

Frequently Asked Questions about Sherline CNC

Following is a summary of the questions answered in this document in order of presentation:

- How hard is it to hook up the computer?
 - I already have a computer. Can I use mine and save some money?
 - I already have a Sherline mill. Can I buy the rest of the components and retrofit my mill?
 - I live someplace other than the United States and our current is 230 volts, can I still use my CNC system here?
 - What format are the instructions in?
 - Where can I find a current copy of all of the instructions on the Sherline Documentation CD?
 - What is Linux?
 - What is EMC?
 - I already know how to use Windows®, why should I have to learn to use a new operating system?
 - Are you just trying to save money or is there a reason you use Linux and EMC as your operating system and control program rather than using Windows® and a Windows-based program?
 - What programming language does the EMC use?
 - What other programs are included on the CD?
 - What do I do if I don't know how to use Linux or CNC?
 - I'm ready to start writing the G-code to make my part. How do I get started?
 - How do I run a CAD file on my CNC system?
 - How do I transfer my G-code files to my CNC computer from my CAD / CAM computer and vice versa?
 - Will G-code created by any given CAD/CAM software work with EMC?
 - What is a "feedback loop" and do I need one?
 - How fast will the stepper motors move the slides?
 - How do I use my inch CNC system in metric mode?
 - My stepper motors make noise, but nothing is moving. What should I do?
 - I can run my g-code in the backplot program but I can't get the stepper motors to move. What's wrong?
 - The instructions mainly address using a CNC mill. What about using it with a lathe?
 - Can I cut threads on a Sherline CNC lathe?
 - I have read the entire Sherline Documentation set front to back, now what to I do?
 - For some reason my axes are going in the wrong direction when I Jog. What am I doing wrong?
 - I'm getting an error message that reads "Radius to end of arc differs from radius to start.." What does that mean?
 - Does Sherline use the latest version of EMC?
- I want to connect a USB device to my Sherline computer. How do I configure Linux?
- Other than calling Sherline, where are some other places I can go to get help on my specific questions on machining, CNC and using Sherline tools?
 - What is GPL and what does it mean to me?
 - Is there a way to convert a photo to g-code? Is there a way to convert a photo to g-code?

- What if I have a question that isn't answered here?

How hard is it to hook up the computer?

Sherline has made everything as easy as possible for you. The computer has the operating system and software pre-installed. The driver and power supply are also pre-installed inside the computer box. All you have to do is plug in some cords to get up and running. There are some good pictures in the printed step-by-step "Quickstart" guide that comes with your system. They show you how to plug in your new Sherline CNC computer and stepper motors. (Don't forget to plug in the parallel cable...) In addition to the printed sheet, it is on the Sherline Documentation CD that comes with the system where it is called quickstart.htm. Here you can see photos of the system and the back of the computer to see exactly how to plug everything in. If you aren't familiar with computers, please be sure to pay attention, as you can damage the system if you force things into the wrong plugs.

I already have a computer. Can I use mine and save some money?

You can, but you might want to think about whether it's worth it. Basically you have two ways to go:

1) Buy a new CNC-ready Sherline machine or retrofit your existing mill, add stepper motors (P/N 67127, \$75.00 each), buy a driver box to power the stepper motors (Sherline's is P/N 8760 for \$600.00) and plug that into the parallel (printer) port on your computer. Then you have the choice of loading Linux and EMC (which comes on CD with the driver box) and running the Sherline system or using a Windows® or DOS based software that you purchased elsewhere. If you are not familiar with Linux, doing the installation yourself could cause you some problems; particularly if all the components of your system are not compatible with Linux, but the new Debian version of Linux is much more user friendly than the earlier Redhat version so it will probably go smoothly. We just can't guarantee it, which is why we offer the system complete with the computer. Purchase of the driver box does NOT include free software support. If you purchase software from another source, make sure it will run with the Sherline driver box. They may want you to purchase a driver box and stepper motors from them. (For more information about implications of installing and using Linux compared to other operating systems, [CLICK HERE.](#))

or

2) Buy the complete Sherline system with a new computer, connect the cords, boot it up and start working right away. When we designed our system we initially planned to let customers do the Linux/EMC installation on their own computer to keep the cost down. When we found out how many variables could effect the installation we decided it would be worth it to include new computer with the OS and software already loaded and tested. That way we avoid having to answer all those phone calls from frustrated users trying to get their system to work. If you are not absolutely sure you are up to the task of installing Linux yourself we highly recommend that you purchase the complete system including the computer.

I already have a Sherline lathe or mill. Can I buy the rest of the components and retrofit my machine to CNC?

Yes. We now offer systems that include a retrofit kits for Sherline mill or lathe, stepper motors and the computer with drivers and software from the full system. Basically it's the complete system less the lathe or mill. Price for either mill upgrade is \$1775.00. Lathes can be upgrade for \$1650.00 (4000 series) or \$1660.00 (4400 series). See www.sherline.com/CNCprices.htm for part numbers and prices.

I live someplace other than the United States and our current is 220 volts, can I still use my CNC system here?

Yes, but you will need to toggle two switches for this to happen. The back of the computer's power supply has a red switch for switching between 110/115 and 220/230 volts. You will need to flip this switch to 230. You can easily switch it with a small screwdriver or any other small flat device. Secondly, you will need to switch the driver board's power supply so that it will be set to use 230 volt power. There is a switch that is basically identical to the one on the back of the computer. You will need to open up the computer's side panel by unscrewing the two screws on the back. The side you will be unscrewing is the one with the power switch for the driver board. MAKE SURE that all power is unplugged, as even if the computer is off it will still have live power to the driver board. Once you have removed the side panel you will need to flip the switch in the middle of the power supply on the side facing you. It is somewhat embedded into the side, so you might need to get a screwdriver instead of just using your fingernail like you might be able to do with the switch on the back of the computer. Be sure that you have the right setting for the power in your area, as having it switch wrong it will cause damage to the system.

If you are using the 8760 driver box to drive your stepper motors, the separate power supply included with that box will accept any incoming current from 100 VAC to 240 VAC and automatically provide the correct DC output current to the drivers. It comes with a USA type 3-prong grounded plug, so a different cord or a wall plug adapter will be required in countries that use other configurations.

What format are the instructions in?

All of the instructions are available in HTML format [.html]. This format is readable on just about any operating system without any troubles. Many of the files are also formatted in Adobe Acrobat [.pdf] and Microsoft Word [.doc] formats. It should be noted that PDF and Word files may not be readable on your system depending on the software you have installed. You will be able to open the HTML file in your favorite web browser, and .pdf files can be opened on Linux computers supplied by Sherline. The Adobe Acrobat Reader program required to open .pdf documents on Windows computers is available for free download at www.adobe.com if you do not already have it.

Where can I find the most current version of all of the instructions on the Sherline Documentation CD?

The most up-to-date 4.xx version of the instructions can be found at www.sherline.com/CNCinst4.htm. For those using a system purchased before January 1, 2005,

instructions for the 2.xx Redhat version can be found at www.sherline.com/CNCinst.htm.

What is Linux?

Linux is an operating system. An operating system is the basic set of programs and utilities that make your computer run. Some other common operating systems are UNIX (and its variants BSD, AIX, Solaris, HPUX, and others); DOS; Microsoft Windows; Amiga; and Mac OS.

What is EMC?

EMC is the Enhanced Machine Controller originally developed by National Institute of Standards and Technology (NIST). It is released under the General Public License (GPL). Currently NIST does very little work on the project, and it has been taken over by average Linux users and a handful of very talented coders. You can get up to date on the project and get more detail on the movement of it by visiting its website at <http://www.linuxcnc.org>

I already know how to use Windows®, why should I have to learn to use a new operating system?

From the standpoint of the cnc operator, the underlying operating system is relatively unimportant as long as it runs your programs properly. Most likely you really don't "use" Windows for much more than simply opening your programs or moving files around, and the Linux desktop works pretty much the same way as the Windows version. You will feel right at home. There are a few minor differences that are easily learned, but the way you navigate, open and close programs, move files and folders around and so on is pretty much the same. The key is that Linux offers some real advantages over Windows when it comes to running a cnc program. The next question goes into more detail on that subject.

Are you just trying to save money or is there a reason you use Linux and EMC as your operating system and control program rather than using Windows® and a Windows-based program?

When deciding which operating system and control program to use for our Sherline system I didn't choose the Linux based EMC program because it was free. I chose it because it was better for three reasons:

- 1) When Linux is controlling the EMC program that is all it's controlling, which isn't the case for Windows. At that time, cnc programs for Windows would stop cutting in the middle of a program and pause for a few seconds while Windows did some internal housekeeping chores which produces an unwanted machining mark on the part. The users of Windows programs at the time addressed this problem by buffering the information to the servo drivers through a second computer, but this adds more hardware and several hundred dollars cost to the cost of your cnc system simply to solve a problem that is not present with Linux. Since that time computer clock speeds have increased and Windows programs are more competitive to EMC, but I still don't think they are any better.
- 2) The EMC was originally developed by National Institute of Standards and Technology. I'm sure it cost millions of dollars to be developed. When the

government felt it no longer had to protect itself by having its own program, EMC was made available to the general public as an open source program; however, it wasn't a program the average person could use. A group of very dedicated engineers and intelligent hobbyists had already spent years working on it when I teamed up with them. With their help I felt I had a program that was superior to any Windows program when I bundled it with our cnc systems. Linux and EMC may be free to Sherline's customers, but I personally spent much time, money and effort testing and customizing it. Even though we pass it on to you at no charge, it sure wasn't "free" for us.

3) At the time I chose the EMC program I felt it had a much better method of handling cutter offsets (g41, g42) than any program available to the home machinist/hobbyist. Others have caught up, but again offer little or no advantage to offset the additional cost to the customer.

Sherline has several million dollars worth of cnc machines running in the factory downstairs, and I couldn't tell you what operating system they use. Why? Because it doesn't matter. What is important is whether it runs the control program as advertised, and the Linux does just that when teamed up with the EMC.

If you still aren't convinced, you do have the option of purchasing a Windows based control program from other suppliers to run your Sherline cnc machines. There are several available that work with our P/N 8760 driver box.

What programming language does the EMC system use?

The Sherline CNC system and EMC uses G-code. G-code is the standard language used to program most CNC machines in industry and is probably supported by your favorite CAD/CAM program. For simple shapes you can write the needed G-code yourself. For complicated shapes you can draw your part in a CAD program and export it to G-code which can then be used in EMC. (Exporting to G-code may require a second translator program if your CAD program can't do it.) More can be read about G-code and its uses in the CNCinst.htm file included on the Sherline Documentation CD-ROM.

What other programs are included on the CD that comes with my system?

We have included a number of free utilities that will help you create g-code. They are located in the "Utilities" folder that can be found on your CD. They are all programs that run on your Windows® computer, and the g-code files you generate can be saved in .txt or .ngc format and transferred to your Linux computer to be run. They can also be found on the Sherline web site at www.sherline.com/CNClinks.htm.

What do I do if I don't know how to use Linux or CNC?

These instructions will teach you what you need to get started. You can click on the icon for the instructions on your EMC desktop. The same set of instructions is also on the Sherline Documentation CD. You can get to it by putting the CD into your CD-ROM drive and opening the index.html file located in the getting-started directory on the CD. You also might want to take a look at the official

Sherline Instructions; they are located both on your desktop, and on the Sherline Documentation CD. The file is called CNCinst4.htm.

There is also a large amount of documentation on the internet. If our documentation doesn't provide the answer you seek, try your favorite search engine and search for your specific problem/question, chances are somebody else has had the same problem. There are also several user groups you can join that will most likely help you with support. While these are not Sherline specific groups, they will most likely be familiar with the system you are using, and if not something very close to it. A Sherline CNC owner's group of over 1000 members can be found at www.yahogroups.com by typing "sherlinecnc" in the query box. Be sure to research your question before posting to these lists. Some list members get frustrated when people ask questions before doing a bit of research first. You can learn more about these mailing lists here: <http://www.linuxcnc.org/links.html>. The Linux site at www.linux.org now incorporates a Google search function to help you find information about all aspects of Linux on their site. Basic help to get you started is available by calling Sherline at 1-800-541-0735 or 1-760-727-5857 during business hours or e-mail sherline@sherline.com. We will be glad to help you with the steps needed to get your CNC system up and operating, but we cannot write your g-code for you or tell you how to fixture the machine to make a specific part. That is part of being a machinist and will be up to you. Our CNC instructions included here will get you started, and the printed 48-page color manual that comes with each machine will give you the basics of machining.

I'm ready to start writing the G-code to make my part. How do I get started?

If you are new to the CNC community, it is suggested that you read the Sherline CNC instructions all the way through before attempting to make parts. It is a great way for those who aren't familiar with CNC or G-code to get started in coding. By the time you are finished reading the instructions you should be able to write simple programs and at the very least understand other, more complicated instructions.

How do I run a CAD drawing on my CNC system?

You can't run a CAD (Computer Aided Drafting) file directly in EMC or in most CNC control programs. You first need to get the vector information in the CAD program translated into the language of G-code, which is a text file consisting of just letters and numbers. This is what the EMC program uses to talk to the stepper motors that move the machine. With complete CAD/CAM programs like MasterCam®, SurfCam® or GibbsCam® you can do a drawing and then output G-code, but these professional level programs can be quite expensive. If you already have a CAD program and need to get files into G-code, there are several free or very inexpensive "translator" programs available that can do this for you. We include links to a few at www.sherline.com/CNClinks.htm. To turn a CAD drawing into G-code you would open one of these programs on your Windows® machine. Then you would open the .dxf or .stl drawing file in that program and save it

as a G-code file. The resulting text file can then be opened in any program that can read .txt files, such as Wordpad®, WordPerfect® or Microsoft Word® in Windows or by DOS and Linux programs. In EMC you would save the file to your “gcode” folder (see below) where it can be opened and run. Keep in mind that EMC uses very simple G-code, and the code written by a high end CAD/CAM or translator program may need some lines removed or modified before it will run properly. You can edit the G-code from within EMC and watch it run in the “Backplot” program before turning on the stepper motors to actually make a part.

How do I transfer my G-code files to my CNC computer from my CAD/CAM computer and vice versa?

There are a few ways you can do this, but the easiest way is to save your g-code as text files on a floppy disk or write them to a CD on your CAD computer, then take the floppy disk or CD and put it into your Linux computer (Note: The file needs to be saved with a name consisting of eight characters or less in order to be able to transfer the file to the EMC program.) On the desktop there will be an icon that says “Floppy.” Left click on it to open a browser window for the floppy drive. Simply drag and drop the file onto the desktop or into your G-code programs folder which is called “gcode.” When you are in the EMC, you can open this file by browsing to it. To save a file to the floppy from your computer, just drag the file from the “gcode” folder to the floppy window. Close the browser window for the floppy by clicking on the “X” in the upper right corner of the window. To save the file you will have to right click on the floppy icon on the desktop and use the “unmount” command. This completes the file saving process. Do not remove the floppy from the computer until the light on the drive goes out after unmounting.

Will G-code created by any given CAD/CAM software work with EMC?

Not necessarily. Some “fine tuning” may be required before g-code created by some CAD/CAM programs will run successfully on EMC or any other g-code control program for that matter, because post processors create code for particular machines that are not all the same. EMC uses industry standards for g-code, but it is up to the manufacturers of CAD/CAM programs to adapt their product to EMC, not the other way around. For example, some unassigned g-codes may be used by one company to do one thing and by another company to do something else. Also, Sherline’s simplified system does not support all g-codes such as some canned cycles, limit switches, etc. EMC will not understand these codes. It will be necessary to pre-run the g-code in the backplot program and, if the program stops running and highlights an area of offending code, delete the necessary lines using the editor. This is why you will still need a basic understanding of g-code even if you are using a CAD/CAM program to generate it for you. As EMC becomes more prevalent in the CNC world, more software manufacturers will write specific post-processors for it, but Sherline has no input into that process or control over how these companies write their software. In the absence of a specific EMC post processor, we recommend you choose the “Mazak” or “Hurco” post processor as the best alternative to get as close as possible

on 3-axis projects right from the start.

What is a “feedback loop” and do I need one?

Most systems that use stepper motors do not have any type of “feedback” arrangement that uses an encoder to report movement/position information back to the processor. A true feedback loop using servo motors with encoders can compare specified movement to actual movement and compensate for any skipped steps. It can also allow reverse engineer existing parts by using a probe to touch points on its surface and create a drawing of the part. In general, servo motors are more expensive than stepper motors. Stepper motor systems rely on the fact that if you put power into stepper motor it will accurately rotate a certain amount that happens to be 1.8° per step. By making the electronics more complex you can micro-step a stepper motor and have it turn only 0.9° or 400 steps for each revolution of the leadscrew. (This is what Sherline does.) By sizing the motor appropriately for the mechanics of the machine you can make a safe assumption that the motor will always have enough power to make the desired move without stalling. Specifying a proper feed rate (see next question) will also help assure that your machine does not “skip steps,” making a feedback loop unnecessary. Sherline feels a stepper motor system gives you the “most bang for the buck.”

How fast will the stepper motors move the slides?

A fact when dealing with stepper motors is that the faster they turn, the less torque they have. This is because the power for each pulse is on for shorter periods of time; therefore, it’s a good idea to keep feed rates 20% below maximum when running programs with many short moves, particularly on the Z-axis. The maximum feed rate in EMC is 22 in/min. Therefore it is good practice to keep your feed rate below about 18 in/min or 450 mm/min. Some of our customers have counter-balanced the weight of the Z-axis with a simple rope-pulley-weight device to reduce wear on the Z-axis leadscrew. See the <http://sherline.com/CNCproj.htm> section of our website for a photo and description of how one person did this. You’ll find that the stepper motor system we use quite reliable and capable of running complex programs without losing steps when used properly.

G-code tips: Start and end each program with a percentage sign. Put a g40, g49, g21 (Metric) or g20 (inch) and g90 in the first line of code to cancel out any previous codes that might be left over from the last program, and always end the program by returning to the same place you started. Keep in mind which g-codes EMC does and does not support. Some canned cycles are not supported, for example.

How do I use my inch CNC system in metric mode?

You can toggle between metric and inch modes by using a g20 or g21 code. A line of code beginning with “g21” tells the machine that all numbers you enter after the g21 are now in millimeters instead of inches. The software will make the calculation to move the machine the correct metric distance using the inch leadscrew. Using the g20 code will allow you to switch back to inch dimensions or to enter inch dimensions on a machine with a metric leadscrew. So if you have an inch machine and always

program in inches, why should you worry about it? Because EMC remembers the last code run even when the machine is turned off. Therefore, it is a good idea to always put in a g20 in the first line of your inch program just in case the last program run had a g21 in it. On metric machines start with a g21 to similarly protect yourself from a g20 code in the previous program.

When starting the EMC program, select the proper file for your machine; that is, on an inch machine, click on the desktop icon that says "Sherline Inch" or for a machine with metric leadscrews, select the icon called "Sherline MM." This tells the program what leadscrew pitch to assume in its travel calculations.

My stepper motors make noise, but nothing is moving. What should I do?

The likely problem here is your mill is binding or not properly lubricated. With the more significant stress of quick back and forth motions and the high number of turns possible in a short period of time your mill will need to be more frequently lubricated than hand driven applications. Under continuous CNC use we recommend that Sherline CNC mills be lubricated once every four hours. Refer to the mill manual or <http://www.sherline.com/lubricat.htm> for more detailed lubrication instructions. Note also that when stepper motors are powered up but not yet running they do tend to make a slight buzzing or hissing noise. This is normal.

What is the difference between "unipolar" and "bipolar wiring for stepper motors?"

Stepper motor driver design can be either Unipolar or Bipolar. Without getting into circuit design details, from the standpoint of the user, bipolar wiring requires a much more complicated (and expensive) driver and does not allow microstepping. Sherline made the engineering decision to design our driver box around a unipolar layout mainly because it allows half stepping (micro-stepping) which provides a very fine 400 steps/revolution resolution. We specified a sufficiently powerful stepper motor to begin with so that you have plenty of power using this more economical, high resolution design. To learn much more about about the differences between the two wiring schemes see <http://eio.com/jasstep.htm#types>.

I can run my g-code in the backplot program but I can't get the stepper motors to move. What's wrong?

That's an easy one. Most likely the parallel cable from the driver box is not plugged in to the parallel port in the back of your computer. Sometimes people don't read the Quickstart Instruction Sheet and forget to do this.

The instructions mainly address using a CNC mill. Can I also use it to run a lathe?

Certainly. Sherline offers CNC upgrades and complete packages for the lathe as well as the mill. Although EMC does not yet have a lathe-specific version written for it, writing g-code for a lathe tool path is pretty easy to figure out. The lathe uses just the X and Z axes. (In machining, the spindle axis is always called the Z-axis.) Keeping this in mind it is not difficult to re-orient your thinking from mill to lathe.

Can I cut threads on a Sherline CNC lathe?

Not using CNC. Cutting threads on a lathe requires a direct relationship between spindle speed and movement of the tool. Manually this is done with a set of gears. In CNC this is done by means of accurate encoders on the spindle and Z-axis in order to maintain the relationship between the two during cutting. EMC does not yet have the provision to accept the input from these encoders. Because CNC threading is not a common operation, we felt it would unnecessarily increase the cost of the system for all users to incorporate this function when only a few would use it. However, threads can still be cut, but not using CNC. You can still cut them manually using the P/N 3100 threading attachment and the handwheels.

I have read the entire Sherline Documentation set front to back, now what to I do?

Well, our aim was to just start you on your path of learning Linux and CNC. There is a whole lot you can learn beyond these instructions. For starters you can check out the EMC website <http://www.linuxcnc.org> and get on the mailing lists. There are also more complex books and documentation on G-code available at your favorite book store or library.

For some reason my axes are going in the wrong direction when I Jog. What am I doing wrong?

This means that your ini file is improperly set. To fix it you will need to open the file called mill_inch_freq.ini for machines with inch leadscrews or mill_mm_freq.ini for machines with metric leadscrews. The files are located at /usr/local/emc. To get there, you click on: K-Menu > Quick Browser > Root Folder > usr > local > emc. To open the .ini file, just single click on it. You will need to change INPUT_SCALE and OUTPUT_SCALE values from +1600 or -1600 to the opposite value; i.e., if yours is +1600 and it is jogging backwards, change it to -1600 and vice versa.

I'm getting an error message that reads "Radius to end of arc differs from radius to start.." What does that mean?

This means the endpoint of the previous line does not match the coordinates specified and required by the radius command. The tolerance for the precision of these two points can be adjusted in the INI files of version 4.51 or higher.

Making Tolerance Changes to EMC

Computers can be maddening devices because they demand accuracy beyond what normal human beings find useful. This can be especially true when producing cnc programs, and where this condition shows up the most is when you are working with radius ending points.

In order to cut a curve, the g02, g03, and r commands must be used. When you use these commands, you must tell your computer the exact ending position of the curve. The computer will give you a message stating that your end x y position is in error but it will not fix it for you. The computer is demanding an accuracy far greater than the user believes the part needs and who may have carelessly made a few rounding errors.

Because Sherline cnc customers have been having problems with this, we decided in 11/06 to open up this tolerance up to a maximum error of 0.001" (0.002 mm) standard.

If you are still constantly having this kind of problem, you can either increase the accuracy of your drawing or force the computer to accept your faulty math by opening up the tolerance in the INI files although Sherline doesn't recommend it, believing two wrongs don't equal one right. The radius point tolerance can be altered in EMC v4.51 and higher in the INI files. Altering the INI files of previous versions will not fix the problem; however, the EMC can be upgraded online without the need for reinstallation of the software.*

To change the tolerance for radius and tangent points, simply edit the numerical values in the INI files (mill_inch_freq.ini and mill_mm_freq.ini). The lines should be by default:

INCH_TOLERANCE = 0.0005 or 0.0010 (SP setting)

MM_TOLERANCE = 0.001 or 0.002 (SP setting)

Changing the Axis Scale

Changing the axis scale (for leadscrews with a thread count other than 20 TPI or 1 mm for example) can be achieved by editing the input and output scale values in the EMC ini files (mill_inch_freq.ini or mill_mm_freq.ini for the Standard and Metric versions of the EMC, respectively). To open these files, go to K-menu > Quick Browser > Root Folder > usr > local > emc > Open In File Manager, which will open the directory containing the EMC files, and then click on the file you wish to edit.

About a quarter of the way into the file you will see the section labeled "; Axes sections -----" where the settings for all four axes are located. For each axis you wish to change the scale on, go down to the lines that read "INPUT_SCALE =" and "OUTPUT_SCALE =". These values are normally 16000 (or -16000) in the Inch ini file and 800 (or -800) in the Metric ini file. Changing these values will affect how far the axes travel for every unit jogged. For example, dividing the values by ten, would cause an axis to move .1" when jogged by 1". So, when a machine is switched from a standard 20-TPI leadscrew to a 4-TPI leadscrew, for instance, changing the scale values from 16000 to 3200 would compensate for this and return the machine to a normal movement scale. When changing these values, make sure to leave the number either positive or negative, as changing this will reverse the direction the axis moves unless this is what you want to do.

There are a number of other values in the ini files that can be altered to change the way the EMC functions, although experimenting with these may have effects ranging from the EMC functioning properly to your program becoming impaired or inoperable. When you're finished making whatever changes you like, simply save the file and restart the EMC if it's currently open. A reboot should not be necessary, but is a good practice to make sure the changes take effect. Should you ever need original copies of the ini files, replacements (and their instructions) can be downloaded at the EMC section of the Sherline website at <http://www.sherline.com/emc/ini.html>.

*Sherline Linux computers of version 4.xx are configured for connection to the Internet. Once connected to the

Internet with the proper network cable, open a terminal (second to right icon on the taskbar) and type:

sudo apt-get update

Hit [Enter] key to start download, then type:

sudo apt-get install emc emc-modules-2.6.16-rtai

Hit [Enter] to install download

Does Sherline use the latest version of Linux and EMC?

Not necessarily, and here's why. Sherline uses the latest version of EMC that we have tested and are sure is working properly. From time to time small bugs are fixed or more features are added by people in the Linux group, and a newer version of EMC may become available on another web site, but we will not use it until we are sure that the other "fixes" that are inevitably incorporated into the new version don't cause other unforeseen problems. The version of EMC that we install on our computers, offer for sale on CD or make available on our web site for download is the latest tested, stable version that is supported by our instructions. If you choose to download a later version, you do so at your own risk. The newer version may fix a fault that we did not consider a problem when using it on a Sherline machine, and other fixes may cause previously unknown malfunctions. If you have problems with any version of EMC newer than the one we currently support, please do not call Sherline for technical assistance.

I want to connect a USB device to my Sherline computer. How do I configure Linux?

Transferring programs from a Windows® computer can be done on a floppy disk or a CD, but flash memory devices and cards are also an easy way to transfer data using the USB port. Sherline Linux versions 4.38 and higher already have USB support, so your flash device is accessible from a USB icon on the desktop. Clicking on the icon mounts the flash drive when it is plugged in. Remember to unmount the drive before removing it or it will not be able to be accessed without rebooting the computer. (Click on the drive icon and select "unmount.")

If you have an older version of Linux that does not have USB support, here is how to configure your Linux computer to accept them:

- Boot up to the Linux desktop.
- Go to root directory.

(NOTE: Type carefully; changes made incorrectly can cause severe damage that will require a complete reinstallation to fix!)

· To open the root directory, go to the lower left-hand corner of the Linux desktop and click on the **K**-icon. (It is where the "Start" menu would be in Windows.) Select *Logout > End session only > OK*.

· In the appropriate boxes type the words (shown in bold)—username: **root**, password: **sherline**

· Run the Terminal Program (in the *K* menu, the icon looks like a little LCD monitor)

· Under *sherline@localhost:~\$* type the following commands (shown in bold):

mkdir /media/flash and press [Enter] (Note the space between *mkdir* and */media*)

In the next line under *sherline@localhost:~\$* type the following command (shown in bold):

mc and press [Enter] twice (to enter the Midnight Commander)

- Select and highlight the */etc directory* and press [Enter]
- Scroll down and highlight the file called *modules*
- Press [F4] to edit the file. In the next empty line add the following text:

usb-storage and then press [Enter]

- Press [F10] to save and exit > Select [Yes]
- Scroll up to find and highlight the file *fstab*
- Press [F4] to edit the file. In the next empty line add the following text:

/dev/sda1 /media/flash msdos,vfat noauto,users 0 0 (0 = zero zero) and press [Enter], (Note the space between: *sda1* and */media, flash* and *msdos, vfat* and *noauto, users* and *0, 0* and *0*)

- Press [F10] to save and exit > Select [Yes]
- Press [F10] to exit Midnight Commander > Select [Yes]
- Under *sherline@localhost:~\$* type the following command:

exit and press [Enter] (this command will end the Terminal Program.)

- To leave the root directory and go back to the sherline directory:

Click on the *K*-icon, select *Logout > End session only > OK*

Type, username: **sherline**, password: **sherline**.

- On the Linux desktop, right click anywhere on the empty desktop and select *Create New>Device>Hard Disc Device*.

- Under *General*, instead of *Hard Disc Device* type the words **Flash Media**.

- Under *Device* from drop down menu choose */dev/sda1 (media/flash)*.

- Click OK and right click again on the new icon (*Flash Media*) and select *Properties*.

- Under *Permissions* choose under *Group and Others, Can Read & Write* and check the box in front of the words *Is executable >* click OK

In newer versions of BDI under *Properties >General* if you click on the hard disc drive icon there is the menu with icons where the appropriate icon can be found and selected.

Note that the Flash Media device uses the same Mount and Unmount commands as a floppy drive.

Other than calling Sherline, where are some places I can go to get help on my specific questions on machining, CNC and using Sherline tools?

Sherline owner discussion groups can be found at <http://www.yahogroups.com>. Just go to the [yahogroups.com](http://www.yahogroups.com) web site, type in the name of the group you wish to monitor or join and select it from the list provided. You can click on “join this group” to become a member or you can just read the archives without joining. These groups can be a very helpful resource for anyone new to machining and a great place to go to get advice from experienced users on specific problems. Here are some of the groups:

- Sherline (2000+ members)
- Sherline CNC (New May, 2003, 1000+ members)

- CAD/CAM/EDM/DRO (not Sherline specific, 4000+ members)

You can always find the most up-to-date version of these instructions at www.sherline.com/CNCinstructions.htm.

What is GPL and what does it mean to me?

GPL is a license that Linux and EMC are both released under. Basically (very basically) it states that all code released under this license can be used, modified or sold as a prepackaged product, but it must be released under the same license after you modify it. This helps the Linux and other open source communities grow as people contribute to it. What does this mean to the average user? Nothing. If you decide to hack up the kernel, however, you are required to submit those changes back to the community. More information about GPL can be found here: <http://www.gnu.org/copyleft/gpl.html>

Is there a way to convert a photo to g-code?

Yes, there are programs that will translate the dark/light scale of a photo into assumed heights (darker is deeper) and create a 3D g-code program to reproduce the contours. One called DeskART will import a BMP, GIF, JPEG, WMF or TIFF image and convert it into a DXF Surface Mesh of 3D faces or write it directly into machinable G-code. See www.deskam.com/deskart.html. They offer a free 30-day trial version of this software. DesKAM also offers several other types of programs for engraving. Prices are listed at www.deskam.com/products.htm.

What if I have a question that wasn't answered here?

If you need more information about prices, accessories or availability, customers in the USA or Canada can call our toll free number: (800) 541-0735 M-F, 7:30-5:00 PM (Pacific). If you have other Sherline-related questions and we can't answer them, we'll do our best to point you in the right direction. Outside the USA or Canada, call (760) 727-5857 or fax (760) 727-7857. You may also e-mail Sherline tool related questions to craig@sherline.com. Linux-specific questions from system owners should be directed to the Linux group at www.Linux.org. The best source for information on Sherline tools 24 hours a day is always our web site at www.sherline.com. Sherline tools and accessories can be ordered factory direct 24 hours a day at www.sherlinedirect.com or you can see our Sherline dealer list for a dealer nearest you.