use the connector screws to secure the connector into place.

3. Connect the male 25-pin connector to the female 25-pin connector on the operator station connector panel. Use the connector screws to secure the connector into place.

4. Connect the female 50-pin J1 connector to the male 50-pin J1 connector on the operator station connector panel. Use the J1 locking tabs to lock the connector in place.

Connecting the machine and the operator station (rear view).
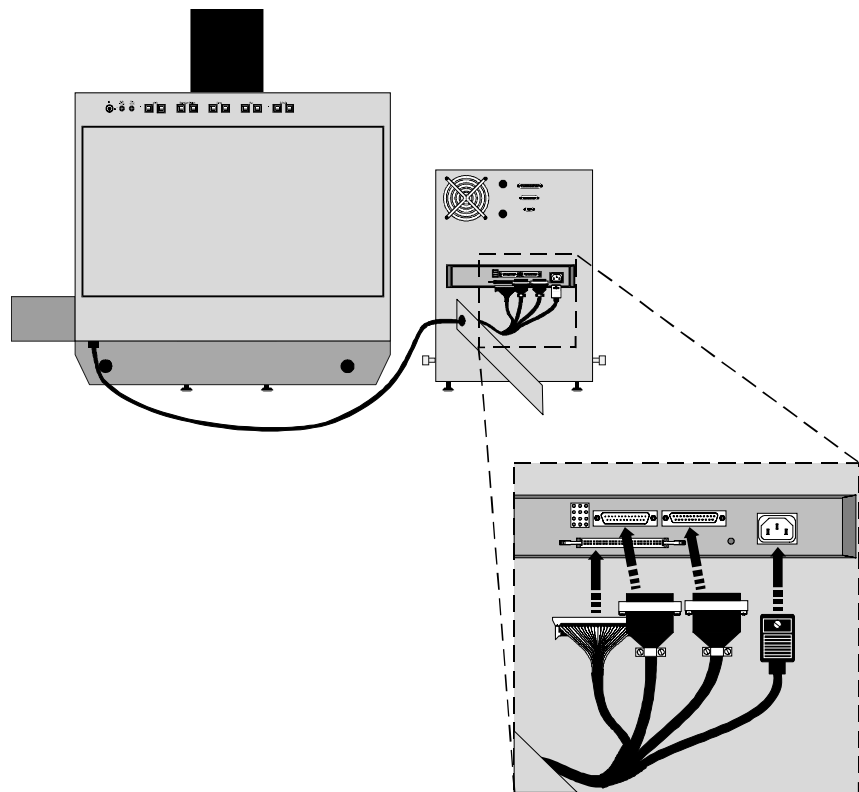
5. Connect the power cord into the power receptacle on the operator station connector panel.

6. Mount the attachment panel to the back of the operator station, covering the connector panel.

Connecting Power to the Components

Note: The VMC-5521 requires 230VAC.

Plug the VMC-5000 into a 115VAC, 60Hz wall outlet. Plug the VMC-5500 into a 230VAC, 50-60Hz, 10A, single phase wall outlet.

The machine has an attached power cord terminated with the appropriate plug. Insert the plug end of this cord into the correct wall outlet. This powers the entire machining center, as the interface cable brings electricity to the operator station.

Be sure the power source for the machining center meets the electrical requirements listed in Section 1 of this guide.

Section 3: Quick Start

Getting Started

This guide should get you started on the road to being able to machine many different parts on the machining center. In this section, you learn how to start the VMC-5000 Machining Center and the Acramatic 2100 Control and begin performing basic operations. Remember, however, that this is only a quick start; refer to other sections of this manual for specific details on the machining center and its features.

This section illustrates both a single tool program and a multiple tool program. MILLONE.NC is a single tool program that you can easily set up and run on the machining center. Use a 1/4" End Mill and a 3 x 2 x 1.5" piece of machinable wax for MILLONE.NC. MILLTWO.NC is a multiple tool program. Use both a 1/4" and 1/8" End Mill for MILLTWO.NC. You need to equip your machine with a quick change tooling package before you can run this program.

The Touchscreen and Keypad of the Operator Station. For more information, refer to the Operation Guide (34--7705-0000).

Touchscreen

Machine On
Machine Off

Emergency Stop button

Keypad
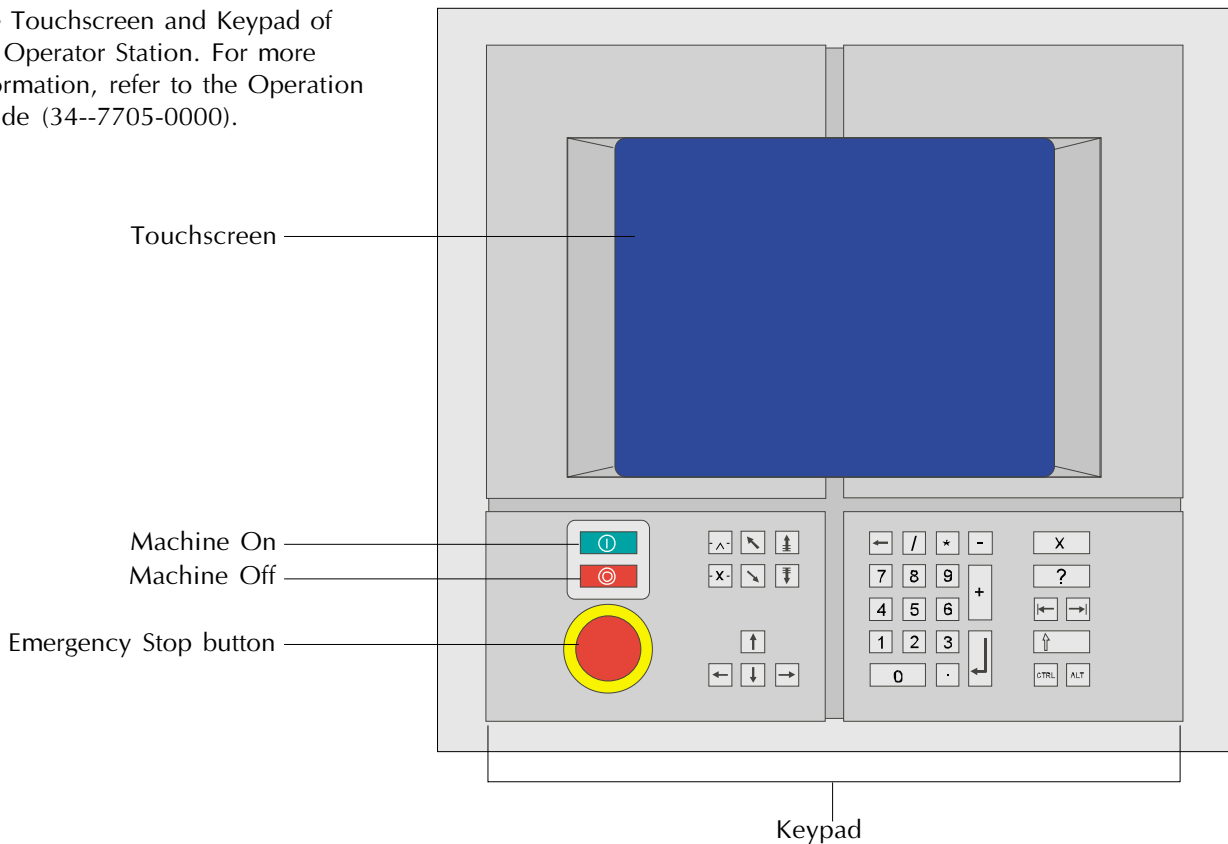
Start the Acramatic 2100 Controller

You cannot run the machine until you turn on the controller. Follow these steps each time you switch the machine on.

1. Press and release the green power switch located on the left hand side of the Control Panel, just above the Emergency Stop button on the operator station. This turns on the controller.

   The start up procedure begins. This takes several minutes as the system goes through a series of internal checks and loads various software applications.

2. After the software is loaded, press the Home button located at the lower right hand corner of the touchscreen. This button brings the software back to the Home screen.

Display Area   Function Buttons

The control Home menu screen. For more information on the control screen, refer to the Operation Guide (34--7705-0000).

Function Menu

Status Area

General Mode Button Bar   Mode Specific Button Bar

Button Bar Area
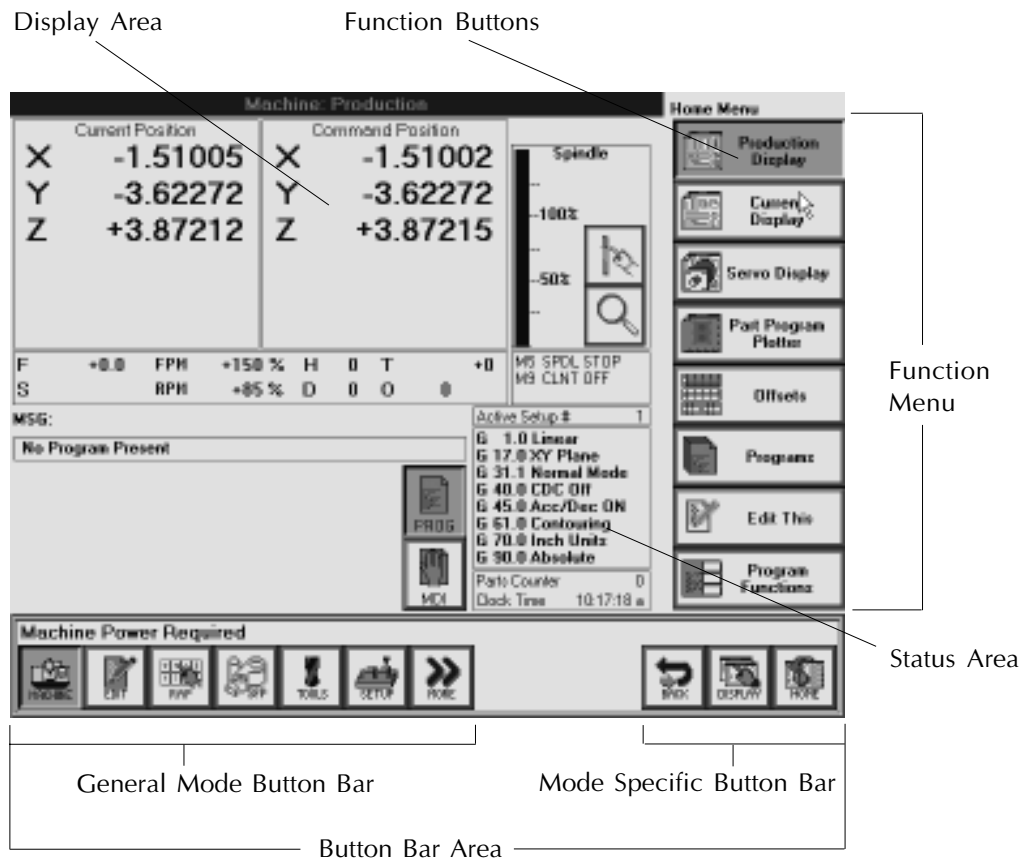
Remember the Home button. If you ever get lost as you navigate through the software, simply press the HOME button to return to the starting location.

Power Up the Machining Center

After the controller is up, you are able to power up the machining center. As a safety precaution, the machining center is not automatically powered up when you switch on the controller. You must power up the machining center separately. You need the pendant to power up the machining center. See the illustration below to find the major components of the pendant.

LCD Display

Emergency Stop Button

Push-buttons

Power Feed

Handwheel

Feedrate Override

Spindle Speed Override
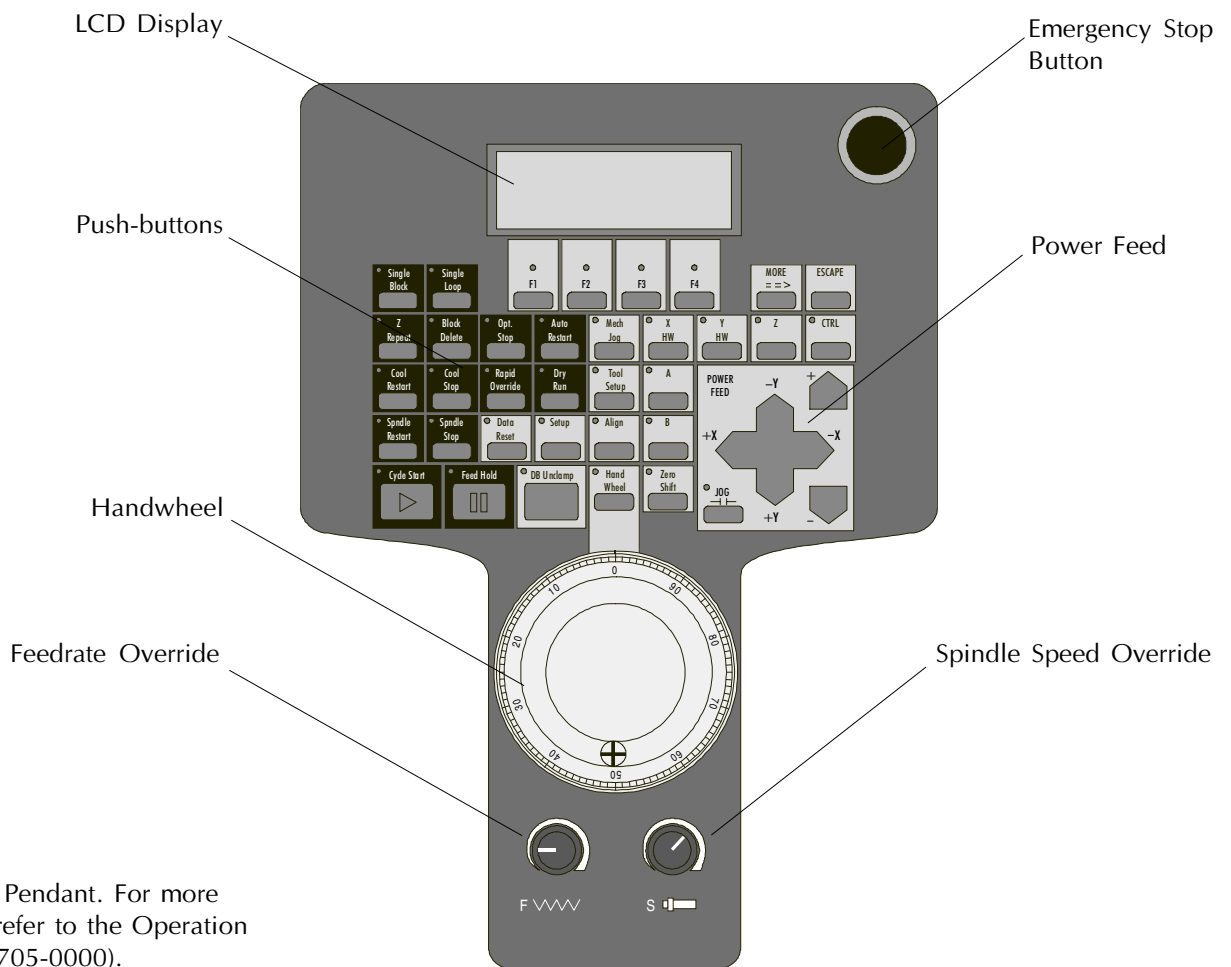
The Machine Pendant. For more information, refer to the Operation Guide (34--7705-0000).

To power the machine:

1. Pull out the Emergency Stop button on the pendant. Locate the Emergency Stop button at the top right hand corner of the pendant. Make sure that this button is pulled out.

2. Pull out the Emergency Stop button on the operator station. You must deactivate the Emergency Stop button on the operator station by pushing the Emergency Stop button inward, rotating it counterclockwise and then releasing.

3. Press in and hold the green power switch on the operator station for two seconds to power up the machining center.

   The machining center is now powered up and ready for use. A message appears above the Button Bar on the controller screen, stating that the machine is unaligned.

Now you need to align the machine.

Align the Machining Center

You need to "align" the machine before you can use it. Aligning the machine allows you to establish a point of origin at the ends of travel on the X, Y, and Z axes of the machine. The machining center then uses this point as a reference for all machine coordinate movements. The machine can then move consistently to the same location when you program it to do so.

1. Locate and Press the Align key on the pendant.

    The LCD display on the pendant now reads Axes - Align above the F1 key.

2. Press and hold the F1 key.

    The machine automatically aligns itself to the +X, +Y and +Z Axes Limit Switches. During the homing sequence a message appears above the Button Bar on the controller screen indicating that the machine is performing the alignment cycle.

After the machine finishes aligning itself, a message "Axes are Aligned" appears above the Button Bar on the controller screen. The machine is now aligned and is ready for use.

3. Press the Escape key on the pendant to take the pendant out of the alignment mode.

    The LED on the Align key stays lit, indicating that the machine is homed.

Load an NC Program

The Acramatic 2100 Control has an active NC file set for production applications. There is additional storage space on the hard disk for storing additional NC files, however the control keeps a list of up to 500 "active" NC files. Use the following procedure to load a new NC file onto the hard disk and to transfer it into the active NC file area.

1. Insert the Samples disk containing the NC program into the floppy disk drive.

2. Press the Program Functions button on the Home menu.

   The Program Functions menu appears.

3. Select the Programs button from the Program Functions menu.

   The NC Part Program dialog box appears.
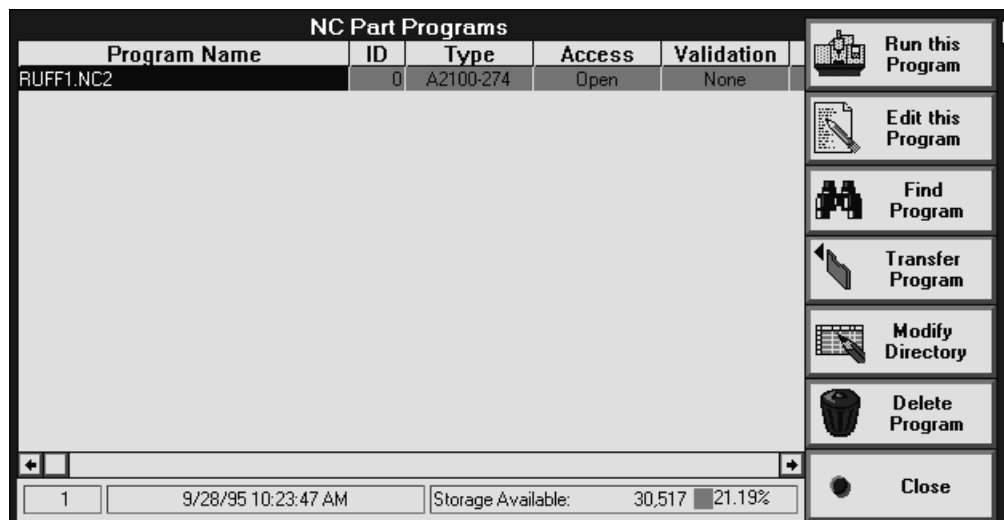
4. Select the Transfer Program button to transfer a new NC file into the active NC program area.

   The Transfer Program menu appears.

5. Click on the Transfer Program In button.

   The Transfer To Program Store dialog box appears.

6. Click on the Floppy Drive button.

7. Click on the Browse button to view the contents of the floppy disk drive.

The Select NC Program File dialog box appears.
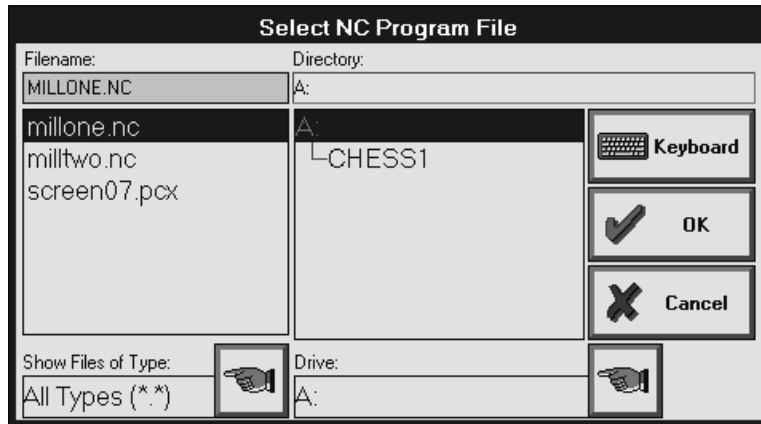
8. Click on the Hand Icon adjacent to the Show Files of Type field to narrow down the number of files in the list.

A pop out listing of the most common files types appears.

9. Select EIA274 (*.NC) from the file type pop out menu.

The filename window now only displays files with the NC extension.

10. Select MILLONE.NC from the file listing, then click on OK to return to the Transfer to Program Store dialog box.
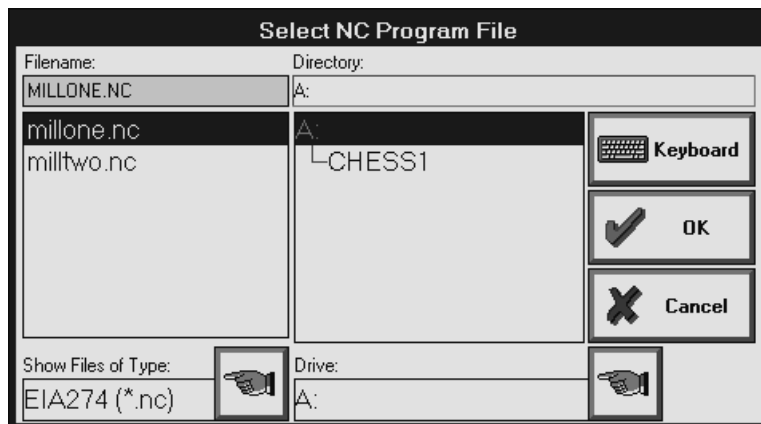
11. Select Start Transfer from the Transfer to Program Store dialog box to transfer MILLONE.NC to the active program store area.

MILLONE.NC is now in the Active NC Store Area and as a result you can see it is on the NC Programs listing.
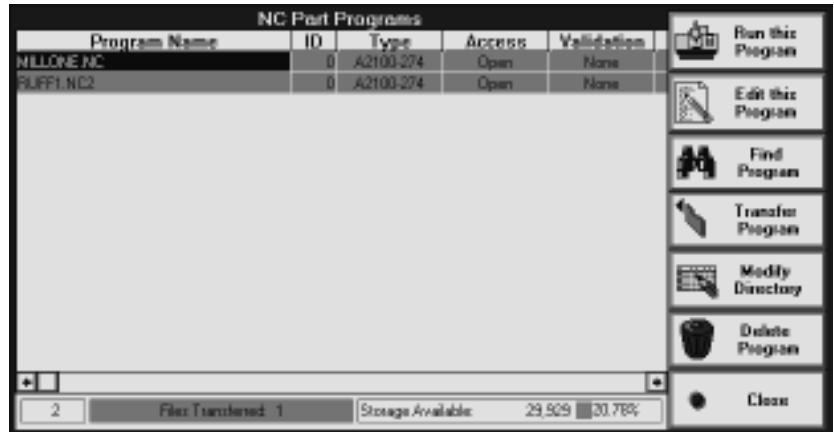
12. If it is not currently selected, select MILLONE.NC from the list and then click on the Run This Program button.

13. Click on the Home button to return to the Home menu.

Notice that MILLONE.NC is now the active NC program for the machining center. The codes for MILLONE.NC appear in the program window on the left hand side of the screen.
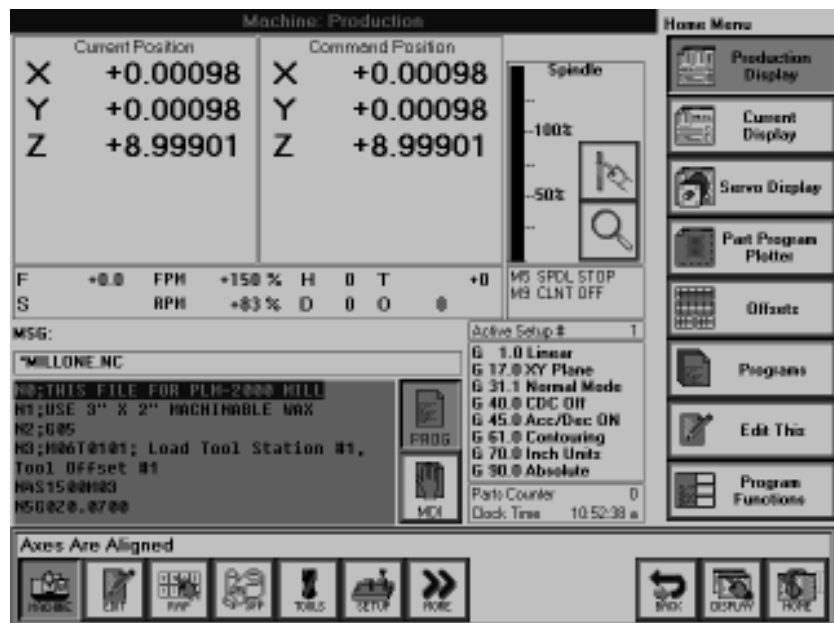
Mount the Workpiece on the Cross Slide

Use the power feed option on the pendant to jog the machine to
any location convenient for mounting the workpiece, hold down,
and tool on the machine.

1. Select an axis direction key from the Power Feed keypad on the
    pendant. Press and hold the key to move the axis. Release the key
    to halt motion.
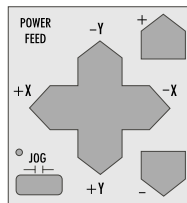
2. Press in the Emergency Stop button on the controller, then
    open the safety shield.

3. Mount a 3" x 2" x 1.5" piece of machinable wax on the cross
    slide. Use a hold-down to secure the workpiece.

4. Mount an 1/8" End Mill in the spindle to cut the part.

5. Close the safety shield and then pull out the Emergency Stop
    button on the controller.

    Whenever you press the Emergency Stop button on the machin-
    ing center, power is completely shut off to the machine. Now
    you must manually restart the machine.

6. Press in and hold the green Power On button on the controller for
    two seconds.

    This returns power to the Machining Center.

If you are running a multiple tool program (MILLTWO.NC in
place of MILLONE.NC) for this tutorial, skip ahead to the Mul-
tiple Part Programming portion of this section, on page 3-20. If
you are running MILLONE.NC, please continue with the next
section on page 3-13.

Jog Tool To Origin of the Workpiece

The MILLONE.NC program uses the top left front corner of the workpiece as the point of origin, which is the zero location for each of the machine's axes.

The zero point of origin on the X, Y, and Z axes.
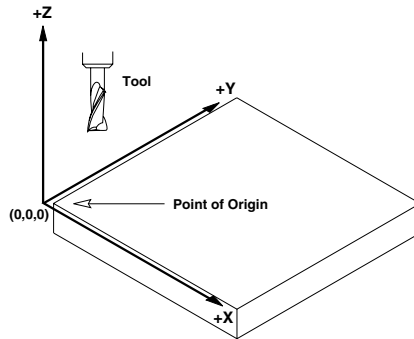
In preparation for running the NC program, you need to jog the tool to this location and set the machine position.

1. Use the Power Feed keypad to jog the tool to the approximate 0,0,0 location on the workpiece.

    When the tool is in the general area of the origin, use the Jog Handwheel to fine position the tool.

2. Press the Handwheel button on the pendant.

3. Select the appropriate axis motion key from the motion keys above the power feed keys. Select the Z key to set the Z axis as the current axis for motion.

The LCD display on the pendant now displays the Handwheel Multiplier Select options (X1 X10 X50 X100). Use the function keys (F1 through F4 respectively)to choose an option. The Handwheel Multiplier options determine how far the axis moves for each increment of the handwheel. If you select X100, the axis moves .010" each time you move the handwheel one increment. If you select X50, the axis moves 0.005", X10 the axis moves 0.001" and X1 the axis moves 0.0001. Select a Handwheel Multiplier by pressing the corresponding F Key.

```
< = = = = =  HANDWHEEL  = = = = = >


Y    +0.00000
MULTIPLIER SELECT
    X1        X10        X50        X100
```

```
   ○          ○          ○          ○
   F1         F2         F3         F4
```
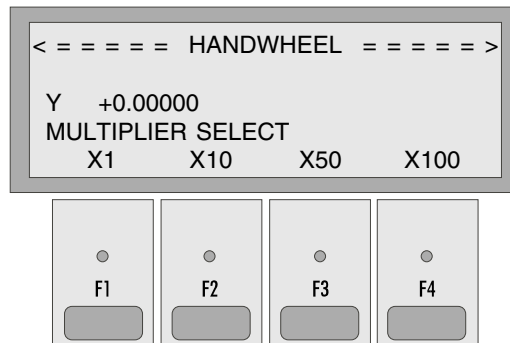
4. Select X50 by pressing the F3 key.

5. Rotate the handwheel to jog the Z axis.

   Rotating the handwheel counterclockwise causes the axis to move downward; clockwise rotation causes the axis to move in the positive direction.

5. Using the Jog Handwheel, the X, Y and Z axes keys, and the Handwheel Multiplier options, move the tool to the 0,0,0 location.

Set the Machine Coordinate to 0,0,0

After you have positioned the tool 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. Press the MDI button on the touchscreen, located to the right hand side of the NC Code listing window, above the menu buttons on the bottom of the screen.

   The NC code disappears from the window when you are in MDI mode.

The NC code disappears from the window when you enter MDI mode.
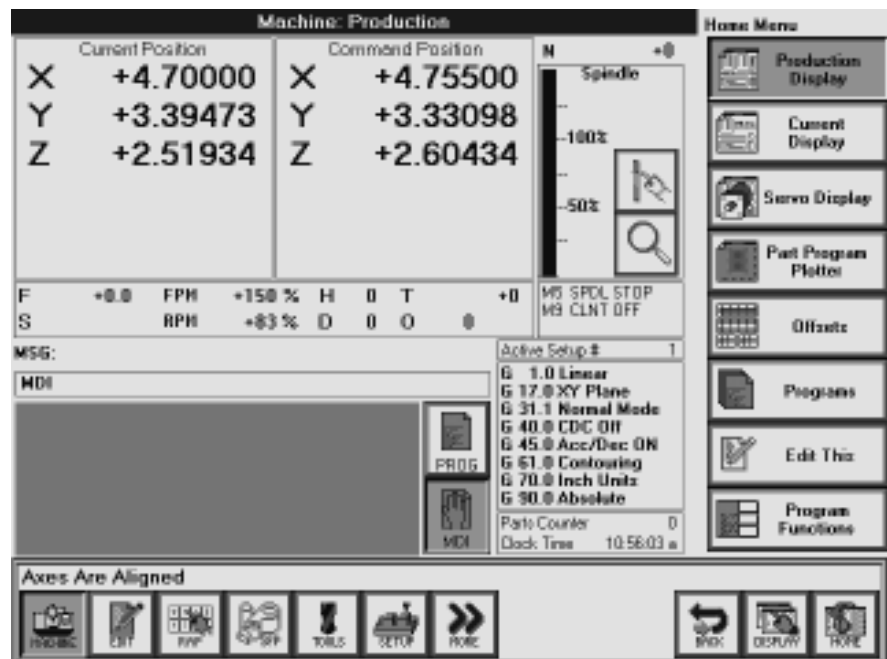
Now open an on screen keyboard to create an NC file. You need to reset the machine position by running a short NC program that resets the current machine position to 0,0,0 using G+M coding. To do this, bring up the on screen keyboard. Use the on screen keyboard and the numeric keypad on the control to create a one line NC program.

2. Click on the Program Functions button from the Home Menu.

   The Program Functions MDI menu appears.

3. Click on the Keyboard button to open the G+M code keyboard. A small keyboard containing all of the letters used in common NC programming appears. (You can access a QWERTY keyboard by pressing the More button on the keyboard.)
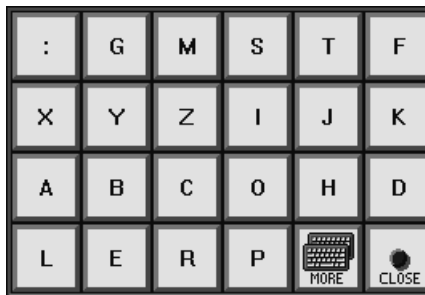
4. Using a combination of the keyboard and the numeric keypad, type G92X0Y0Z0.

The NC code appears in the NC code window as you type.
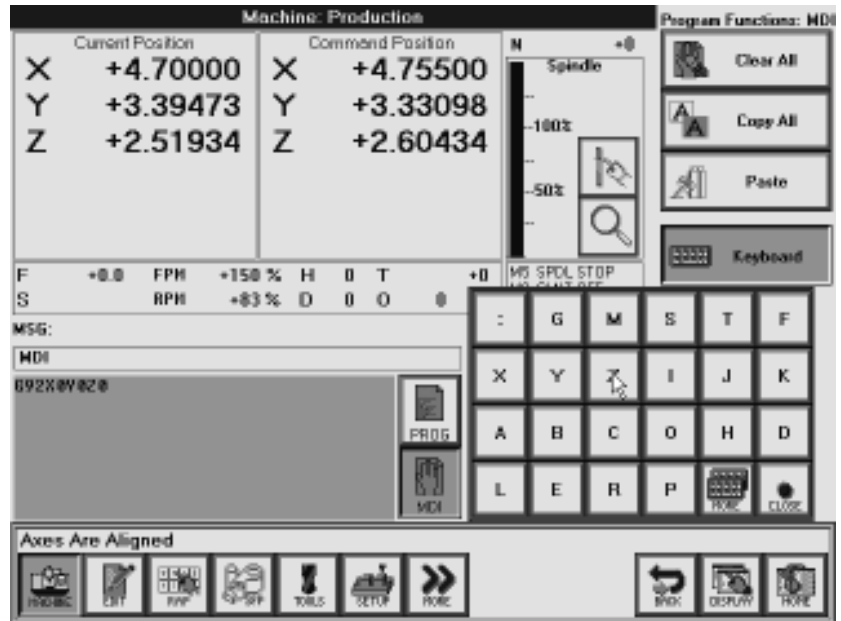
5. Click on the Close button to close the keyboard.

   Now run the NC program to set the machine's working coordinates to the current tool location.

6. Press the Cycle Start button on the pendant to run the program.

   The control updates the machine's position on the display. Note that the X Y and Z axes positions now read 0.0000
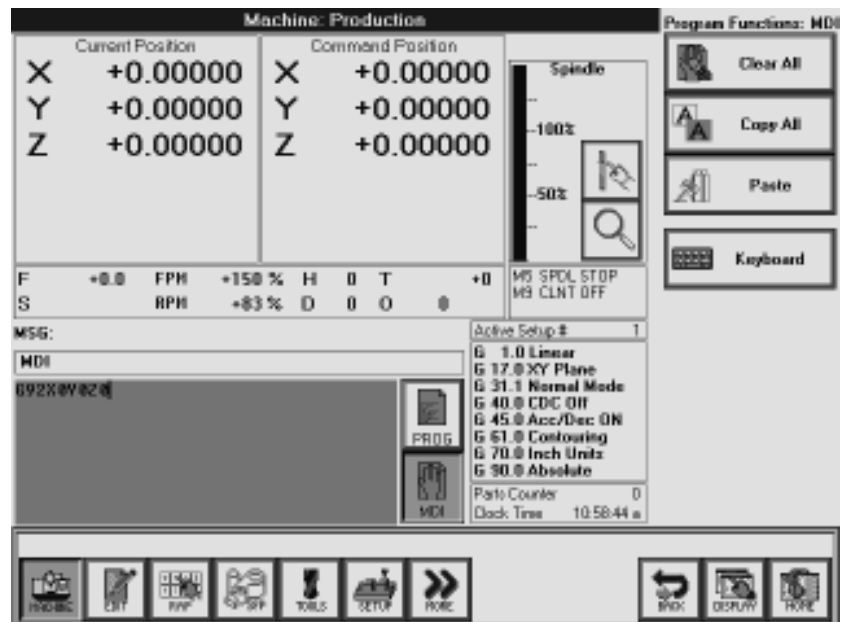
7. Return to MILLONE.NC by clicking on the Prog button above the MDI button.

   MILLONE.NC returns as the active NC program.
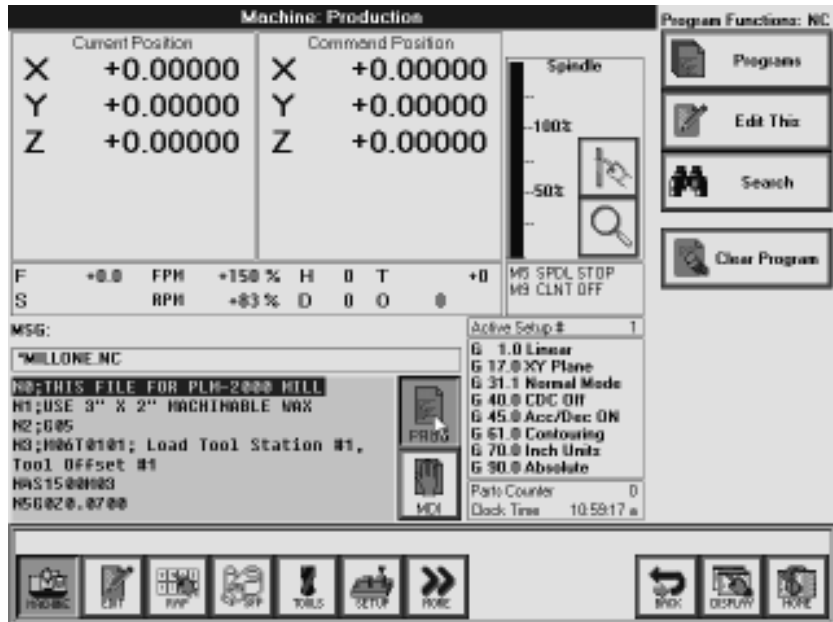
Run MILLONE.NC

You are now ready to actually machine the MILLONE.NC program. Review the Safety Checklist at this time to ensure that you meet all safety considerations.

WARNING:

Read safety precautions before operating the machine. Failure to follow safety instructions may result in serious personal injury.

It is a good habit to jog the tool slightly above the workpiece in preparation to running the part. After you position the tool above the workpiece, you may run the NC file.

1. Use either the Power Feed keypad on the pendant or the Jog Handwheel to jog the tool above the workpiece.

    If you use the Power Feed, press the + key to jog the tool upward.

    If you use the Jog Handwheel, press the Handwheel button on the pendant, then press the Z key on the pendant to select the Z axis. Select an appropriate Handwheel Multiplier from the function keys, then rotate the Handwheel clockwise to raise the spindle.

2. Press the Cycle Start button on the pendant to run the NC program.

    The machine automatically runs your NC program. Be sure to watch the machine as it runs your program. Be prepared to press the Emergency Stop button in the event of a problem.

3. When the machine has finished running your program, use either the power feed or Jog Handwheel to move the tool to a suitable location in preparation for removing the workpiece.

4. Press in the Emergency Stop button on the controller before opening the Safety Shield and removing the workpiece.

5. Remove the workpiece from the machine and inspect it.

Congratulations, you have just machine your first part on the VMC-5000 Machining Center. Before continuing on to more difficult machining situations, be sure you read and understand all of the VMC-5000 documentation.

Multiple Tool Setup

After you have transferred the MILLTWO.NC program from the floppy disk, and mounted the workpiece on the cross slide, you must perform the Multiple tool setup.

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.

Tool 1   Tool 2
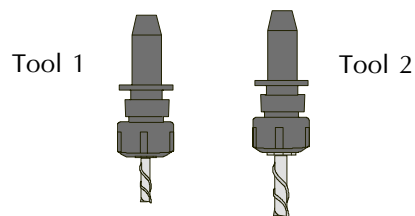
As you can see from the diagram, tool #1 is shorter than tool #2 (your setup may vary). When the NC program is run, the spindle head will have to be offset by the difference in tool lengths to ensure that the tools cut to the proper depth relative to the stock. In the case of the tools shown in the diagram, 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 known as the Tool Length Offset. For this part 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 does, 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). Install Tool #1 in Spindle, and set the Tool Length Offset Sensor on the Machine Cross Slide.
2). 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.
3). 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.

3). 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.

4). 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.

5). Jog the tip of the tool so it just touches the tool height offset sensor. The LED's on the sensor should light, indicating contact has been made.
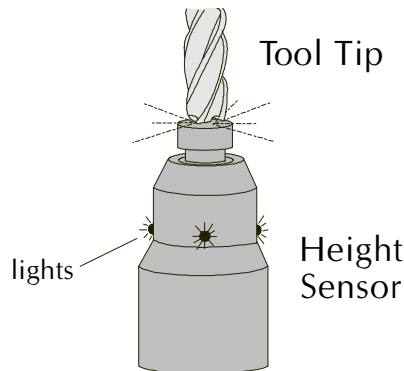
Set Z Axis Coordinate

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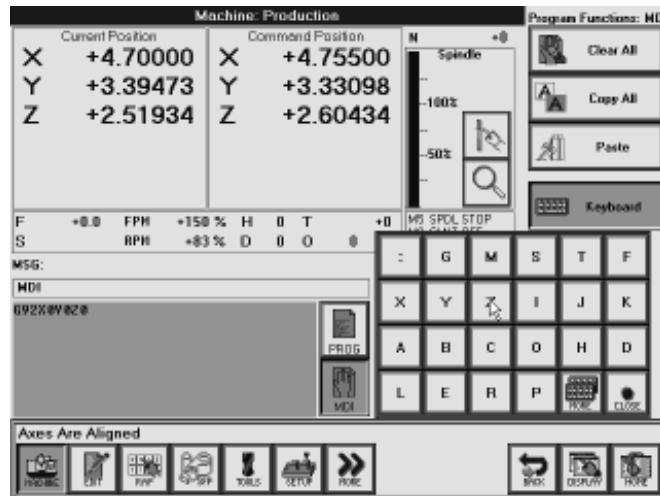
4). Open the G+M code keyboard by selecting Keyboard. A small G+M Code specific keyboard appears. This keyboard has all of the letters used in common NC programming. (A QWERTY keyboard is available by pressing the More button on the keyboard.)

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 in 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.
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 should light.

Note that the Z axis coordinate on the control unit now reads a number different from Tool #1. If Tool #2 is longer than Tool #1, the Z axis position will read a positive value. If Tool #2 is shorter than Tool #1, the Z axis position will read 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 will be 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 Automatic Tool Changer. When Auto Load is selected, tools are automatically loaded from and unloaded to the ATC.

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 the Data Reset button to save the tooling information to the controller.

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 will be used to position Tool #1 over the workpiece origin.
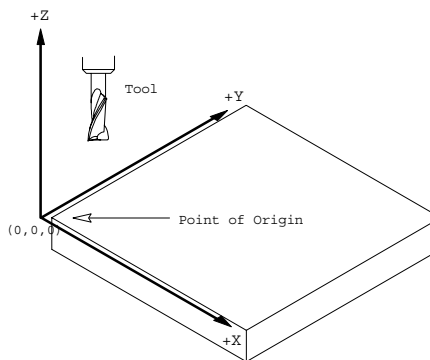
1). Reinstall Tool #1 in the spindle. Tool #1 is the reference tool, so we will be using this tool to set the workpiece origin.

2). Use the Power Feed keypad to jog the tool to the approximate 0,0,0 location on the workpiece. Once the tool 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 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 in correspondence 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 CRT. 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.)
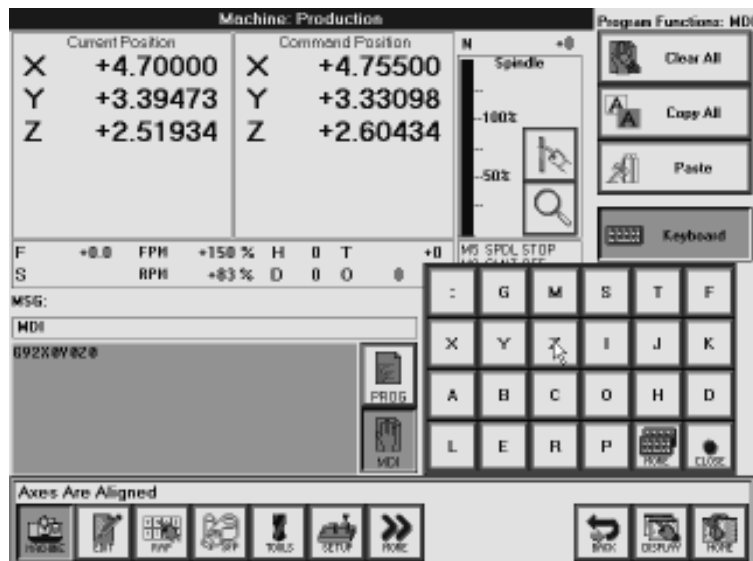
4). Using a combination of the keyboard and the numeric keypad, type G92X0Y0Z0. The NC code will appear in the NC code window as you type.

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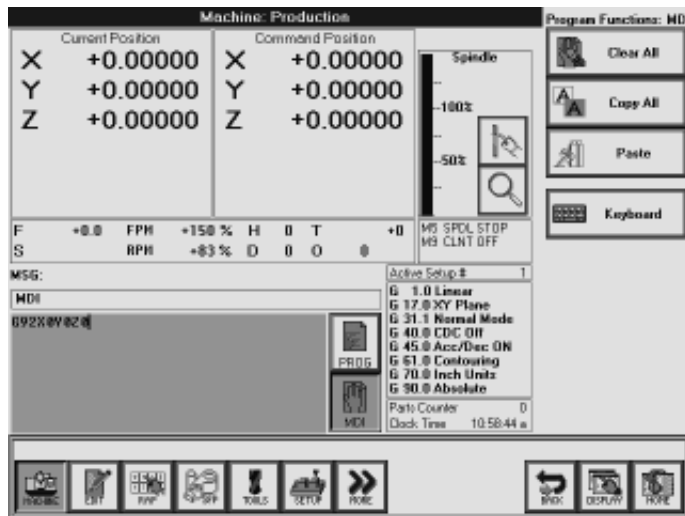
7). Return to MILLTWO.NC by pressing the PROG button above the MDI button. MILLTWO.NC is now returned as the active NC program.

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 in 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.

Section 4: Supported G+M Codes

Supported G+M Codes

The Acramatic 2100 Control supports the following G+M codes. Due to hardware considerations on the VMC-5000 Machining Centers, however, Light Machines does not support certain codes. For example, the VMC-5000 Machining Center uses an open loop control on the spindle motor; as a result those codes that require closed loop spindle control such as rigid tapping are not supported.

G Codes

Group	G Code	Usage	Comments
Interpolation	0 1 2 3	Rapid Traverse Linear Interpolation Circular/Helical CW Circular/Helical CCW	
Interpolation Hole Making Cycles	80 81 82 83 84 84.1 85 86 87 88 89	Reset Fixed Cycle Drill Cycle Counterbore with Dwell Deep Hole Drill Tap Cycle (Conventional Tapping) Tap Cycle (Rigid Tapping) Bore/Ream Cycle Bore Cycle, Dead Spindle Retract Back Bore Cycle Web Drill Bore/Ream with Dwell	 Not Supported Not Supported Not Supported Not Supported Do not use UV Offsets
Pattern Cycles	37 38 39	Cancel Pattern Rectangluar Pattern Circle Pattern	
Plane Selection	17 18 19	XY Plane XZ Plane YZ Plane	
Cutter Compensation	40 41 42	Cutter Compensation OFF Cutter Compensation LEFT Cutter Compensation RIGHT	
Acceleration/ Deceleration	45 46	Acceleration/Deceleration ON Acceleration/Deceleration OFF	
Local Coords.	52	Local Coordinate System	
Cornering	60 61	Positioning Mode Contouring Mode	

Group	G Code	Usage	Comments
Inch/Metric	70	Inch Programming	
	71	Metric Programming	
Abs/Inc	90	Absolute Programming	
	91	Incremental Programming	
Feedrate	93	Inverse Time Feedrate (1/T)	
	94	Feed Per Minute	
	95	Feed Per Tooth	
Spindle	97	Constant Spinde Speed (S=RPM)	
	97.1	Constant Spindle Speed (S=Surface Speed)	
Scaling	150	Scaling OFF	
	151	Scaling ON	
Non Modal	4	Dwell	
	12	Contouring Rotary Axis Unwind	
	28	Auto Return to Reference Point	
	29	Auto Return From Reference Point	
	36	Move to Next Operation Location	
	51	Vector Probe a Surface	Not Supported
	68	Tool Probe Cycle Set Tool Length	Not Supported
	69	Tool Probe Cycle Check Tool Length	Not Supported
	72	Set Stylus and Tip Dimension	Not Supported
	73	Set Probe Stylus Tip Dimensions	Not Supported
	74	Set Probe Length	Not Supported
	75	Locate Internal Corner	Not Supported
	76	Locate External Corner	Not Supported
	77	Locate Surface	Not Supported
	78	Locate and Measure Bore of Boss	Not Supported
	79	Measure Pocket or Web	Not Supported
	92,92.1	Position Set	
	99	Position Set Cancel	
Non Modal Modifiers	9	Exact Stop	
	50	Pallet Coordinates	
	98	Machine Coordinates (Tool Tip)	
	98.1	Machine Coordinates	

M Codes

Group	M Code	Usage	Comments
Mechanism Control	0	Program Stop	
	1	Optional Stop	
	2	End Of Program	
	3	Spindle ON CW	
	4	Spindle ON CCW	Not Supported
	5	Spindle Stop Coolant OFF	
	6	Tool Change	
	7	Coolant #2 ON	
	8	Coolant #1 ON	
	9	Coolant OFF	
	10	Clamp Axis	Not Supported
	11	Unclamp Axis	Not Supported
	13	Spindle ON CW Coolant ON	
	14	Spindle ON CCW Coolant ON	Not Supported
	19	Spindle Stop (oriented)	Not Supported
	26	Spindle Axis Full Retract	
	27	Coolant #3 ON	
	28	Coolant #4 ON	
	29	Coolant #5 ON	
	30	End Of Program	
	34	Enable Data Acquisition	
	35	Disable Data Acquisition	
	41	Spindle Constant Power Mode	
	42	Spindle Constant Torque Mode	
	48	Feed and Spindle Speed Override Enable	
	49	Feed and Spindle Speed Override Disable	
	58	Probe (Disarm Spindle Probe)	Optional
	59	Probe (Arm Spindle Probe)	Optional

Section 5: Optional Machining Capabilities

Quick Change Tooling Installation

Using the Tool Height Sensor

Multiple Tool Programming

Optional Tool Length Offset Method

Using the Low Voltage Light

Quick Change Tooling Installation

A Quick Change Tooling option is available for the VMC-5000 (ACC-5141). Quick Change Tooling is not available for the VMC-5500. The installation procedure is described below.

Installing the Quick Change Tooling Option on the VMC-5000 Machining Center is a three-step process:

1. Installing the tool body in the machine spindle

2. Attaching the cutting tool to the tool holder

3. Mounting the tool holder to the tool body

Installing and Removing the Tool Body

Refer to the drawing on the following page for the location of the parts referenced in these instructions.

Note: Before using Quick Change Tooling, clean the taper of the tool holder and the bore of the tool body.

Clean and coat the surface of the tool holder with oil once a week.

To install the tool body into the spindle:

1. Insert the draw bolt and washer (from the Machining Center Accessory Kit) into the top opening of the spindle shaft.

2. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.

3. Insert the tool body into the bottom opening of the spindle until it makes contact with the spindle taper.

4. Screw the draw bolt into the tool body until secure.

5. Remove the spindle locking pin.

To remove the tool body from the spindle:

1. Insert the spindle locking pin into the opening on the side of the spindle.

2. Loosen the draw bolt approximately two turns.

3. Use a hammer (preferably brass) to hit the top of the draw bolt to release the tool body from the spindle taper.

4. While holding the tool body in place, loosen the draw bolt completely and allow the tool body to drop into your hand.

5. Remove the spindle locking pin.

Draw Bolt

Spindle

Quick Change Tool Body

spindle taper

Collar

Tool Holder

set screw

Cutter
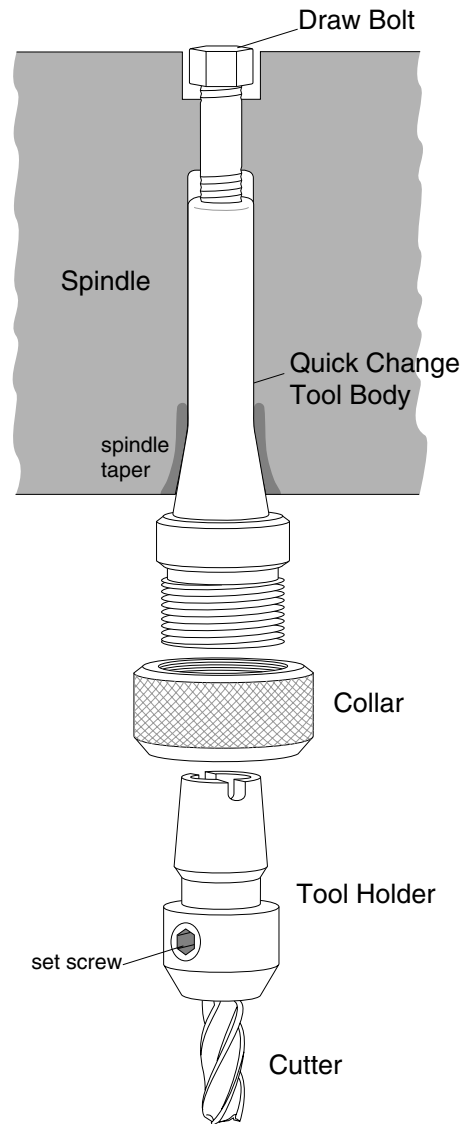
Attaching the Cutting Tool to the Tool Holder

1. Insert the tool into the bottom opening of the tool holder.

2. Tighten the set screw on the side of the tool holder while holding the tool in place. Make sure the set screw is pressing against the flat of the tool.

Installing and Removing the Tool Holder

To mount the tool holder to the tool body:

1. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.

2. Screw the collar all the way onto the tool body.

3. Now unscrew the collar approximately 1-1/2 turns.

4. Press the tool holder up into the tool body until it seats into place (This occurs when the two indents on the top of the tool holder align with the locking pins inside the tool body.)

5. Hand-tighten the collar. Do not overtighten.

6. Remove the spindle locking pin.

To remove the tool holder from the tool body:

1. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.

2. Hold the tool holder, then unscrew the collar approximately 1-1/2 turns. Remove the tool holder.

3. Remove the spindle locking pin.

Using the Tool Height Sensor

A tool height offset sensor is included with every Quick Change Tooling unit. This sensor helps to establish a constant reference point for setting tool offsets for multiple tools.

The battery powered sensor uses the machine's frame to provide an electrical circuit. When a tool contacts the sensor the circuit closes, energizing the sensor's LED indicators. You must place the sensor on a conductive surface, such as the cross slide, for this to take place.

See Multiple Tool Setup in Section3 for information on establishing reference points and offsets with the Quick Change Tooling unit.
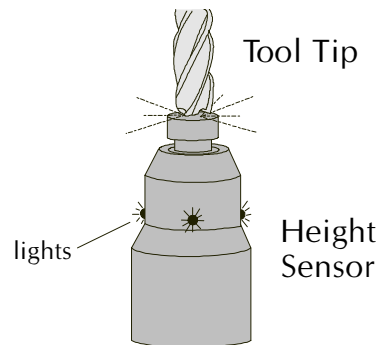
Tool Tip

lights

Height Sensor

Multiple Tool Programming

There is a sample multiple tool NC program on the Samples disk. Section 3, Quick Start, provides tutorials for both single and multiple tool programs. The procedures for MILLTWO.NC are the same as MILLONE.NC, with the addition of the following sections.

Multiple tool programming is not available for the proLIGHT 5500.

Programming Tool Changes

When programming tool changes for multiple tool programs, use M06 and a T code to specify the tool change, then M03 and an S code to turn the spindle on. Repeat this procedure for each tool change.

For example:

M06T01; Specifies tool change to Tool #1

M03S1500; Turns the spindle on with a speed of 1500

M06T02; Specifies tool change to Tool #2

M03S1500; Turns the spindle on with a speed of 1500

M06T03; Specifies tool change to Tool #3

M03S1500; Turns the spindle on with a speed of 1500

When you run your program, the machine stops at your first programmed tool change. A message appears on the control screen, telling you that the machine has stopped for an operator requested tool change.

1. Open the door to the machine.

2. Change the tool.

3. Close the door to the machine.

4. Clear the message on the control screen by pressing the CLEAR button.

5. Press the Cycle Start button on the pendant to start the machine.

6. Repeat this procedure for each tool change.

Optional Tool Length Offset Method

Section 3 details one method of setting the tool length offset. There are other methods you may use. This method sets the offset for each tool based on a Tool Gauge Plug used as a reference point.

Use a non-cutting reference plug installed in the spindle to set the tool length offsets for all the tools.

1. Install a reference plug into the spindle.

2. Use the pendant to jog the reference plug to a reference point (such as the top of the stock) on the Z axis. A height offset sensor also works well.

3. Use MDI to set the Z position to zero at the reference point. To do this, run program G92Z0 when the reference plug is at the reference point.

4. Install Tool #1 in the spindle. Jog Tool #1 to the same Z axis position.

    The Z axis readout on the pendant (or on the Home display on screen) now has a number other than zero. This is the tool length of Tool #1 (relative to the reference plug).

5. Enter this Z axis value in the length field for Tool #1.

6. Repeat steps 4 and 5 for additional tools.

7. Install Tool #1 in the spindle.

8. Mount your workpiece on the cross slide.

9. Position Tool #1 to the Z axis reference on the workpiece (usually Z=0 is the top of the workpiece.

10. Run the MDI program to set the Z axis position (G92Z0).

11. Run your NC program.

Using the Low Voltage Light

You can purchase the VMC-5000 Machining Center with an optional low voltage light (ACC-5720). This factory installed option is capable of illuminating the entire work surface within the machine's safety enclosure.

Positioning the Light

The light is attached to a gooseneck mounted on the inside back panel of the machine. Position the light by bending the gooseneck and moving the light.

When positioning the light, remember to:

❑ Keep the light away from cutting fluids or anywhere cutting fluids may splatter.

❑ Keep the light away from moving machinery, including the machine's spindle head, 4th axis rotary positioner, and automatic tool changer.

Turning On the Light

You can turn the light on and off in two ways:

❑ using M codes

❑ pressing the Ctrl and More keys on the pendant

Using M codes

Use the M codes to turn the light on and off during an NC program. Insert these codes in your NC program when you want to turn the light on and off:

M100   Turns the light ON

M101   Turns the light OFF

Pressing the Ctrl and More keys

Press the Ctrl and More keys to turn the light on and off. Pressing these keys toggles the light on and off. Pressing the Ctrl and More keys overrides any M codes you have programmed for turning the light on and off.

Changing the Light Bulb

The light uses a 12 volt, 20 watt Halogen light bulb. To change the light bulb:

1. Turn off power to the light, then disconnect the VMC-5000 from all power sources. Let the light cool off before you remove the bulb.

2. Unscrew the front of the lamp housing, which contains the lens and reflector.

3. Pull the bulb straight out of the socket.

CAUTION:

Do not touch a new bulb with your bare hand. Impurity on a Quartz Halogen bulb can adversely affect the life of the bulb.

4. Using a plastic wrapper or tissue as protection for the new bulb, firmly push the replacement bulb straight into the socket until it bottoms out.

5. Screw the front of the lamp housing back in place.

WARNING:

Disconnect the machining center from all electrical power sources before changing the light bulb. Failure to do so may result in serious injury or death.

WARNING:

Do not touch the light bulb immediately after using the light. Touching the light bulb may result in serious personal injury.

Section 6: VMC-5000 Machining Center Maintenance

Maintaining the VMC-5000

Maintaining the VMC-5000 Machine

Maintaining the VMC-5500 Machine

Maintaining the Operator Station

Maintaining the 4th Axis Rotary Positioner

Maintaining the VMC-5000

Only qualified, designated operators or maintenance personnel should operate or perform maintenance or repair work on the VMC-5000 Machining Center. Maintenance personnel should only begin such work after reading all safety, general, and maintenance instructions.

The instructions in this section are intended for qualified personnel only. If you are unsure of proper maintenance procedures, do not attempt to perform maintenance on the machining center or its components.

Performing preventative maintenance on your VMC-5000 Machining Center ensures a longer, trouble-free life for the machine.

This section describes the maintenance you need to perform on various parts of the machine, as well as describing the proper maintenance for the operator station and the pendant.

Follow these procedures to maintain the various parts of the machine.
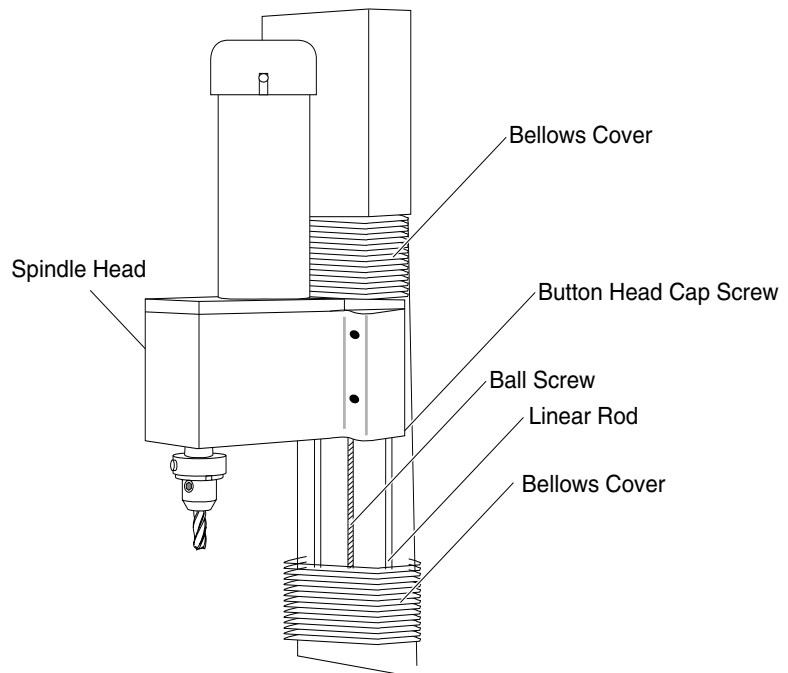
Ball Screws

The VMC-5000 machine uses pre-loaded ball screws on all three axes. The screws are factory-lubricated with a long-life, waterproof, multi-purpose Lithium base ball screw lubricant. This lubricant lasts for 100 hours of machine use. After 100 hours, apply more lubricant over the entire length of the screw.

To gain access to the ball screws:

1. Jog the cross slide and spindle to the extreme negative end of travel on all axes.

2. Remove the bellows cover by removing the two button head cap screws from the bracket holding the bellows at one end.

Use a multi-purpose Lithium base lubricating grease to lubricate the ball screws.

3. Use a small brush to apply the lubricant evenly along the entire length of the ball screw.

In this illustration, the Z axis bellows cover is removed to expose the Z axis linear rods and ball screw.

Bellows Cover

Spindle Head

Button Head Cap Screw

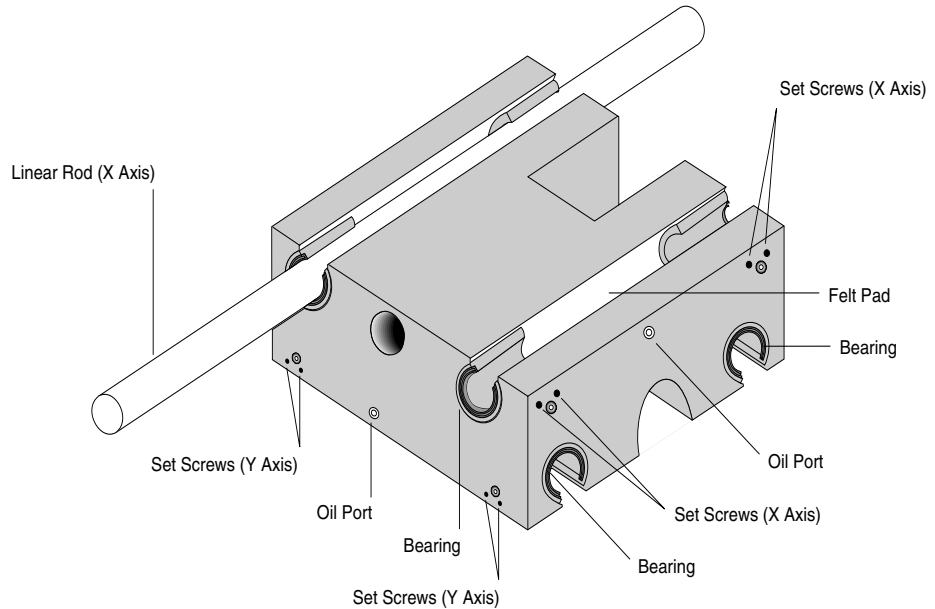Ball Screw

Linear Rod

Bellows Cover

The Saddle

The saddle engages the linear rods attached to the base of the machining center. A ball screw moves the saddle along the Y axis. The linear rods running through the top of the saddle engage the cross slide. A ball screw moves the cross slide along the X axis. The oil ports for the X axis and Y axis linear rods are located on the saddle.

The Machining Center's Saddle.

Set Screws (X Axis)

Linear Rod (X Axis)

Felt Pad

Bearing

Set Screws (Y Axis)

Oil Port

Oil Port

Set Screws (X Axis)

Bearing

Bearing

Set Screws (Y Axis)

Linear Slides

The linear slides on the VMC-5000 consist of linear rods and linear bearings. It is *very important* that you maintain a thin film of lubricant on the surface of the linear rods to minimize wear. The linear rods wear very quickly with no lubrication.

Lubrication

Properties of the required lubricant:

Viscosity Centistoke		Density g/cc at
100°F (38°C)	210°F (99°C)	60°F (16°C)
41	6.0	0.870

Light Machines provides a small oil can with the machining center. Use SAE 10W engine oil (see properties at right) for lubricating the linear rods.

There are thick oil-impregnated felt pads between the bearings in the saddle that keep a thin film of oil on the linear rods. Lubricate each rod (there are six) through its own oil port. The oil ports are located on the saddle for the X and Y axes, and on the spindle head for the Z axis.

You should lubricate the rods every 30 days or 100 hours of use, whichever comes first. When applying oil to the oil ports, pull the trigger on the oil gun slowly to allow the felt pads to soak up the oil.

A side view of the Saddle.

Adjustment

You can adjust the linear bearings to remove any play in the saddle or spindle. The bearings are factory-adjusted and should not require adjustment for at least 1000 hours of use. Be very careful not to over-tighten the bushings because overtightening can cause overworking and overheating of the motor, and excessive wear to the rods, bushings and ball screws. To adjust the linear bushings, follow the steps below.

1. Jog to the extreme positive end of motion on the axis you are adjusting. Shut off power to the system. Unplug the machine.

2. Remove part of the servo motor cover for the axis you are adjusting. Loosen the screws but do not remove the portion of the cover that has the wires coming out of it. Do not remove the whole cover at once or you may damage the wiring.

3. Loosen all eight recessed allen setscrews on the axis you are adjusting. They are located on the four sides of the saddle for the X and Y axes, and on the sides of the spindle for the Z axis.

Set Screws push on the bearing when tightened → | Bearing

4. Tighten the two setscrews for each bearing while checking the force it takes to turn the ball screw. Tighten the setscrews (applying approximately 4-8 in/lb of torque) until the force required to turn the ball screw pulley increases when turning it by hand. Then back the setscrews off by 1/8-turn. Repeat this procedure for all four bushings on each axis.

   If you need to use excessive force to turn the lead screw, you have adjusted the bearing too tightly.

5. Replace the servo motor cover.

Spindle

Two Piece Spindle Head

Light Machines ships the VMC-5000 Machining Center with a factory-aligned two piece spindle head. You should not attempt to align the spindle head without first contacting Light Machines.

Spindle Motor

The spindle motor on the VMC-5000 Machining Center is a brushless DC motor. The ball bearings on the motor shaft are the only wearing parts on the motor. The ball bearings are sealed, life-time-lubricated bearings that do not require special maintenance.
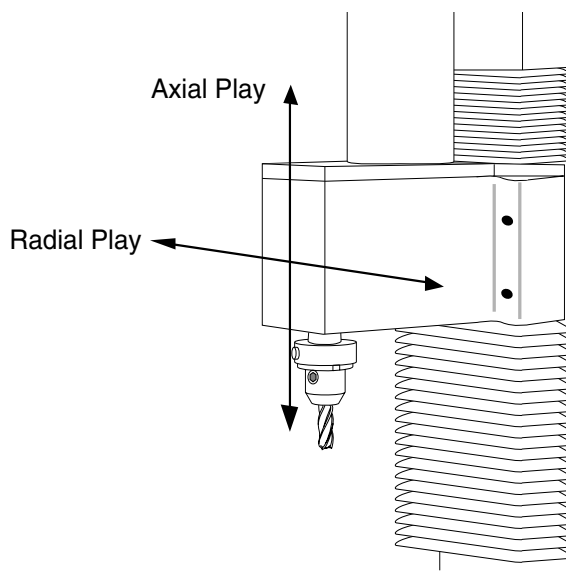
Checking for Spindle Shaft Play

The spindle shaft is pre-loaded against sealed ball bearings that do not require lubrication or user maintenance. You should, however, check the spindle shaft for both radial play (side to side) and axial play (up and down). If the spindle shaft starts to develop play (or begins to make unusual noises while in operation), contact Light Machines (800-221-2763).

To check for play, grip the spindle shaft and push and pull it in each direction along both axes. The spindle shaft should be firm against your pressure.
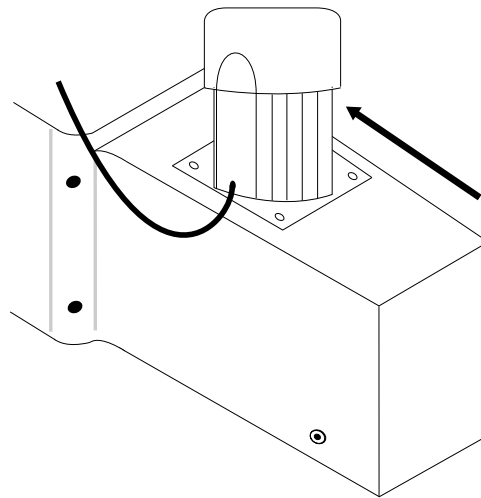
Belts

Spindle Belt

The spindle motor drives the spindle shaft with a serpentine belt. If the belt becomes loose, it wears out quickly and slips during moderate spindle loads.

To adjust the spindle serpentine belt:

1. Make sure all power to the system is shut off. Unplug the machine.

2. Loosen the four motor bolts one revolution.

3. Using your hand, push the motor towards the back of the machine until you have taken up any slack in the belt.

4. While still holding the motor in place towards the back of the machine, tighten the four motor bolts.

5. Turn the spindle shaft to make sure the spindle runs freely by hand and the spindle motor turns. If everything appears all right, try turning on the spindle motor and slowly turning up the spindle speed. Check for excessive vibration.

WARNING:

Always disconnect power to the machining center before performing maintenance on the operator station. Failure to do so follow may result in serious personal injury.

Push the spindle motor towards the back of the machine until you have taken up any slack in the belt.

Checking and Adjusting the Y Axis Drive Belt

The axis drive belts are located between the servo motors and ball screws on each axis. Normally, they should not need adjustment, but you should check every 250 hours.

1. Remove the 5 button head cap screws on the motor box cover and the two on the top of the motor box.

2. Slide the motor box cover off.

Remove the two
top screws.

Pull the cover towards you
to expose the drive belt.

Remove the five
bottom screws.
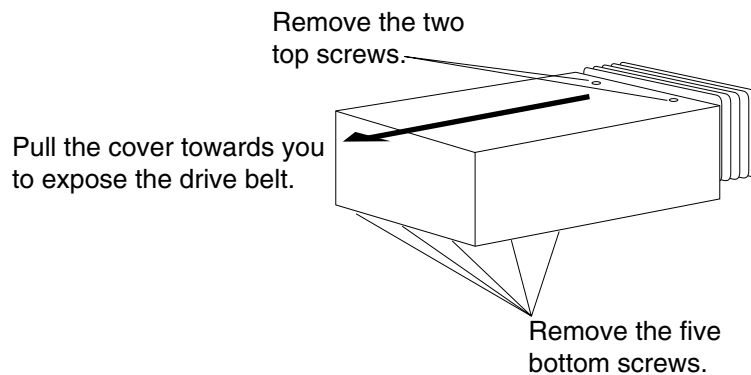
3. Apply approximately 3 pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).

CAUTION:

If the belt needs adjustment:

Do not overtighten the drive belt. Excessive tension will damage the motor shaft.

1. Loosen the four screws holding the servo motor onto the motor mount..

2. Slide the motor to increase tension on the belt and retighten the screws.

3. Check the deflection again.

Checking and Adjusting the X Axis Drive Belt

1. Remove the two button head cap screws from the top of the motor box and the seven button head cap screws from the bottom of the motor box cover.

2. Pull the motor box back approximately 1 inch until it comes off the motor mount. Lift the left side of the motor box up and over the motor to expose the drive belt.

Remove the screws.

Pull back then lift left side up.

3. Apply approximately 3 pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).

If the belt needs adjustment:
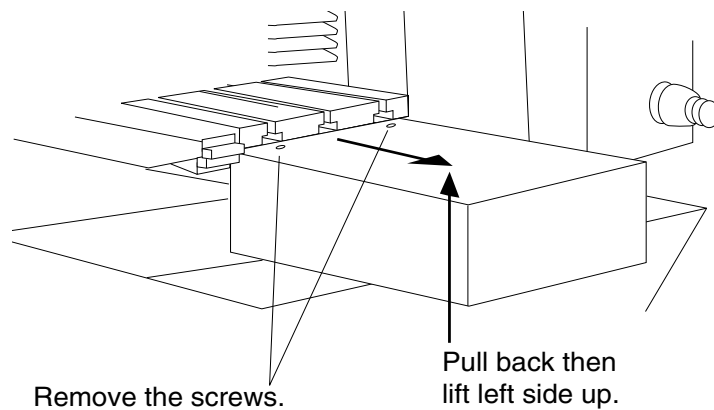
CAUTION:
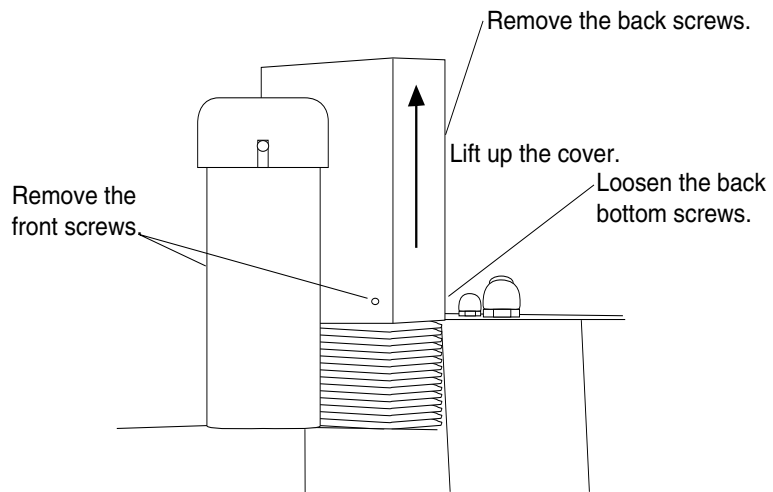
Do not overtighten the drive belt. Excessive tension will damage the motor shaft.

1. Loosen the four screws holding the servo motor onto the motor mount.

2. Slide the motor to increase tension on the belt and retighten the screws.

3. Check the deflection again.

Checking and Adjusting the Z Axis Drive Belt

1. Remove the two button head cap screws on the front of the motor box and five of the seven button head cap screws on the back. Loosen but leave in place the two bottom screws on the back of the motor box cover.

2. Gently move the box front and back while pulling it up until it is completely off of the motor mount.

Remove the back screws.

Lift up the cover.

Loosen the back bottom screws.

Remove the front screws.

If the belt needs adjustment:

1. Loosen the four screws holding the servo motor onto the motor mount.

2. Slide the motor to increase tension on the belt and retighten the screws.

3. Check the deflection again.

Exterior Surfaces

To clean the exterior surfaces of the machine, including the safety shields:

❑ Use a damp, soft, lint-free cloth.

❑ Do not use aerosol sprays, solvents, or abrasives.

❑ To clean the safety shields, put household glass cleaner on a soft cloth and wipe until clean.

Maintaining the VMC-5500

Performing preventative maintenance on your VMC-5500 Machining Center ensures a longer, trouble-free life for the machine. We provide instructions for preventative maintenance in the following paragraphs.

Ball Screws

The VMC-5000 machine uses pre-loaded ball screws on all three axes. The screws are factory-lubricated with a long-life, waterproof, multi-purpose Lithium base ball screw lubricant. This lubricant lasts for 100 hours of machine use. After 100 hours, apply more lubricant over the entire length of the screw.

Note: If you run your machine continuously (8 hours per day) at high feed rates, you should lubricate your machine every 16 hours of use.

To gain access to the ball screws:

1. Jog the cross slide and spindle to the extreme negative end of travel on all axes.

2. Remove the bellows cover by removing the two button head cap screws from the bracket holding the bellows at one end.

Use a multi-purpose Lithium base lubricating grease to lubricate the ball screws.

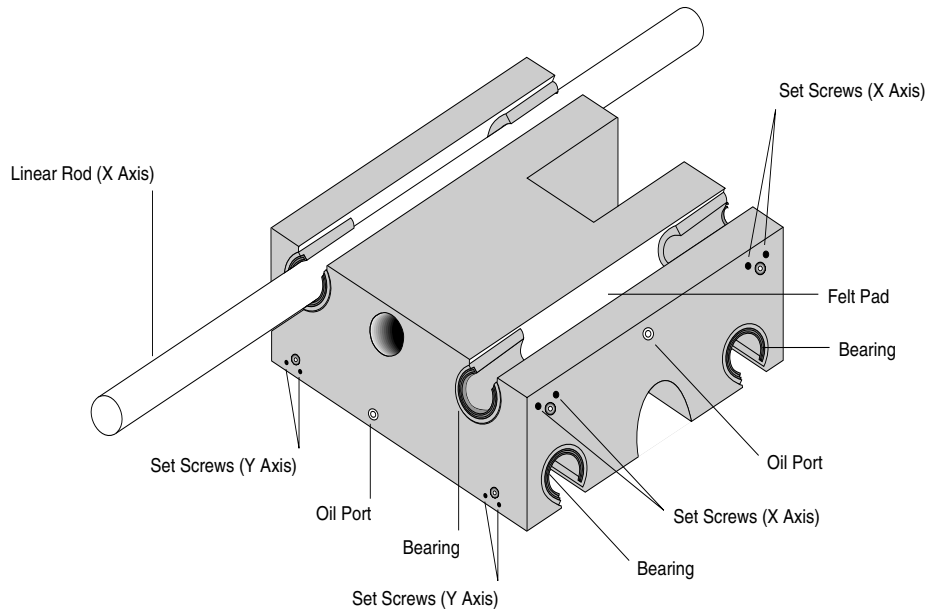3. Use a small brush to apply the lubricant evenly along the entire length of the ball screw.

Here is the VMC-5500 with the Z axis bellows cover removed to expose the Z axis linear rods and ball screw.
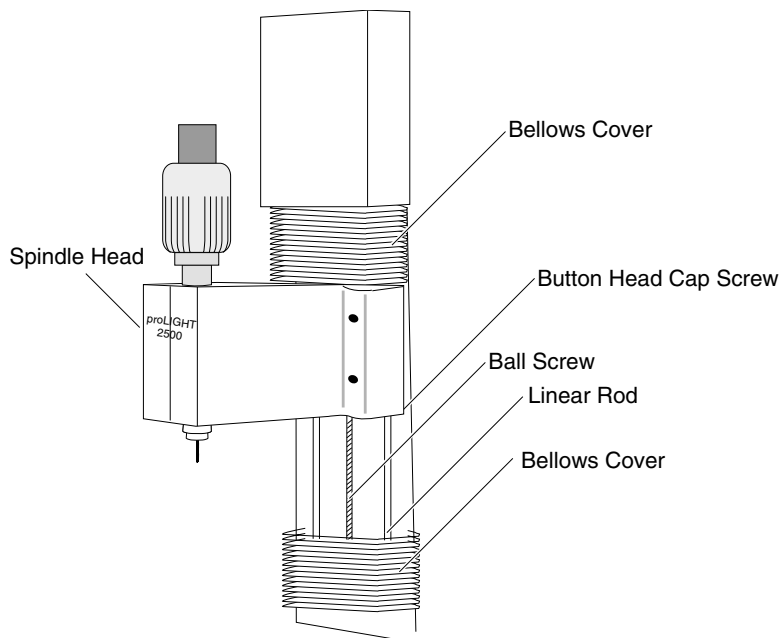
The Saddle

The saddle engages the linear rods attached to the base of the machining center. A ball screw moves the saddle along the Y axis. The linear rods running through the top of the saddle engage the cross slide. A ball screw moves the cross slide along the X axis. The oil ports for the X axis and Y axis linear rods are located on the saddle.

The Machining Center's Saddle.

Linear Slides

The linear slides on the VMC-5000 consist of linear rods and linear bearings. It is *very important* that you maintain a thin film of lubricant on the surface of the linear rods to minimize wear. The linear rods wear very quickly with no lubrication.

Lubrication

Properties of the required lubricant:

Viscosity Centistoke		Density g/cc at
100°F (38°C)	210°F (99°C)	60°F (16°C)
41	6.0	0.870

Light Machines provides a small oil can with the machining center. Use SAE 10W engine oil for lubricating the linear rods.

There are thick oil-impregnated felt pads between the bearings in the saddle that keep a thin film of oil on the linear rods. Lubricate each rod (there are six) through its own oil port. The oil ports are located on the saddle for the X and Y axes, and on the spindle head for the Z axis.

You should lubricate the rods every 30 days or 100 hours of use, whichever comes first. When applying oil to the oil ports, pull the trigger on the oil gun very slowly to allow the felt pads to soak up the oil.

A side view of the Saddle.

Adjustment

You can adjust the linear bearings to remove any play in the saddle or spindle. The bearings are factory-adjusted and should not require adjustment for at least 1000 hours of use. Be very careful not to over-tighten the bushings because overtightening can cause overworking and overheating of the motor, and excessive wear to the rods, bushings and ball screws. To adjust the linear bushings, follow the steps below.

1. Jog to the extreme positive end of motion on the axis you are adjusting. Stop just before the limit switch trips. Shut off power to the system. Unplug the machine.

2. Remove part of the servo motor cover for the axis you are adjusting. Loosen the screws but do not remove the portion of the cover that has the wires coming out of it. Do not remove the whole cover at once or you may damage the wiring.

3. Loosen all eight recessed allen setscrews on the axis you are adjusting. They are located on the four sides of the saddle for the X and Y axes, and on the sides of the spindle for the Z axis.

4. Tighten the two setscrews for each bearing while checking the force it takes to turn the ball screw. Tighten the setscrews (applying approximately 4-8 in/lb of torque) until the force required to turn the ball screw pulley increases when turning it by hand. Then back the setscrews off by 1/8-turn. Repeat this procedure for all four bushings on each axis.

    If you need to use excessive force to turn the lead screw, you have adjusted the bearing too tightly.

5. Replace the servo motor cover.

Spindle

Two Piece Spindle Head

Light Machines ships the VMC-5500 Machining Center with a factory-aligned two piece spindle head. You should not attempt to align the spindle head without first contacting Light Machines.
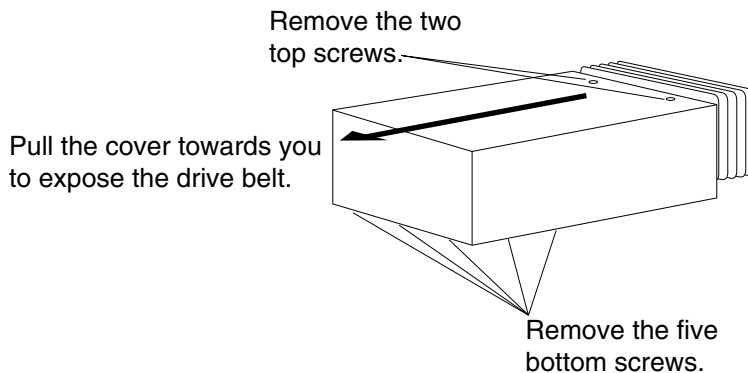
Spindle Assembly

The VMC-5500 spindle components have been specially lubricated for maintenance free operation. Never attempt to lubricate or otherwise service the VMC-5500 spindle assembly. Only qualified specialists should service your machine's spindle assembly.

Axis Drive Belts

The axis drive belts are located between the servo motors and ball screws on each axis. Normally, they should not need adjustment, but should be checked every 250 hours.

Checking and Adjusting the Y Axis Drive Belt

1. Remove the 5 button head cap screws on the motor box cover and the two on the top of the motor box.

2. Slide the motor box cover off.

Remove the two top screws.

Pull the cover towards you to expose the drive belt.

Remove the five bottom screws.

3. Apply approximately 3 pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).
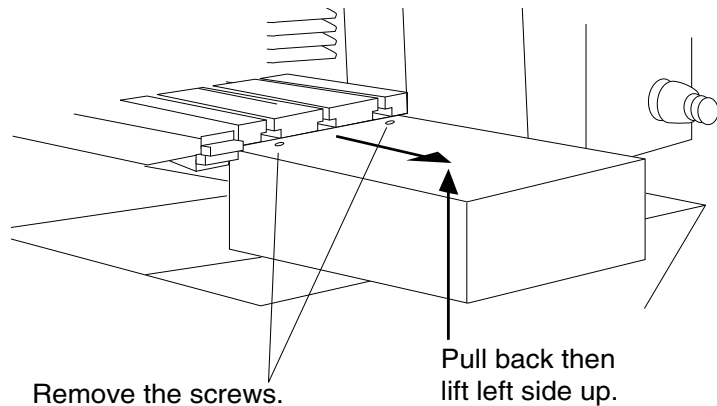
If the belt needs adjustment:

1. Loosen the four screws holding the servo motor.

2. Slide the motor to increase tension on the belt.

3. Tighten the screws and check the deflection again.

Checking and Adjusting the X Axis Drive Belt

1. Remove the two button head cap screws from the top of the motor box and the seven button head cap screws from the bottom of the motor box cover.

2. Pull the motor box back approximately 1 inch until it comes off the motor mount. Lift the left side of the motor box up and over the motor to expose the drive belt.

Remove the screws.
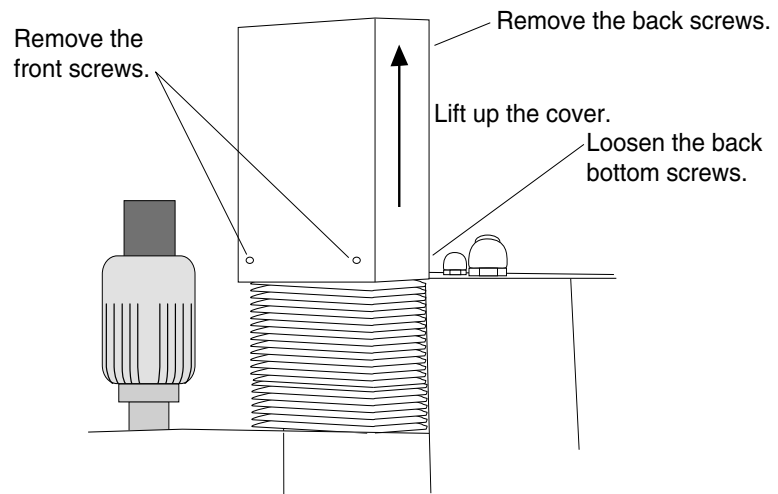
Pull back then lift left side up.

3. Apply approximately 3 pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).

If the belt needs adjustment:

1. Loosen the four screws holding the servo motor onto the motor mount.

2. Slide the motor to increase tension on the belt and retighten the screws.

3. Check the deflection again.

Checking and Adjusting the Z Axis Drive Belt

1. Remove the two button head cap screws on the front of the motor box and five of the seven button head cap screws on the back. Loosen but leave in place the two bottom screws on the back of the motor box cover.

2. Gently move the box front and back while pulling it up until it is completely off of the motor mount.

If the belt needs adjustment:

1. Loosen the four screws holding the servo motor onto the motor mount.

2. Slide the motor to increase tension on the belt and retighten the screws.

3. Check the deflection again.

Exterior Surfaces

To clean the exterior surfaces of the machine, including the safety shields:

❑ Use a damp, soft, lint-free cloth.

❑ Do not use aerosol sprays, solvents, or abrasives.

❑ To clean the safety shields, put household glass cleaner on a soft cloth and wipe until clean.

VMC-5000 Machining Center Routine Maintenance Schedule

Maintenance Period

Action	After Every Use	After 100 Hours	After 250 Hours	After 1000 Hours
Clean Chips Out of Machine	✓			
Coat Exposed Surfaces with Light Oil	✓			
Lubricate Ball Screws		✓		
Lubricate Linear Rod Pads		✓		
Check Spindle Belt for Play			✓	
Adjust Linear Bushings				✓
Check Spindle Shaft for Play				✓
Check Limit Switches and Wiring				✓

Maintaining the Operator Station

There are a few guidelines for maintaining your operator station in a shop environment.

General Guidelines for Operator Station Care

Follow these general rules for operator station care.

❑ Keep the operator station out of direct sunlight, away from sources of heat, and in a relatively clean environment.

❑ Keep liquids away from the operator station.

❑ Keep oil, grease, metal chips and excess dust away from the operator station.

❑ Don't block the fan or air filter on the operator station; they are required for air circulation.

Cleaning the Exterior of the Operator Station

Keep the exterior of your operator station clean and dust free. To clean the exterior surfaces of the operator station, including the touchscreen:

❑ Use a damp, soft, lint-free cloth.

❑ Do not use aerosol sprays, solvents, or abrasives.

❑ To clean the touchscreen, put household glass cleaner on a soft cloth and wipe until clean. Do not spray the cleaner directly on the screen, because the cleaner may drip into the operator station.

Cleaning the Air Filter

You need to check the air filter on the operator station every 100 hours of operation. The air filter is located on the bottom panel of the operator station. To clean the air filter:

1. Disconnect power to the machining center.

2. Carefully lay the operator station on its right side (when looking at it from the front). Be sure to use proper lifting techniques when moving the operator station.
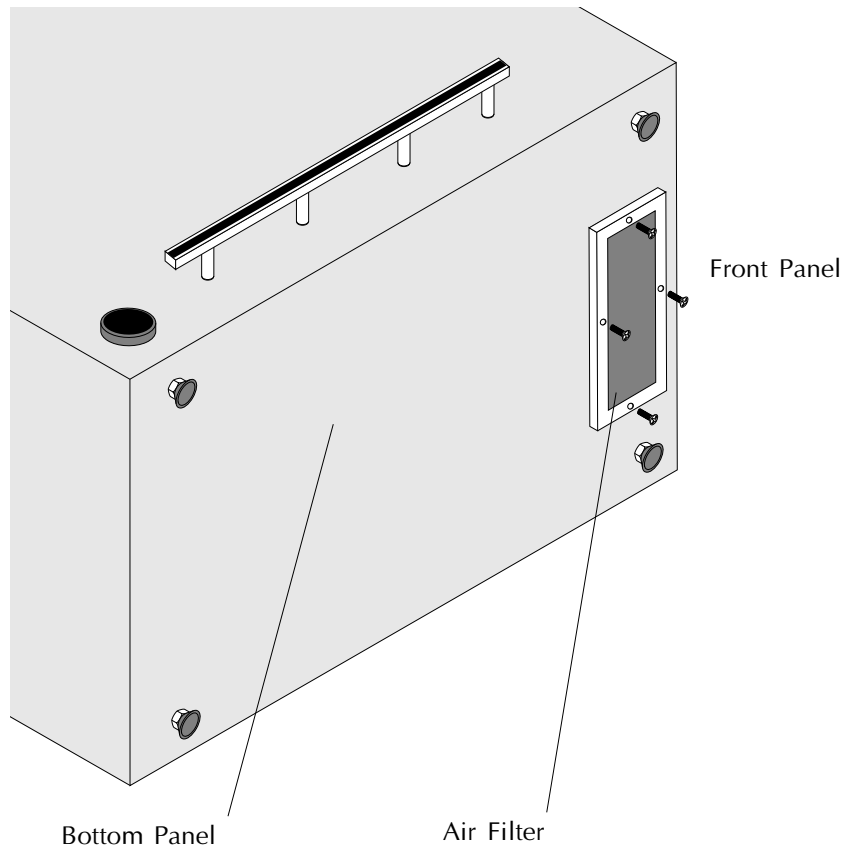
3. Remove the four phillips head screws from the air filter.

The Operator Station on its side. Notice the air filter on the bottom panel, towards the front of the operator station.

Front Panel

Bottom Panel

Air Filter

4. Remove the filter and wash it under running water to remove any foreign particles.

5. Let the air filter dry, or air dry it carefully. Be sure not to remove any filter material from the filter when drying.

6. Remount the air filter on the operator station using the four phillips head screws.

7. Lift the operator station to its normal upright position.

8. Reconnect power to the machining center.

Maintaining the Rotary Positioner

If you purchased the 4th Axis Rotary Positioner option for your VMC-5000 Machining Center, you must keep it properly lubricated. This will extend its life and ensure maximum operational performance.

Lubricating the Rotary Positioner

Oil level and cleanliness should be checked regularly. Change the oil once or twice a year. If there is any evidence of contamination, the gear box should be flushed with clean solvent, the seals checked and replaced if required, and more frequent oil changes considered.

Characteristics of the recommended oils:
Agma Grade: 4-EP
Flash Point: 400°F
ISO Grade: 100-150
SUS Viscosity: 725 @ 100°F
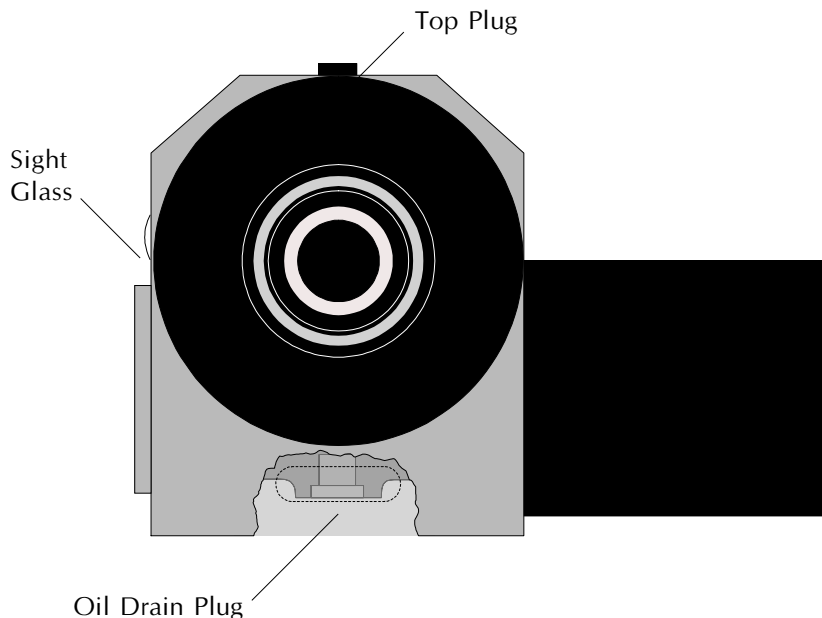   75.5 @ 210°F

The lubricant must be a premium quality heavy-duty industrial gear oil for enclosed gear sets. The oil must provide good rust and corrosion protection, oxidation stability, foaming resistance, and pressure characteristics which minimize temperature rise.

Since some oil could be lost through the seals, the oil level should be checked weekly or more frequently should you find it necessary to add much oil at the weekly checks.

Oil is added through the top of the unit after removing the top plug. As oil is added, check the oil level through the sight glass on the side of the unit. When draining oil, loosen the top plug to vent. Replace the drain plug before adding oil.

Top Plug

Sight
Glass

Oil Drain Plug

Section 7: Dismantling and Storing

Getting Ready

Disconnecting the Machining Center

Packing the Machining Center

Storing the Machining Center

Getting Ready

Follow the instructions in this section to correctly dismantle, transport, and store the VMC-5000 Machining Center. Failure to follow the instructions in this section may cause damage to your machine.

Before you begin dismantling your machining center, be sure you have all of the following:

❑ The original packaging materials for the machining center.

❑ An allen wrench for removing the attachment panel from the operator station.

❑ A phillips head screwdriver for removing the X axis servo extension box.

❑ A lifting device for moving the machining center.

Contact Light Machines Corporation if you do not have the correct packaging materials. Do NOT transport the machining center without the correct packaging materials. Light Machines Corporation is not responsible for any damage caused during shipping when components are not shipped in their original containers.

Disconnecting the Machining Center

The following paragraphs explain the procedures for disconnecting the operator station with the machine.

Preparing the Machine

Before disconnecting power to the machine, be sure to properly position the spindle head and X and Y axes:

1. Remove all tooling from the machine.

2. Bring the machine's spindle head all the way down on the Z axis.

3. Bring the Y axis all the way to the back of the machine.

4. Center the X axis on the machine.

Disconnecting Power to the Components

Unplug the machine's power cord from the wall outlet. This removes power from all components of the machining center.

Remember to always disconnect power to the machining center before performing maintenance.

WARNING:

Disconnect electricity to the machining center before disconnecting machining center components. Failure to disconnect electricity may cause serious personal injury.

Disconnecting the Operator Station and the Machine

When you installed the machining center, you attached the connectors from the interface cable to the back of the operator station. Now you need to remove the attachment panel from the operator station and disconnect the connectors.

1. Remove the eight screws that secure the attachment panel to the back of the operator station.

The attachment panel on the back of the operator station.
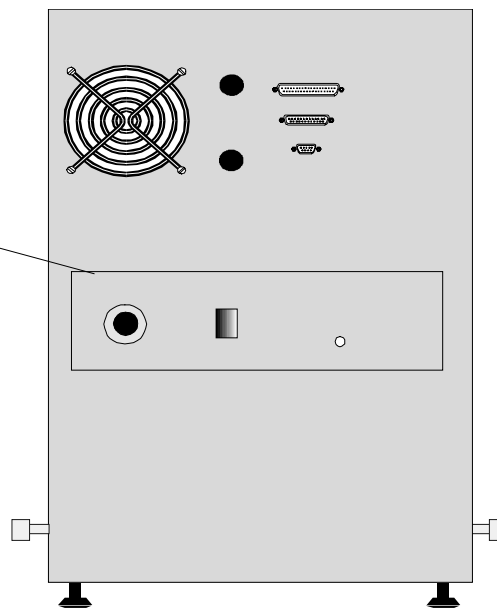
2. Remove the attachment panel, but be sure to hold the attachment panel close to the connector panel. You need to disconnect the connectors before you can place the attachment panel down.

3. Disconnect the all of the connectors from the connector panel. Unscrew the connector screws on all of the connectors except the J1 connector. For the J1 connector, push the two J1 connector tabs outward simultaneously. This releases the J1 connector from the receptacle.

4. Carefully place the attachment panel down. Be sure not to place the attachment panel on any of the cables or the connectors.

Packing the Machining Center

The VMC-5000 Machining Center originally came to you carefully packed in three shipping cartons. You should use the same materials when you pack the machining center for transport.

General Packing Guidelines

❑ Use only correct materials to package the machining center. Be sure the packaging materials are in good condition.

❑ Always use proper lifting techniques when moving the machining center.

❑ Never place anything weighing over 100 lbs. on the top of the machining center.

Lifting the Machine onto the Pallet

After you have disconnected the components of the machining center, you can move the machine to the pallet in which it was originally shipped. Use the machine's lifting ring to lift the machine. Contact supervision if you are unsure about the proper procedures for lifting the machine.

1. Lift the machine off of the table using the machine's lifting ring.

   When you lift the machine, it tilts forward $20°$.

2. Guide the machine to the pallet. The feet of the machine should fit into the four holes on the pallet.

3. Insert the four bolts that hold the machine base to the pallet from the underside of the pallet. Tighten the bolts securely.

Packing the Machine

Now pack the machine on the pallet.

1. Place the cardboard container over the machine on the pallet.

2. Staple the cardboard container to the pallet.

Packing the Operator Station

Now pack the operator station.

1. Position the container for the operator station near the table on which you placed the operator station.

2. Now place the operator station in the container. The operator station weighs 130 lbs (59 kg), so you need two people to lift it. Contact supervision if you are unsure about the proper procedures for lifting and carrying.

   With two people, lift the operator station using the handles on both sides. Place the operator station in the container.

3. Pack the pendant in the pendant box.

4. Strap the carton to a pallet for shipment.

Storing the Machining Center

Follow these guidelines when storing the machining center for longer than one month:

❑ Be sure to disconnect all power to the machining center.

❑ Coat all unpainted metal surfaces of the machining center with a rust-preventative. Such surfaces include the spindle, cross slide, rods of the machining center, and any tool holders or other fixtures you may be shipping.

❑ Properly package the machining center in the original packaging materials.

❑ Store the machining center in a dry location with a moderate temperature. Keep away from direct sunlight and excessive dust.

Section 8: Working with spectraCAM

Starting spectraCAM

Using a Mouse with spectraCAM

Starting spectraCAM

For more information on spectraCAM, refer to the spectraCAM Milling User's Guide (34-6741-0000) for more information.

For more information on spectraCAD, refer to the spectraCAD User's Guide (34-7791-0000).

The VMC-5000 Machining Center includes copies of spectraCAM and spectraCAD, which you can use to create part drawings in CAD, then create machine tool paths in CAM. Using spectraCAM, spectraCAD, and the VMC-5000, you can design and manufacture parts all from one location.
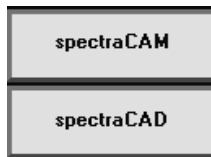
You can access spectraCAM from the control's Home screen. To access spectraCAM/spectraCAD:

1. Press the More button on the General Mode Button bar.

A table with more buttons pops out.

2. Press the spectraCAM button to open spectraCAM or the spectraCAD button to open spectraCAD.

The appropriate application opens. You can now begin your CAD/CAM session.

For more information on spectraCAM, refer to the spectraCAM Milling User's Guide (34-6741-0000) for more information.

For more information on spectraCAD, refer to the spectraCAD User's Guide (34-7791-0000).

Using a Mouse with spectraCAM

You need a mouse to use all of the features of spectraCAM. Use the mouse to move the pointer around the spectraCAM window on the control screen. Use both the left and right mouse buttons to perform various actions in spectraCAM.

Using the Left Mouse Button

Use the left mouse button to:

❑ make menu and tool bar selections

❑ select CAD geometry in the session window

❑ select data fields in dialog boxes

❑ close, reduce, and enlarge windows and dialog boxes

❑ accept or cancel actions from dialog boxes or menu selections

Using the Right Mouse Button

Use the right mouse button to:

❑ check which layer a selection is on

❑ change the cutting side of a selection

❑ change the cutting direction of a selection

❑ select primary and secondary layers for selections

❑ change start points of cutting paths for selections

For more information on using the mouse in spectraCAM, refer to the spectraCAM Milling User's Guide (34-6741-0000) for more information.

For more information on using the mouse in spectraCAD, refer to the spectraCAD User's Guide (34-7791-0000).

Installing the Mouse on the Operator Station

Connect the mouse through the serial port on the back of the operator station. See Section 1 of this guide for information on the serial port and the operator station.

Appendix A : Flood Coolant

Flood Coolant

The VMC-5000 can be ordered with optional Flood Cooling. This provides coolant to the workpiece resulting in reduced tool wear, and evacuation of chips. The coolant pump and plumbing are installed at the factory; the operator is required to fill the coolant sump.

Coolant

The VMC-5000 requires a minimum of seven (7) gallons of coolant, and holds a maximum of nine (9) gallons. Light Machines recommends the use of Valcool coolant. The sump is shipped dry, and must be filled before using the flood coolant system.

To fill the sump:

1. Make sure the coolant is **thoroughly** mixed before filling the sump. The coolant concentrate should be diluted with water according to the following formula: 1:17 (6%) to 1:25 (4%).

2. Press the Emergency Stop button on the Controller.

3. Open the safety shield of the machining center.

4. Carefully add at least seven (7), and not more than nine (9) gallons of coolant mixture to the sump.

5. Close the safety shield, and return power to the machine.

Because of evaporation of water in the coolant and other minor losses, coolant levels should be checked weekly, along with other maintenance. The coolant also becomes dirty, and should be replaced periodically.

Flood Coolant Setup and Operation

Flood Coolant comes as a factory installed option. However, the user is required to seal the X axis servo cover extension box. Light Machines provides a tube of sealant to use to form the gasket. See Section 2, Installing the X axis Servo Extension Box, for more information about this installation.

Coolant Flow

The pump draws coolant from the sump in the bottom of the enclosure, and cycles it through the adjustable nozzle directly onto the workpiece. The coolant pump is located beneath a cover in the right rear corner of the enclosure. It is turned on and off through M codes in the NC program. The nozzle tube is adjustable, and has a petcock built into it to control the flow of coolant.

The coolant runs off the workpiece, taking the chips with it, and back into the sump area. There are screens over the sump to separate the chips from the coolant. These screens should be cleaned periodically. If the chips are not removed from the screens, they will block the flow of coolant back into the sump.

Important! Do not drill or make holes of any sort in the enclosure, especially on the back panel. If holes are drilled to mount an aftermarket accessory or other device, coolant is likely to leak out and could be hazardous to other components of the machine.

Maintenance

Flood coolant machines require only a little more maintenance than non coolant machines. It is important the following additional checks be made.

☐ Keep sump screens clean to maintain the flow of coolant through the system.

☐ Wipe down the work area any time the machine is not going to be used for more than a day. The coolant is slightly corrosive, and if left on the cross slide and other components it will cause a blackening or staining.

☐ The spindle should be cleaned and a good quality commercial machine tool rust inhibitor should be sprayed up into the spindle nose once a week.

☐ Check and refill coolant levels once a week.

1. Open the safety shield.

2. Remove one of the sump screens.

3. The coolant should be within an inch of the screen.

☐ Coolant should be replaced once a month, depending on machine usage.

1. Press the Emergency Stop Button on the controller and remove power from the machining center.

2. Place an approved ten (10) gallon container below drain plug at the right rear corner of the machining center.

3. Remove the drain plug.

4. When coolant has drained into the container, replace plug and properly dispose of the used coolant.

5. Open the safety shield and refill sump with fresh coolant. Be sure the coolant is mixed thoroughly before filling the sump. **Do not** try to mix coolant in the sump.

Appendix B: System Backup

Backing up the Controller

It is a very good idea to make backup copies of all programs and tool data run on the VMC-5000. Should you experience a controller failure or be required to replace the controller, you will be able to reload the information rather than spend many hours reprogramming the lost data.

The A2100 Controller provides the capability to backup specific programs and/or configuration data to an external device or disk. This allows the user to store the information in case of a hardware failure or accidental deletion of files. The files may be backed up to either a floppy disk, a network server if the controller is connected, or any other device that accepts file transfer protocols.

You may use the backup programs to save Part Program Data, Tool Data, Offset Information, Machine Data, or an entire System Backup. The first part of this appendix explains how to backup part and tool data. The second part describes the steps involved in the more thorough backup of machine and system data. There are advantages and disadvantages to each of these choices.

The Part, Tool, and Offset information backups are shorter and can usually be done before or after running a program. Getting in the habit of this type of backup ensures that you have the most recent information saved.

Backing up the Machine Data or an entire System Backup is more involved and is usually done on a weekly basis. In practice this is an excellent idea. However, because of software issues and file saving procedures, these backups are better done as individual data backups rather than a whole system backup.

Backup Part Program Data

To backup a single part program to a floppy disk, use the following procedure:

1. With the controller on, touch the Home button on the screen.

2. Select the NC Programs menu button.

3. In the NC Part Program directory, choose the program to be saved to disk.

4. Touch the Transfer Program button, then press the Transfer Program Out menu button.

5. Insert a formatted disk into the floppy drive.

6. Select the Keyboard menu button on the screen to activate the keyboard. Enter A:\ and the name of the file to be backed up.

7. Press the Start Transfer menu button to begin the backup process. When backup is complete, the NC Part Programs directory will be displayed.

Tool Data Backup

1. With the controller up and running, touch the Tools button.

2. Choose the Active Tool Set menu button.

3. Touch the Copy Table menu button.

4. Then touch the To Backup File menu button.

5. Select the Backup menu button. If you have not selected the Tool Data Tables menu button, do so now.

6. Insert a formatted disk into the floppy drive. Activate the keyboard by selecting the Keyboard button.

7. Enter A:\TOOLDATA.BCK (or the filename you choose).

8. Touch the Start Backup menu button to activate the backup process. When the backup process is complete, the Tool Manager will be displayed.

Saving Offset Tables

1. Go to the Home Screen.

2. Press the Display Mode button.

3. Select the Offset Displays menu button.

4. The Offset Tables that can be backed up are listed in the display. For this example we will use the Machine Offsets Table. Touch Machine Offsets to select it, then touch the green check menu button.

5. Press the Copy Table button, and then the To Backup File menu button.

6. Select the Backup button if you haven't already.

7. The Machine Offsets Table Menu button should already be selected. If it is not, touch it now.

8. Insert formatted disk into the floppy drive.

9. Select the Keyboard menu.

10. Type A:\MACHOFF.BCK (or whatever name you select)

11. Touch Start Backup to begin the process.

When the backup is complete, the Machine Offsets Table will be displayed.

Suggested Backup Programs

It is also suggested that you make backup copies of the Machine Application, Servo Configuration, and Axes Configuration Programs. All three of these programs follow the same steps to backup.

1. From the Home screen, select More.
2. Select the Configuration button.
3. Choose the Machine Application (or Servo Configuration, or Axes Configuration) button.
4. Select the Backup and Restore option.
5. Choose Backup.
6. Insert a formatted disk into the floppy drive. Activate the keyboard by selecting the Keyboard button.
7. Enter A:\TOOLDATA.BCK (or the filename you choose).
8. Select the Start Backup button.

When the process is complete, the program screen that you are backing up will be displayed.

Service Level Backups

In addition to the single NC file or program backups, there are procedures that allow you to backup machine operation data. These are the backup operations that will save your information should the controller go down or the hard drive become damaged. The Machine Data Backup and the System Backup are accessed through he Service button. Once inside the service menu, you see a selection, "Backup and Restore Machine Data". When you select this button, the Backup Menu appears with the options Machine Data, and Everything. The next sections explain this in more detail.

Backup Machine Data

Machine Data Backup is accessed at the Service Password level. Selecting this will create a file about 200K in size and contains the following files.

♦ Cycle Parameter Table

♦ Axes Configuration

♦ Servo Configuration

♦ MAI Configuration

♦ Friction Compensations

♦ Security Configuration

♦ Part Program Translation Setup Tables

♦ Tools Configuration

♦ CNC Communications Configuration

♦ Axes Error Compensations

Machine Data Backup

The following procedure will backup Machine Data. It is recommended that this data be backed up when the machine is installed and any time machine dependent data is changed.

1. From the Home screen, press the More button, then the Service button, and then the Backup and Restore button. The Backup Manager screen should appear.

2. In the Backup manager Screen, select "Backup" under Operation, and select Machine Data under Data.

3. Note that the default file name is M1Y-xxxx.BCK, where xxx is the last four digits of the serial number on the controller. Using the default file name will back machine data up to the User directory on the controller hard drive. If you wish to store the information on a floppy disk, (the recommended method) make sure A:\ is at the beginning of the file name (A:\M1Y-xxxx.BCK).

4. Insert a floppy disk into the drive.

5. Select the Keyboard button and type A:\ at the beginning of the filename (if you are backing up to a floppy disk).

6. Press the Start Backup button. The process will take about a minute if you are backing up to a floppy.

The Service menu will reappear when the backup is complete. Label the disk with the control serial number and store in a safe place.

System Backup

The complete System Backup is done the same way as the Machine Data Backup, with the exception of selecting the EVERYTHING button in the Backup Manager, under Data.

Choosing EVERYTHING will create a file approximately 800K in size. This file will contain data from the following:

- Active Tool Set
- Axes Configuration
- Multiple Setup Table
- Cycle Parameter Table
- Fixture Offsets Table
- Home Menu Configuration
- Machine Offsets Table
- Friction Compensations
- Pallet Offsets Table
- Process Control Data Table
- Programmable Tool Offsets
- Local Tool Resource File
- MAI Configuration
- Servo Configuration

Restoring Data

To restore data from the Backup Disk:

1. From the Home screen, select the More button.
2. Press the Service button, the Backup and Restore button, and then the Restore button.
3. When the screen appears, press the Keyboard button and type in the file name and directory you will be restoring FROM.
4. Touch the Start Restore button. This process will take several minutes.

Index