



## G CODE & THE SHERLINE MILL



# TABLE OF CONTENTS

I. Purpose.....	p4
II. Background .....	p4
III. Overview of the <i>Sherline Mill</i> .....	p4
IV. Setting Up the <i>Sherline Mill</i> .....	p4
V. <i>Sherline Mill</i> Accessories.....	p5
A. End Mills.....	p5
B. End Mill Holders.....	p5
C. Flycutter.....	p6
D. Milling Collet Set.....	p6
E. Mill Tooling Plate.....	p6
F. Mill Vise.....	p6
G. Step Block Hold-down Set.....	p6
VI. Orientation of the <i>Sherline Mill</i> .....	p7
VII. Limits of the <i>Sherline Mill</i> .....	p7
VIII. Centering the <i>Sherline Mill</i> .....	p8
A. Manually.....	p8
B. Using a Test Program.....	p9
IX. Operating the <i>Sherline Mill</i> Manually.....	p9
X. Operating the <i>Sherline Mill</i> Using the <i>Sherline CNC (Inch) v4.51 Program</i> .....	p10
XI. Operating the <i>Sherline Mill</i> Using the <i>Sherline CNC (Metric) v4.51 Program</i> .....	p10
A. Starting the Program.....	p10
B. Program Basics.....	p11
C. Different Screen Information.....	p12
D. Operating In Manual Mode.....	p13
E. Operating In Automatic Mode.....	p15

<b>XII. Overview of G Code.....</b>	<b>p16</b>
<b>A. Syntax.....</b>	<b>p16</b>
<b>B. Tables.....</b>	<b>p17</b>
<b>C. Example G Code.....</b>	<b>p17</b>
<b>(1) A Basic Circle Program.....</b>	<b>p17</b>
<b>(2) A Circle Program with Offsets.....</b>	<b>p19</b>
<b>(3) Rectangular Box with Cutter Compensation.....</b>	<b>p20</b>
<b>(4) Pulley Program.....</b>	<b>p21</b>
<b>D. Other Useful G Codes.....</b>	<b>p22</b>
<b>XIII. G Code Writing Suggestions.....</b>	<b>p25</b>
<b>XIV. Suggestions for Locking Down a Part.....</b>	<b>p28</b>
<b>XV. Feedrate vs Cutting Rate.....</b>	<b>p28</b>
<b>XVI. Other Resources.....</b>	<b>p29</b>
<b>APENDIX A: G &amp; M Code Reference Tables.....</b>	<b>p30</b>
<b>APENDIX B: <i>Sherline</i> Accessories Shop Guide For the Mechatronics Lab.....</b>	<b>p33</b>
<b>A. CNC Mill Z-axis Backlash Lock, P/N 4017Z.....</b>	<b>p34</b>
<b>B. <i>SHERLINE</i> End Mill Holders.....</b>	<b>p37</b>
<b>C. <i>SHERLINE</i> Flycutter.....</b>	<b>p39</b>
<b>D. <i>SHERLINE</i> Milling Collet Set.....</b>	<b>p41</b>
<b>E. <i>SHERLINE Mill</i> Tooling Plate.....</b>	<b>p42</b>
<b>F. <i>SHERLINE</i> Mill Vice.....</b>	<b>p44</b>
<b>G. Sherline Step Block Hold-down Set.....</b>	<b>p47</b>

## I. Purpose

The purpose of this document is to serve as an introductory explanation of how G Code functions. The aim is that those reading this will be able to learn the basis of G Code, and, more specifically, G Code writing for the Sherline CNC Mill in use by the Mechatronics Lab (GE 423) at the University of Illinois at Urbana-Champaign.

## II. Background

G Code is the programming language supported by a variety of CNC Mill Machines. While it is standard in theory, many different manufacturers and different milling machines will adapt G Code to serve their desired functionality. The G Code discussed in this overview is compatible with the *Sherline Mill*.

## III. Overview of the *Sherline Mill*

In principle, the Sherline Mill is has the physical ability to move freely on four axes: 'X', 'Y', 'Z', and 'A'. However, the model in use by the Mechatronics Lab is restricted to the 'X', 'Y', and 'Z' axes. Control of each axis can be accomplished in one of three ways:

- (1) Manually: If the Power Switch\* to the Mill is in the off position, then the crank wheel on each axis can be moved freely.
- (2) By the Sherline CNC (**Inch**) v4.51: If the Power Switch\* to the Mill is in the on position, then you can use this program to operate the 'X', 'Y', 'Z', and 'A' axes.\*\* Once turned on, the control of the axis is done by motors and the mill operator will not be able to turn the crank wheel(s) manually.
- (3) By the Sherline CNC (**Metric**) v4.51: If the Power Switch\* to the Mill is in the on position, then you can use this program to operate the 'X', 'Y', 'Z', and 'A' axes.\*\* Once turned on, the control of the axis is done by motors and the mill operator will not be able to turn the crank wheel(s) manually.

\*The Power switch for the *Sherline Mill* is located on the left side of the CPU Shell. It is a DPST switch. When the switch is in the up position, a red LED and the *Sherline Mill* turn on.

\*\*Once again, even though the program can control a fourth 'A' axis, there is no motor control for one on the current *Sherline Mill*.

The *Sherline Mill* model in the Mechatronics lab is marked with metric units. This is not to say that you cannot use the second choice above, but the first of third options are preferred when working with the mill.

## IV. Setting Up the *Sherline Mill*

There are several basic steps that need to be accounted for prior to turning on the mill.

- (1) It is important that the mill is properly secured and free of obstacles. Check to ensure that the mill is bolted down to a level surface and is able to move without restriction in its entire range of axes movement.
- (2) It is important that the axes control cords are connected. Check to ensure that 'X', 'Y', and 'Z' axes cords are connected from the CPU to each axis appropriately.

There should be one cord that is left 'hanging' (the 'A' axis control cord from the CPU).

- (3) It is important that the 'Z' axis is 'unlocked' before moving it by hand or with a program. The lock for the 'Z' axis can be found on the threaded screw for the 'Z' axis movement. To check to see if the 'Z' axis is locked, try to move the 'Z' axis crank wheel before turning the power on. If the crank wheel turns freely, then the 'Z' axis is unlocked.

To lock or unlock the 'Z' axis {Please see Figure 1 for reference of part names.}:

- (1) Loosen the Flat Point Thumbscrew.
- (2) Move the Adjusting Arm to the desired position:
  - i. Locking: Move the Adjusting Arm as far left as possible so that the Brass Locking Lever is as far left as possible.
  - ii. Unlocking: Move the Adjusting Arm as far right as possible so that the Brass Locking Lever is as far right as it will go (as seen in Figures 1 and 2).
- (3) Tighten Down the Flat Point Thumbscrew.

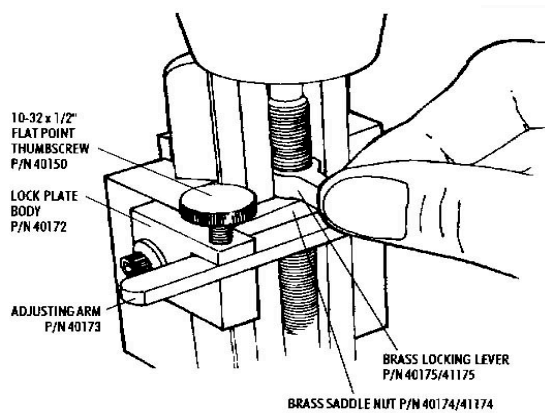


Figure 1

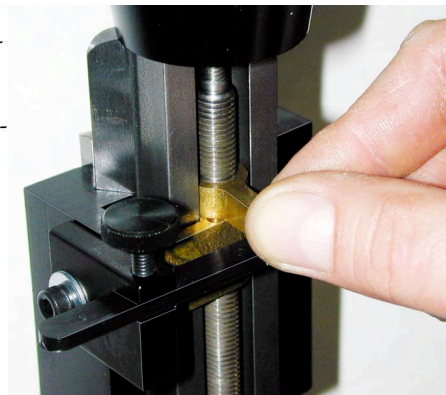


Figure 2

*For more information on the locking mechanism for the 'Z' axis, please see CNC Mill Z-axis Backlash Lock, P/N 4017Z on p34.*

## V. Sherline Mill Accessories

### A. End Mills

End Mills consist of a double sided cutting tool used to cut through material on a mill or lathe. They come in a variety of different sizes. The smaller sizes can be help by a milling collet, while the larger sizes need to be held by a End Mill Holder.

### B. End Mill Holders

End Mills Holders are used to hold End Mills of 3/8" size or larger in diameter. The End Mills are held in place with a set screw that goes into the flat ground on the shanks of the End Mills. When setting an End Mill, tighten the screw to the point where the End Mill is held, but it is not tight. Then, press the End Mill up as far as possible and tighten.

*For more information, please see the SHERLINE End Mill Holders (P/N 3079), page 37.*

### C. Flycutter

The Flycutter is intended to help machine flat surfaces. The cutter is essentially a left-handed lathe tool. It is important when working with the Flycutter to not exceed an appropriate cutting speed for the material being machined.

*For more information (including how to calculate cutting speed), please see the SHERLINE Flycutter (P/N 3052), page 39.*

### D. Milling Collet Set

The Milling Collet Set is used to hold End Mills less than 3/8" in diameter. The collets are held in the spindle with a drawbolt. Their shallow angle gives them the high clamping pressure needed to effectively cut the material being milled.

*For more information, please see the SHERLINE Milling Collet Set (P/N 3090), page 41.*

### E. Mill Tooling Plate

The Mill Tooling Plate is used as a way to protect the mill table. It slides onto the track of the 'X' axis and locks down with six screws. The Mill Tooling Plate can be modified as needed to hold down a part and comes with a hole pattern already pre-drilled. Use the predrilled holes to attach clamps such as the Step Block Hold-down Set.

*For more information, please see the SHERLINE Mill Tooling Plate (P/N 3560), page 42.*

### F. Mill Vise

The Mill Vise acts much in the same way as a normal vise, except it is designed to hold the movable jaw down as well as closed while clamping. The result is that the vise will not lose its grip when a vertical force is applied. There is both a horizontal and a vertical "V" groove in the face of the jaw plate to facilitate holding a round object securely.

*For more information, please see the SHERLINE Mill Vise (P/N 3551), page 44.*

### G. Step Block Hold-down Set

The Step Block Hold-down Set is intended to help to lock down a part before beginning to mill it. The set has two step block clamps that can be used with the step blocks to lock down two points on the object to be milled. The Step Block Hold-down Set works best with flat surfaces.

*For more information, please see the Sherline Step Block Hold-down Set (P/N 3013), page 47.*

*Please make sure you fully understand all the different tools and how and when to use them before reading further.*

## VI. Orientation of the *Sherline Mill*

The orientation of the *Sherline Mill* comes into play when using the *Sherline CNC* programs. It is important to keep the orientation in mind when writing G Code. Figure 3 shows the mill with its orientation. The arrows for each axis point in the positive direction of the axis.

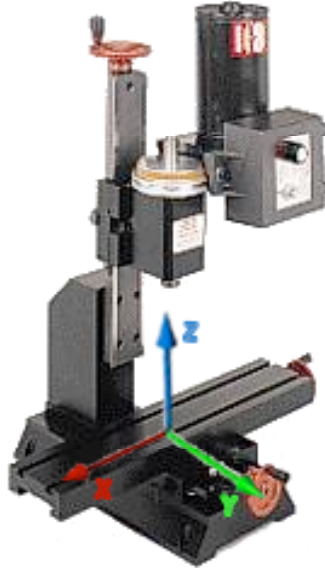


Figure 3

## VII. Limits of the *Sherline Mill*

As you write the G Code for the part to be milled, it is important to keep in mind a center or “origin” to use as a reference point throughout the design. Consider the physical restrictions of the *Sherline mill* on each axis before you begin to design. The ‘X’ axis has the most mobility of the three available axes, followed by the ‘Z’ axis, and then the ‘Y’ axis. Please see Table 1 for specifics. All of the measurements in Table 1 are in millimeters (mm).

Axis	Maximum Movement (mm)	Physical Center Point (mm)	Axis Center Point (mm)
‘X’	243.7736	121.8868	108.9900
‘Y’	130.2752	65.1376	57.50000
‘Z’	168.2450	-	-
‘A’	-	-	-

Table 1

*The above information is not exact. It is the average of a number of tests and is a good guideline but not perfect. It is reliably accurate to the tenths place.*

- The Maximum Movement (mm) given for each axis is the physical extreme that each can move along its threads (from one side to the other) before the axis can move no further.
- The Physical Center Point (mm) is taken by dividing the Maximum Movement in half. This is to say, if you were to center the mill exactly, you would be able to move a maximum of 121.8868 mm from the center in the X-direction and 65.1376 mm in the Y-direction.

- The Axis Center Point is the point on the axis where, if lined up with the axis marker, will mean that the axis is in its exact center.

The 'Z' axis has only its Maximum Movement listed for several reasons. The placement of the axis is dependent on the length of the tool being used. Each tool effects how long the Maximum Movement of the 'Z' axis *should* be before it will damage either the 'X' or 'Y' axes. The Maximum Movement listed in Table 1 is the maximum the 'Z' axis can move along its thread without a tool attached.

There is not official marker to reference where the center of the 'Z' axis is. In most cases this is not important. The main thing that needs to be determined in setting the 'Z' axis base/zero position is determining where the tool attached is in relation to the material/object to be milled.

There is a very important concept that must be realized now. While designing, it is easy to fall into the trap of thinking that you can actually move the limits given above.

The 'X' axis is limited by its surroundings as well as its Maximum Movement listed in Table 1. If the piece to be milled "hangs over the sides" of the axis, please make sure that the 'X' axis still can move the maximum movement listed in Table 1 without hitting any surrounding objects.

The 'Y' axis is limited by the size of the object to be milled much more than axis movement. If the object to be milled is larger than 69.5 mm in the Y-direction (the width of the X-Plane) or more than 26 mm in height (in the Z-direction), then the object will bind on the support for the 'Z' axis before the limit listed in Table 1 is reached. Most of the binding in the 'Y' axis occurs while traveling in the negative Y-direction.

The length of the tool in use limits the 'Z' axis. It is common to want to use more then one size tool for one object with a tool change coded in. For this reason, it is important to let go of the idea of a general and constant limit for the 'Z' axis. Instead, understand and write G Code related to the 'Z' axis appropriately for the tool being used, not the object being milled.

*Suggestions for dealing with common alignment issues and how to prevent binding can be found in Suggestions for Locking Down a Part on page 28.*

## **VIII. Centering the Sherline Mill**

Each program you write may require a different center position. It is important that you are easily able to find the exact center of every axis easily if you wish to use the center point and optimize your project layout (i.e. try to get as close to Table 1 as possible with axis limitations). There are two ways to achieve the center position in the 'X' and 'Y' axes.

### **A. Manually**

To center the *Sherline Mill* manually, turn the wheels attached to each axis motor by hand. You will know that the 'X' and 'Y' axis are centered when Point A in Figure 4 is lined up with the 'Y' axis center point (57.50 mm\*) and Point B is lines up with the 'X' axis center point (108.99 mm\*). Do not worry about centering the 'Z' axis.

*\*These values were taken directly from Table 1.*



## B. Using a Test Program

To center the *Sherline Mill* using a test program, run the “Reset X&Y Axis.ngc” program.

To run the program:

- (1) Open Sherline CNC (Metric) v4.51
- (2) Open the program titled “Reset X&Y Axis.ngc”
- (3) Load the program
- (4) Double check to ensure that the area around the mill is clean and that the mill is turned on
- (5) Execute the program

You will hear a grinding sound as each axis reaches its limit. Do not worry, the mill will not hurt itself. The program is very simple. It sends the ‘X’ and ‘Y’ axis to their limit and then moves each to their axis center point  $(X,Y) = (57.50, 108.99)$ .

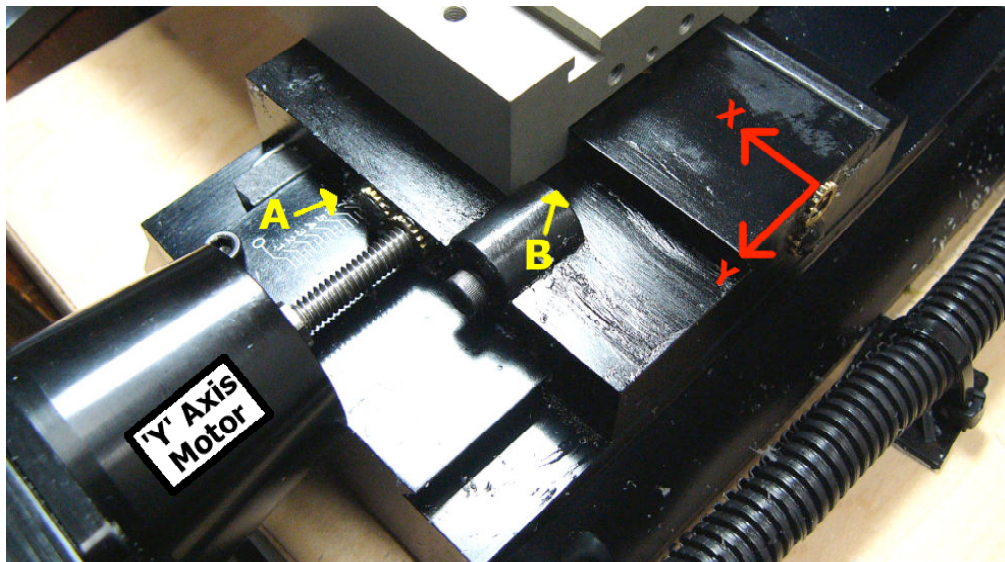


Figure 4

## IX. Operating the *Sherline Mill* Manually

To operate the *Sherline Mill* Manually, the switch on the side of the CPU Tower needs to be set to off. This switch provides the power to the motors and once it is turned on, you will no longer be able to make adjustments by turning the crank wheels attached to each access.

Operating the mill manually is fairly straightforward. With the Power to the motors off, you can freely turn the crank wheel attached to each axis and manually move the piece you are working on.

Manually operating a mill can be useful for setting the start point for a program and milling out rough details. If you plan on milling a complex part I do suggest that you write some G Code and run the program instead.

## X. Operating the *Sherline Mill* Using the Sherline CNC (Inch) v4.51 Program

For the purpose of this manual/overview, I will not discuss the specifics of the Sherline CNC (Inch) Program. The *Sherline Mill* you will be using should be operated manually or with the Sherline CNC (Metric) v4.51 program.

The main difference between the two is the system of units implemented in each respectively. It is very easy to go from one program to the other. Almost any G Code you write will work on both programs.

If you write G Code in one program and decide to use the other later, please make sure to adjust the movement specifics. Keep in mind that there is a big difference from one centimeter to one inch.

## XI. Operating the *Sherline Mill* Using the Sherline CNC (Metric) v4.51 Program

Operating the *Sherline Mill* with the CNC Programs has one of two modes: Manual and Automatic. For both methods, start by starting the Sherline CNC (Metric) v4.51 Program.

The mill has the ability to move along its 'X', 'Y', and 'Z' axes. It is important to keep the limits of the movement for each in mind.

### A. Starting the Program

- (1) Make sure that the toggle switch on the side of the CPU is off. (The red LED next to the switch will glow when the switch is set to On.)
- (2) Turn on the CPU.
- (3) Once the CPU starts up, your screen should be similar to the Figure 5.

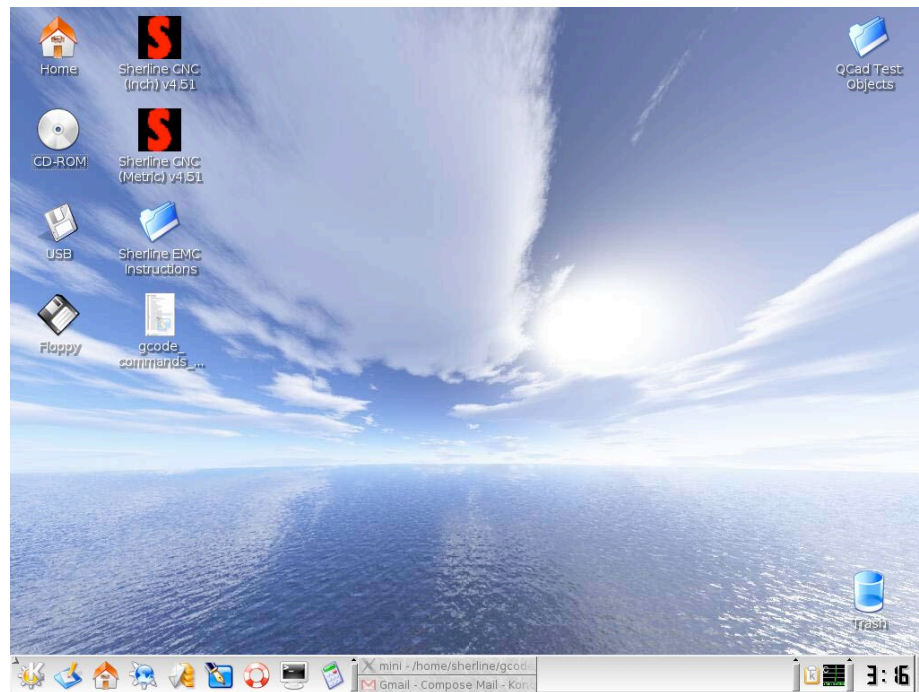


Figure 5

- (4) Left Click one time on the Sherline CNC (Metric) v4.51 icon. Remember, this is a Linux CPU, double clicking will open two copies of the program.

After the program opens, your screen should look similar to Figure 6.

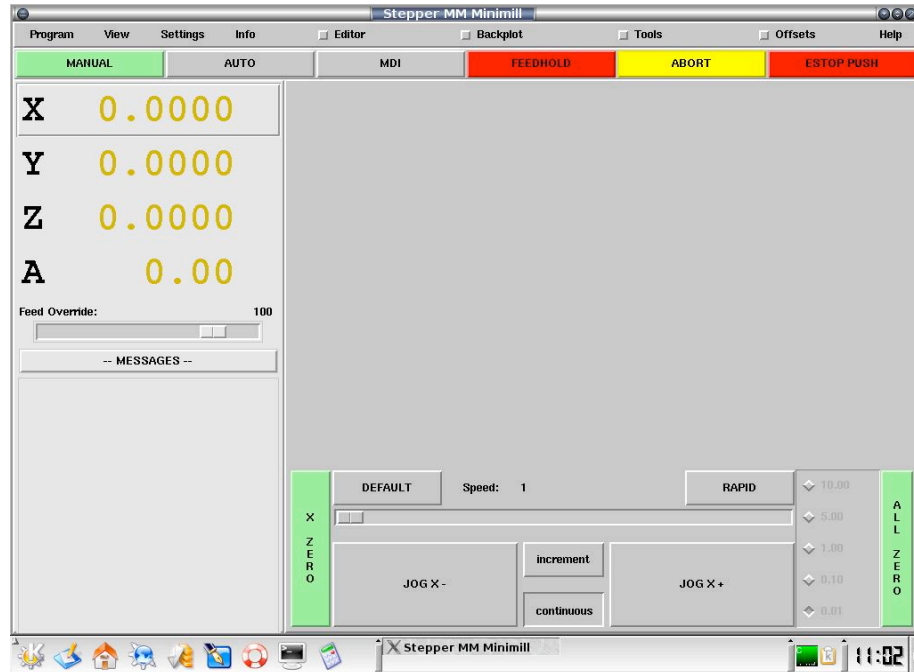


Figure 6

## B. Program Basics

The ‘X,’ ‘Y,’ ‘Z,’ & ‘A’ axis are shown on the left side of the screen (see Figure 6). The numbers to the right of them shows what the program *believes* each axis is positioned. If the mill is operated manually or if it binds (tries to move further on it axis than its physical limits allow) these numbers will not be accurate. They are indicators of where the program has virtually moved axis positions. The Feedrate Override slide is located just beneath the axis position information. Adjust this in both Manual and Automatic Mode to override the speed at which the axes move.

Any error message that occurs can be seen directly below the “---Messages---” button on the left side of the screen. Errors show up if you try to load a program while in manual mode or there are errors in your code. To clear the list of errors, left click on the “---Messages---” button.

There are several different screens that can be viewed from within the program. These include: Editor, Backplot, Tools, Offsets, and Help. To view one or several, click on the appropriate button for each. The buttons can be seen on the top of the screen (see Figure 6). Left click on a grey button to turn it red and make the screen viewable. Once multiple screens are open, you can right click anywhere on the page to toggle between screens. The screens will toggle in the order of which they are selected.

## C. Different Screen Information

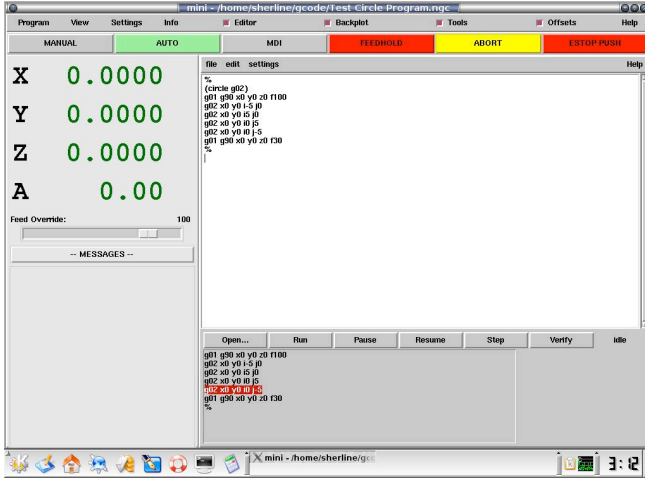


Figure 7

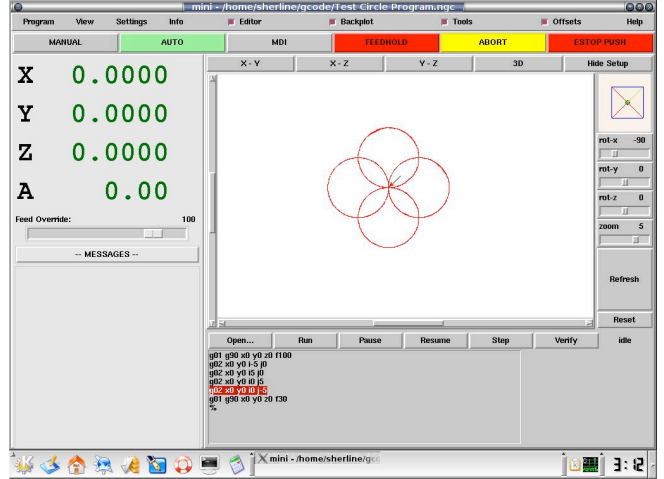


Figure 8

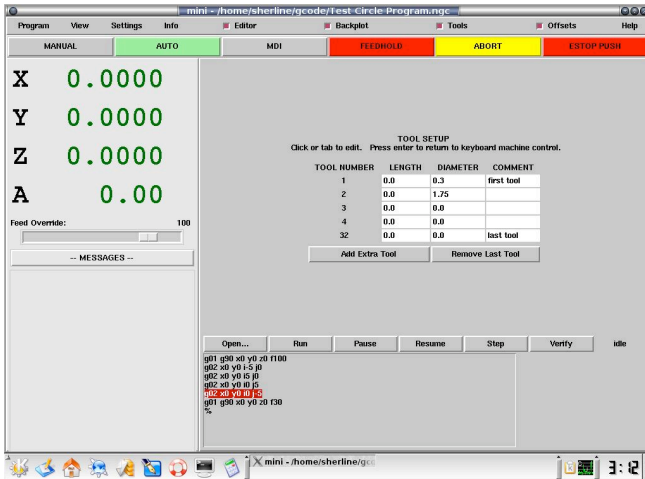


Figure 9

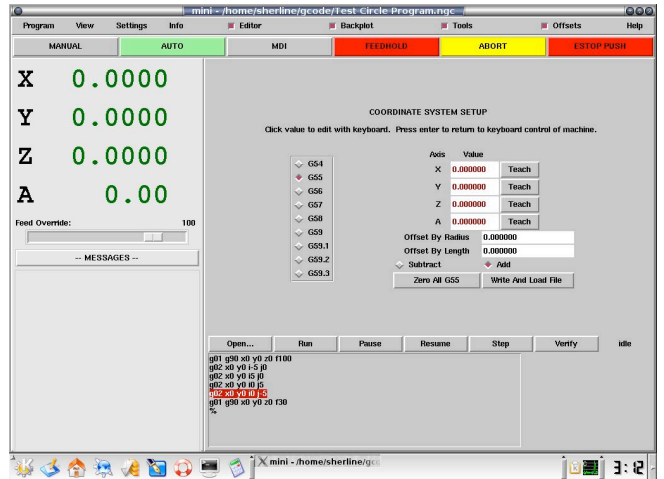


Figure 10

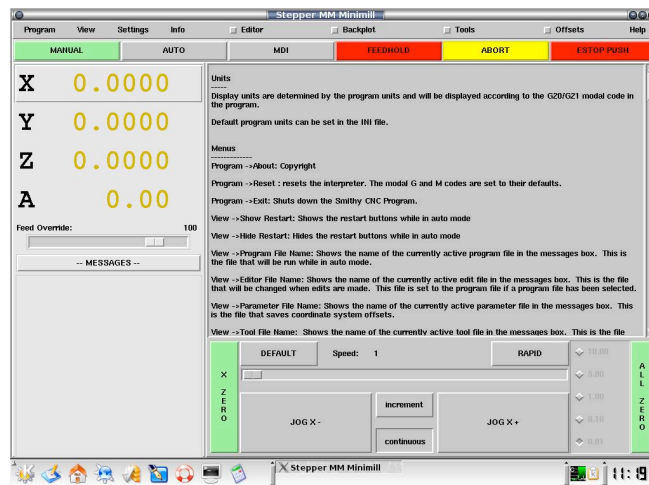


Figure 11

Please reference the above figures when reviewing their descriptions below.

(1) *Editor Screen*

The Editor Screen (see Figure 7) consists of a primitive text editor that is linked to the interpreter. You can write, open, and load G Code files in the Editor.

(2) *Backplot Screen*

The Backplot Screen (see Figure 8) consists of a display that shows the total motion of the program/mill. When operating in Automatic mode (running a program), the color of the display shows the kind of motion. Green is rapid, black is feedrate, red is clockwise rotation viewed from positive (g02) and blue is counterclockwise (g03).

You can click change the current view of the display in several ways. If you click on the “X-Y,” “X-Z,” or “Y-Z” buttons on the screen, you will see the display from the selected view. You can also rotate and zoom the view by moving the sliding tabs on the right side of the screen and clicking refresh. The program will automatically refresh what has been drawn, clicking the refresh button refreshes the view specifics (angle and zoom).

If you would like to reset the display completely, click the reset button on the right.

(3) *Tools Screen*

The Tools Screen (see 9) is used when setting tool length and diameter for the tools used in a program.

(4) *Offsets Screen*

The Offsets Screen (see Figure 10) is used to set the values of any of the coordinate offset systems. Use this screen to teach current locations or enter offset values.

(5) *Help Screen*

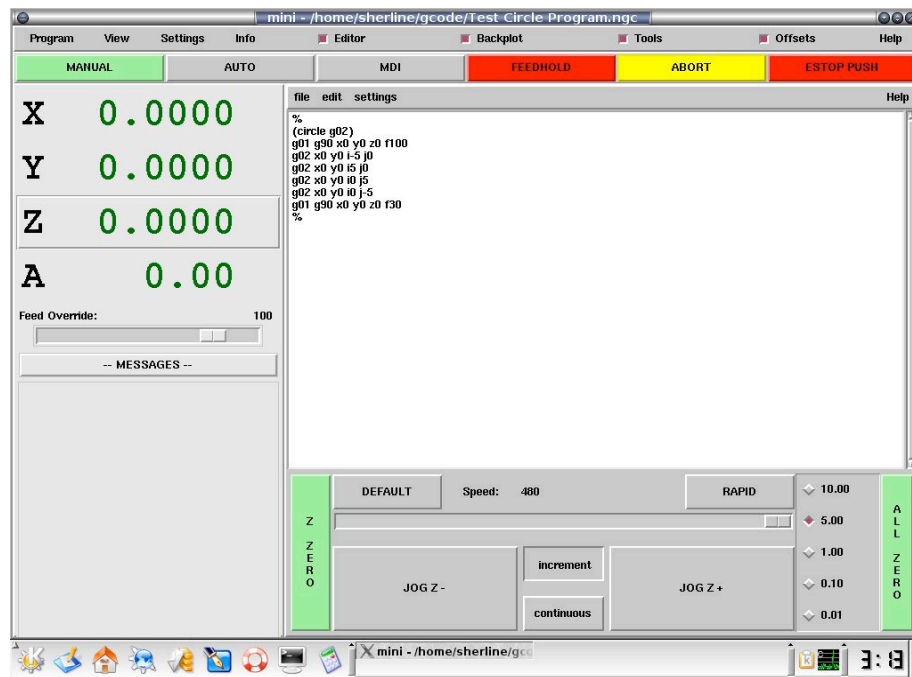
The Help Screen (see Figure 11) contains a list of help for many of the different programs used in the *Sherline* CNC v4.51 programs.

D. Operating In Manual Mode

Operating in Manual Mode is similar to operating the mill manually. For both, it is up to the operator to pay careful attention and control the mill throughout the entire milling process (instead of a CPU doing it for them).

The key difference is that, when in Manual Mode, the operator tells the CPU to turn the crank wheels on each axis instead of doing so by hand. Operating in Manual Mode allows the operator to keep a consistent feed-rate and to allows for accuracy in movement up to the hundredths place.

To enter Manual Mode, Click on the Manual Button on the upper left side of the screen. It will appear green once it has been selected. (Fun fact: When the program opens, it defaults to Manual Mode.)



**Figure 12**

Figure 12 shows the Manual Display with the Editor Screen open. Changes can still be made to the Editor Screen, but no program can be loaded while in Manual Mode.

Once in manual mode there are a variety of options/controls to choose from.

- Moving the long slider on the bottom of the screen controls the feedrate speed. Click and drag the slider to change the feedrate speed. If you wish to quickly move from one end of the feedrate spectrum to the other, click on either the “DEFAULT” or “RAPID” buttons. The “DEFAULT” button sets the feedrate to one, and the “RAPID” button sets the feedrate to its maximum.
- You can select the axis you wish to move in two ways: (1) Click on the axis you wish to move on the left side of the screen, or (2) Type the name of the axis you wish to move on the keyboard.
- You can choose to increment or continuously move an axis by pressing either “increment” or “continuous” on the bottom of the screen. If you are incrementing the axis, you can then select from the list of five values (in millimeters) to increment by on the bottom right of the screen.
- Once you have selected an axis and a mode to move the axis by, pressing the “JOG” buttons can move the axis. If you are incrementing the axis, the axis will move the pre-selected distance and stop. If you are continuously moving the axis, the axis will move as long as the “JOG” button is depressed.
- Instead of clicking on the “JOG” buttons, you can use keyboard shortcuts to move each axis. The axis will move according to the same mode selections

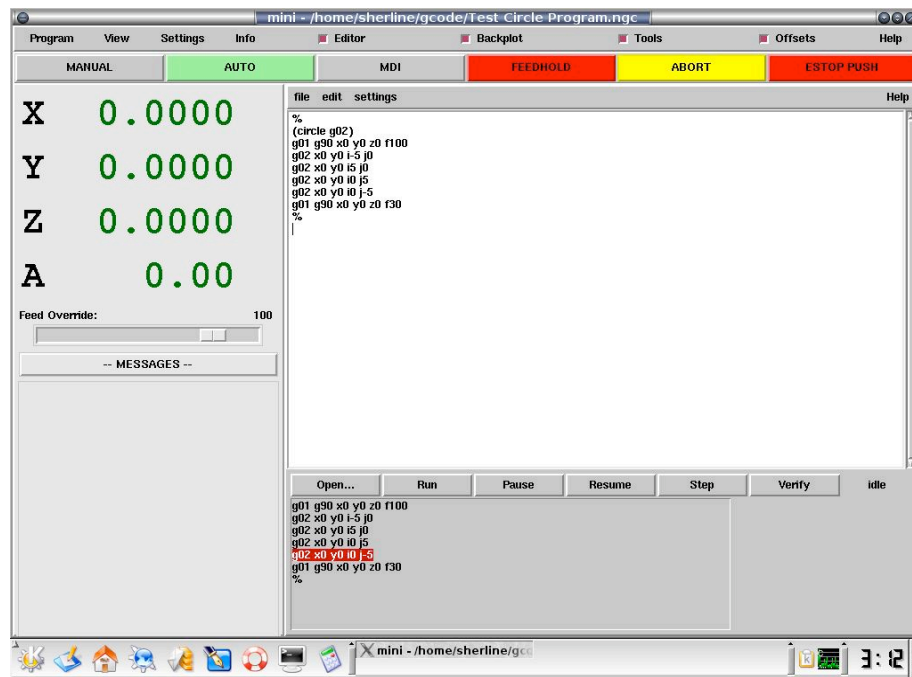
(increment vs continuous and feedrate speed) when using keyboard shortcut keys as when pressing the “JOG” buttons.

- ‘X’ axis - Left arrow moves axis in negative direction, Right arrow moves axis in positive direction.
- ‘Y’ axis - Down arrow moves axis in negative direction, Up arrow moves axis in positive direction.
- ‘Z’ Axis - Page Down key moves axis in negative direction, Page Up key moves axis in positive direction.

#### E. Operating In Automatic Mode

Operating in Automatic Mode is where G Code comes into the process of milling. In Automatic mode you write programs and load them into the program.

To enter Automatic Mode, Click on the Auto Button on the upper left side of the screen. It will appear green once it has been selected.



**Figure 13**

*Figure 13 shows the Automatic Display with the Editor Screen open. Changes can be made to the Editor Screen and then loaded.*

Once in Automatic mode you can create and run G Code programs.

Creating, Opening, Adjusting, Saving, Loading to G Code programs you wish to run is all done in the Editor Screen.

- To create a new program go to File>>New or type “Ctrl+n.”
- To open a file go to File>>Open and select the program you wish to open or type “Ctrl+o.”

- To save a program go to File>>Save or type “Ctrl+s.” If you wish to save the program under a different name go to File>>Save As.
- To Load a program go to File>>Save and Load. If you see your program copied to the clipboard on the bottom of the screen you know that the file was loaded successfully
- To run a program once it has been loaded, click on the “Run” button on the bottom of the screen.

There are a few key things to note when working in Automatic Mode.

- If you wish to pause a program at any point in time, press the “Pause” button. To resume press the “Resume” button.
- Open the Blackplot Screen to view the status of the execution of the program.
- If you realize that you wish to completely halt the program, press the yellow “ABORT” button on the top of the screen. If you abort a program, a you cannot rerun the program until it has been reloaded.
- Following the completion of a program you can rerun it immediately by pressing the “Run” button (you do not need to reload the program).
- You can edit the program you are currently running, open a different program, or start a new one at any point during the execution of the program. You can save as well. The only thing you cannot do is load the program while a program is still being executed. Trying to do so will activate a prompt making sure you really want to do so. Saying yes halts execution of the current program.

## **XII. Overview of G Code**

### **A. Syntax**

The layout of G Code is rather simplistic and similar to that of Assembly. The first and last lines are marked by a percent sign {'%'}. Comments in the code are noted in parentheses {'(...)'} and can be placed either at the end of a line or on a separate line.

A line of code consists of the executable information in the program. The program executes in a linear fashion. That is, line three will always be executed before line five and after line two.

There is no difference between g00 and G00 or M05 and M5. The compiler for the program ignores leading zeros for code words and does not care if you use all lowercase letters, all capital letters, or a mix of both.

Line numbers are optional, but need to be entered by you line by line. At the beginning of each line you would like to number place a letter 'N' followed by the current line number.

Each line of executable code consists of one or more "words." A word consists of a letter followed by a number (or something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, "G1 X3" is a valid line of code with two words. "G1" is a command meaning "move in a straight line at the



programmed feed rate," and "X3" provides an argument value (the value of X should be 3 at the end of the move). Most RS274/NGC commands start with either G or M (for miscellaneous). The words for these commands are called "G codes" and "M codes."

## B. Tables

The complete listing of the G Code and M Code and their modal groups (as supported by the *Sherline Mill*) can be found in Appendix A. The tables list all the G Code and M Code words and what their effect is.

Pay careful attention to the Modal Group listings. You cannot have two words from the same modal group on the same line of code. For example, you could not write G00 G01...F10 as part of a line of code. G00 tells the program to make the next cut at the maximum feedrate, while G01 tells the program to make the next cut with the feedrate at the speed of 10.

## C. Example G Code

There are numerous example of G Code out in books and on the internet. If you do not know how to write the code for a specific part you will most likely be able to find some guidance online. Once you realize the simplicity of G Code you should not have any problems.

Essentially, all of G Code breaks down into drawing lines and arcs. Rather than simply reiterating what the tables in Appendix A state, lets see some working programs. It is highly suggested that you write out these programs yourself to help to learn the workings of writing G Code

### (1) *A Basic Circle Program*

Milling a circle is very simple and is nearly identical to milling out an arc. Keep in mind that a circle is just an arc that is drawn for a full 360 degrees.

G02 and G03 signify milling a circle. G02 will initiate an clockwise circle, and G03 will initiate a counterclockwise circle.

There are two key things to note with milling circles or arcs.

- (1) The circle will be milled starting at the mills current location and its starting point lies on the arc of the circle/arc.
- (2) You cannot mill out a circle without a feedrate. There are two ways to set the feedrate.
  - (a) Have `g01...f#` be on a line prior to milling a circle (Where # is the feedrate value.)
  - (b) Write a feedrate value at the end of the line of code for a circle (i.e. `g0A xB yZ iC jD f#`).

The line of code for both a circle and an arc in the XY plane is as follows:

- `g0A xB yZ iC jD {where the bold letters are replaced with the appropriate values for the particular circle or arc}`.

- **A** signifies the direction of the circle (clockwise or counterclockwise).
- **B** and **C** signify the end point value of the circle or arc. Here is where milling a circle or an arc differentiates. If you wish to draw a full circle, have your **B** and **C** values be equal to your starting point. For an arc, set the values for **B** and **C** to where you would like the arc to end.
- **C** and **D** signify the center of the circle or arc. The values are not the point of which you wish the center to be, but instead a reference from the current point. For instance, if you are at  $(X,Y) = (10,10)$  and you wish to have a circle drawn around  $(X,Y) = (0,0)$  your **C** and **D** values should both be -10.

```
%
N02 (circle g02)
N03 g01 g90 x0 y0 z0 f2
N04 g02 x0 y0 i-.500 j0
N05 g01 g90 x0 y0 z0 f2
%
```

N01) % – Always start a new program with a % sign. This informs our robot that a program is coming in.

N02) (g02 circle program) – A comment for the program telling future users that this program will draw a circle with the g02 command, meaning it will draw it clockwise.

N03) g01 g90 x0 y0 z0 f2 – The mill will move in a straight line in absolute distance mode to the point  $(X,Y,Z) = (0,0,0)$  with a feedrate of 2. Essentially, this will set the mill to a home position of  $(0,0,0)$  if it is not there already before starting the program.

N04) g02 x0 y0 z0 i-.5 j0 – This line is the code that is going to actually produce the x y movements to create that circle. All the previous commands are still in effect except the g02 is replacing the g01.

N05) g01 g90 x0 y0 z0 f2 – It is a good idea to add a line of code to return to the home position. If you are milling multiple copies of the same piece, resetting to the home position for the program makes starting the next piece/rerunning the program simpler. Another consideration is that even after a program terminates, the last G Code words stays in effect until they are overwritten by another line of code. In this case it would have left the

control in a circle command if we didn't add this line of code. That wouldn't be a good practice.

N06) % – This signals the end of a program.

## (2) *A Circle Program with Offsets*

Adding offsets to a program helps to ensure that the piece being milled will be exactly the size you want and that different length pieces used in one program will still have the same reference points. However, before we begin to write a program with offsets, we need to understand the code words related to setting and taking off offsets.

The length and diameter of each individual tool is stored in a file called “Tools,” located along the top row of the control panel. The tool length is usually called up in the first z-axis move using each individual tool by entering a g43 h1,2,3,--. The g43 tells the computer to use the tool length (h) stored in the line (1,2,3,---) of the tool data file. A g49 is entered when the z-axis is brought back to its home position to cancel out the tool length compensation that was called up when you first started using the tool.

You need to enter in the value for the offsets manually into the Tools Screen. This means that you need to decide what the base length for a tool should be and then carefully measure and record the difference with the tools you are using.

Now for the actual program. Notice that the offsets ensure that the length of each tool is appropriately accounted for.

```
%  
N02 (circle using g40, g41, g42)  
N03 g00 g90 g40 x1 y1 z1  
N04 g42 d1 x.5 y1  
N05 x0  
N06 z0  
N07 g01 z-.3 f10  
N08 g03 x0 y1 i0 j-.5  
N09 g00 z1  
N10 g40 x1 y1 z1  
%
```

N01) % – Signals the start of a program.

N02) (circle using g40, g41, g42) – Comment that lists the name of the program.

N03) g00 g90 g40 x1 y1 z1 – Set the mill to the home position./Cancel any previous Cutter Compensation.

N04) g42 d1 x.5 y1 – As the mill approaches the part to be milled, tool compensation is entered. “d1” defines the offset as tool 1 as the mill moves to the new g42 position.

N05) x0 – There is no need to write g00 here. G00 has not been overwritten since it was used in N03. This line sends the mill to position ‘X’ = 0 while keeping the Y and Z positions constant.

N06) z0 – This line brings the mill down from ‘Z’ = 1 to ‘Z’ = 0. For the purpose of this program, ‘Z’ = 0 is level with the surface of the object to be milled.

N07) g01 z-3 f10 – Moves the mill to a depth of 3 millimeters into the piece at a feedrate of 10.

N08) g03 x0 y1 i0 j-.5 – A circle is cut counterclockwise at the feedrate of 10.

N09) g00 z1 – The mill moves to just above the surface of the object after completing its cut.

N10) g40 x1 y1 z1 – Sets the mill in the home position.

N11) % – Signifies the end of a program.

### (3) *Rectangular Box with Cutter Compensation*

To draw a straight lined object such as a box there is not a specific G Code word that will do it automatically. Instead, simply tell the mill to trace along the connecting end points of the box to draw the rectangle or square

```
%  
N02 (cut rectangle with cutter comp)  
N01 g00 g90 g40 x0 y0 z1  
N02 x5 y10  
N03 z-2  
N04 g41 d1 x.5  
N05 g01 y-10 f10  
N06 x-5  
N07 y10  
N08 x5  
N09 z-4  
N10 g01 y-10 f10  
N11 x-5  
N12 y10  
N13 x5  
N14 g00 g40 z1  
N15 x0 y0 z1  
%
```

N01) % – Signifies Start of Program.

N02) (cut rectangle with cutter comp) – Comment that lists name of program.

N01) g00 g90 g40 x0 y0 z1 – Set starting point for program/Cancel and previous Cutter Compensation.

N02) x5 y10 – Move to upper right corner of box.

N03) z-2 – Mill down 2 millimeters for first pass on a 4 millimeter box.

N04) g41 d1 x.5 – Set Cutter Compensation.

N05) g01 y-10 f10 – Move to lower right corner of box.

N06) x-5 – Move to lower left corner of box. (Note: You do not need to write g01 again. Until you write g00 again, the mill will move with a feedrate of 10.)

N07) y10 – Move to upper left corner of box.

N08) x5 – Move to upper right corner of box.

N09) z-4 – Mill down 2 more millimeters for second pass on a 4 millimeter box.

N10) g01 y-10 f10 – Move to lower right corner of box.

N11) x-5 – Move to lower left corner of box.

N12) y10 – Move to upper left corner of box.

N13) x5 – Move to upper right corner of box.

N14) g00 g40 z1 – Cancel Cutter Compensation and move mill above level of object.

N15) x0 y0 z1 – Reset to starting position.

N16) % – Signifies end of program.

#### (4) *Pulley Program*

Milling out a Pulley is very good practice in working with all the fundamentals we have gone over so far. The following is the program in its entirety. I will leave the commenting to you. It is important that you can fully understand this program without my own comments. If you can do that, you are ready to begin writing your own G Code.

```

%
N02 (pulley program)
N03 g00 g40 g90 x0 y1 z0
N04 x-.500
N05 g41 d1 y.500
N06 g01 y0 f12
N07 g02 x-1.0625 y-.4961 i-.500 j0
N08 g01 x-3.0313 y-.2480
N09 g02 x-3.0313 y.2480 i.0313 j.2480
N10 g01 x-1.0625 y.4961
N11 g02 x-.500 y0 i.0625 j-.4961
N12 g00 g40 x0
N13 x0 y1 z0
%
```

#### D. Other Useful G Codes

There are many different code words for G Code. There are a few that I would like to highlight as you may find them very useful when writing your own G Code.

- **G92**

This will reset your home position values. This command is used to reload the registers in the control system with desired values and could be handy if you end up using a home position that isn't all zeros.

No movement of the tool along any of the axes will occur; it will simply reset the values in the axis readout window to the home position values or whatever values you enter. This is used when, for example, you shut down the computer at the end of the day with the machine in the home position. When you turn it on the next time, the axis values may read all zeros or some other numbers for x, y, z and a on the coordinate part of EMC screen instead of your home position. By placing the home position values after the g92 it will reset your axis readouts to the home position (no movement of tool along any of the axes will occur) when you hit [Run] and begin the program.

*Example*

```

%
g92 x1 y0 z2
g00 g90 g20 g40 z2 x1 y0 (home position)
g89 x2 y1 z0.50 r1.5 p.2 f5
g00 g80 x1 y0
%
```

**g92 x1 y0 z2** – Brings the coordinates on the axis screen in EMC from any number that happens to be currently displayed to the (1, 0, 2) home position without moving any axes (stepper motors) on the machine. Remember you must make sure your machine is actually located in the home position to make this

effective. It does not physically home the machine, it simply resets the values to whatever coordinates you enter after the g92.

- **G81**

This command is intended for drilling, and because x, y movements usually take place with the drill located well above the part to avoid accidental collisions with fixtures, the most efficient method is to lower the drill in rapid (g00) to the starting point.

However, the computer needs to know the drill starting point to accomplish this. This second z-axis position is given by entering an r-command defining an absolute position in the same line of code that contains the g-command that requires it. If the depth of the hole exceeds 1.5 times the diameter of the drill you should use the g83 command. Also note that we are using the absolute mode.

*Example (Drilling One Hole)*

```
%  
N02 g90 g40 g20 g00 g80 z2  
N03 x0 y0  
N04 g81 x2 y1 z-0.50 r0.010 f3  
N05 g00 g80 x0 y0  
%
```

N01) % – Signifies start of program. (Consider the top of your part to be the 0 position.)

N02) g90 g40 g20 g00 g80 z2 – It's a good idea to get the z-axis up out of the way before moving the x- or the y-axis. For example, consider what happens if you start the program from a different position than you intended and the drill hits the fixture on the move to its home position.

N03) x0 y0 – Move the mill to position (X,Y) = (0,0).

N04) g81 x2 y1 z-0.50 r0.010 f3 – This line of code contains all the information to put the g81 canned cycle into effect, and you'll be able to repeat it each time a new x, y position is given.

N05) g00 g80 x0 y0 – When you have completed drilling all your holes, a g80 is entered to cancel the canned g-cycle command.

N06) % – Signifies the end of the program.

- **G83**

This command is intended for deep drilling because you'll break the drill if you don't retract it periodically. When you retract the drill it allows the chips to fly off and lets cutting fluid wet the bit and the work before they gall or weld themselves to the drill bit. The general rule when using a CNC machine is to have the drill cut no deeper than its diameter without retracting it for the above reasons. This is called peck drilling.

The peck drilling cycle should also be used when center drills are used to create starting points for your drills. Center drills twist off very easily because their drilling section is not relieved as much as a drill is. This is done to force the drill to seek center; however, they become very vulnerable to breakage. The z-axis should be retracted when center drill's depth has reached 60% of the needed depth. Note that drills will wander by a far greater amount than you can imagine (especially if they are old and dull) if a center or spotting drill isn't used to develop a starting point. I prefer spotting drills (short drills with a 90 degree point that cuts to center) over center drills because there is no small point to twist off. They also have a proper chamfer angle and a faster machine operation.

This cycle takes a q value given as an increment value. This represents the increment of movement along the z-axis before the drill is retracted to clear the chips.

1. Move the z-axis only at the current feed rate downward by delta or to the z-position, whichever is less deep.
2. Retract at traverse rate to clear z
3. Repeat steps above until the z-position is reached.
4. Retract the z-axis at traverse rate to clear z.

*Example* (Using a 1/4" drill and putting 3-holes in 1.00" deep)

```
%  
g00 g40 g20 g90 g80 z2  
x0 y0  
g83 x1 y1 z-1 r0.010 q.25 f2  
x2  
x3  
g00 g80 x0 y0  
%
```



### **XIII. G Code Writing Suggestions**

When writing G Code there are a few things to keep in mind.

#### **(1) Draw First**

Before writing a single line of G Code, you should first have fully dimensioned drawing of what you want to make done. Doing so makes writing the code much faster. First, you can visualize what you are making. Secondly, the important points on the piece should be marked so that you don't need to do any math once you start to write your code.

It would be well worth it to use AutoCAD or another CAD program to draw out your piece. If you use a CAD program, you can be sure that your values are accurate for your piece before you write your code.

There is software on the Web that will translate CAD programs into G Code. However, from my experience in working with the *Sherline Mill*, I have not found one that is functional for the G Code accepted by the mill.

#### **(2) Set a Zero Point**

When you write your code you should reference all values from a zero point that is on your piece to be milled. I highly suggest choosing the exact center (or as close to exact center as you can get) of the piece. The mill has very limited mobility when it comes to milling larger parts.

CAD programs come in handy here. You can mark your reference point on the diagram and take all your measurements from there.

I have found the following three steps dramatically improve my consistency when milling and reduce the chance of binding:

- Center the 'X' and 'Y' axis by hand or with a program to their exact center.
- Know where the center point on the piece you will be milling is. Write your code referenced from this center point.
- Center the piece in the 'X' direction on the 'X' axis so that the 'Z' axis is directly in line with the 'X' center point and lock down the piece.
- Adjust the 'Y' axis until the 'Z' axis points directly above the center point of your piece.

If you do the above, and have written your code to reference from the middle of the piece you are making, you will increase the range of your mobility in the 'X' and 'Y' axis and reduce the chance of binding.

When it comes to choosing a zero point for the 'Z' axis, I highly suggest you choose the highest point on the surface of your piece as the zero point. Doing so allows for easy set up of the mill and adjustments later on while milling.

**(3) Lock Down Your Piece**

One of the hardest lessons to learn is how to lock down your piece. Locking down your piece well prevents the mill from pushing it around and misaligning it while it is milling.

**(4) Test Axis Movement**

While it may seem to be tedious to center your mill and then test axis movement, it is well worth the effort. Once you know exactly where the piece will be on the mill, turn the 'X' and 'Y' axis to their maximum +/- positions and check to see that the piece will not bind/reach its limit in a direction before it should. The most common problems occur in the negative 'Y' direction and binding on the 'Z' axis support (if the piece is too tall).

Once you have ensured that your piece will not bind, you can be sure that, if the G Code is correct, there should be minimal problems when you mill

**(5) Run a CPU Test First**

Please do not learn the hard way! Before you ever turn on the mill, make sure you have done a dry run through. To do so, save and load your program as you would if you were to run the mill, and run the mill with the power to the mill (switch on the side of the CPU box) off. You can view the result on the Backplot Screen.

Doing so allows you to make sure that you don't accidentally slice through your piece when you move to a new area and that the piece will turn out as you expect it to.

You don't need to do a dry run in real time. You can change your feedrates to faster rates and run your test. Remember to reset your feedrate speeds if you do this! An easy way to do this is to just add zeros to the end of all your feedrate numbers. Then you simply need to go back through and delete the extra zeros, reload the program, and mill your part.

**(6) Override Your Feedrate**

Do not be afraid to manually override your feedrate if you feel that the mill is going too fast or too slowly. There is a limit to how much you can override it, but generally it should be enough.

**(7) Add Coolant**

When you mill it is very likely that your piece will heat up. It is important to use coolant, especially when milling metals. Adding coolant lubricates the bit so that you do not cause too much friction and helps to remove the pieces/debris that have been milled off.

Do not be afraid to add coolant. Too much is better than not enough.

**(8) Build Pauses into the Program/Changing Bits**

You can add in m00 (pause) statements to let the mill rest or to change bits. When you wish to change a bit for a different cut make sure that you bring the 'Z' axis high enough to remove the current bit and add the new one before pausing the program.

If you do not like dealing with determining your tool length offsets prior to starting to mill, you can pause the program to check the zero point after changing bits. Once you have changed your bit, the first action should be to move the 'Z' axis downward to the piece. If you pause the program when it is at zero you can check to see if the bit is level with the piece.

The program only thinks it knows where the mill is. If you change bits you change the 'Z' axis reference point. Once the program is paused, turn the mill off and manually turn the crank wheel for the 'Z' axis until the bit is level with your zero point (hopefully that means level with the top of your piece).

***NOTE:** Do not switch to Manual Mode to adjust the 'Z' axis. Doing so forces you to reload the program and start the process all over again. \*Make sure that you turn the mill back on prior to resuming the program or you will need to start the piece over.*

To resume the program, simply press the "Resume" button on the bottom of the screen. (Pressing the "Run" button will prompt an error message and won't resume the milling process but will not cause any damage.)

**(9) Determining How Far Down to Cut**

How far down you can mill in one pass depends greatly on the material. In general you don't want to go more than 3x's the diameter of the bit (since for most bits this is the length of the cutting space on the bit). This is not always the case. With light plastics you can follow this rule, but with aluminum you should not do more than 3 millimeters in one pass.

**(10) Drilling Holes**

If you are drilling holes, please use a drill bit. While the cutting bits used by the mill will function as a drill bit, doing so quickly wears down the bit.

**(11) Comment Your Code**

This may seem a bit obvious and redundant to those who have coded in other languages, but realize that it is just as important to comment your G Code as any other code. When someone looks at your code they may be able to see what each G Code word means, but the 'X' and 'Y' values will not help to visualize what part of the piece is being milled and why.

#### **XIV. Suggestions for Locking Down a Part**

One of the hardest things to learn when milling and the largest time sync is how to lock down a piece to be milled. The more complex the part, the more difficult it is to lock down.

When locking down a part keep the following in mind:

- Use the Tooling Plate whenever possible. The Tooling Plate has uniformly drilled holes that make locking down easier. It also helps to prevent unintended damage to the 'X' axis if you drill or mill down too far.
- When you attach the Tooling Plate or any other locking tool to the 'X' axis, only attach the piece with a hand tightening. Over-tightening will damage the 'X' axis and make it harder to slide pieces on and off the 'X' axis.
- If possible, put a scrap piece of plastic or wood beneath what you plan to cut out to help ensure that you won't damage the 'X' axis.
- Be careful with where you lock down the piece. You want to be able to ensure that the piece won't slip (even as it completes its last cut) and that how you lock down does not cause the piece to bind.
- If you are drilling holes, drill them first and place screws or small rods into the holes to help to prevent slipping. This is especially helpful when cutting out a full shape (prevents the piece from slipping on the last pass).
- If you will be milling out multiple copies of the same piece, consider spending the time to create a template that allows for quick change of material.
- Experiment with how to lock down a piece. There are several *Sherline* pieces that will help, but you should not be restricted to using just them.
- If at all possible, put braces along the sides of the material you lock down. The mill has a lot of torque/force. It will easily force your piece to slip (especially when using coolant) if it is not braced on the sides.
- Only lock down your piece to the 'X' axis. If you lock down the piece to the table or other axis bad things will happen.

#### **XV. Feedrate vs Cutting Rate**

When you mill out a part there are two very important things that need to be balanced at all times: feedrate and cutting rate. Feedrate is the rate of which you feed the material to the mill. Cutting Rate is the speed at which the bit turns.

Feedrate is controlled by your G Code (g01...f#). If you would like to change your feedrate once a program is running, adjust the Feedrate Override on the left side of the screen. Generally, a feedrate of 10 works for Aluminum and 25 for plastic (use these numbers as a reference point, but not a rule).

Cutting Rate is controlled by a knob on the 'Z' axis and can be adjusted at any time. Setting Cutting Rate is more subjective than setting feedrate. Start slowly and increase the speed as the mill is running. At an ideal speed, the chips that come off are nice and long (rather than many tiny pieces).

If your feedrate is too fast or your cutting rate is too slow or too fast, you will put unneeded stress on your bit. The best advice is to watch the bit and set both so that the bit is not bending and the chips coming off are of a good size.

## **XVI. Other Resources**

If you would like to read further on the *Sherline Mill* or read further suggestions for writing G Code, please see the following:

- On the Desktop of the CPU used with the *Sherline Mill* there is a file titled “gcode commands.” If you would like a quick reference of the tables in Appendix A, please go to this file.
- On the Desktop of the CPU used with the *Sherline Mill* there is folder titled “Sherline EMC Instructions.” In this folder there are different files from *Sherline* that go over more examples of G Code and other frequently asked questions.
- If you would like to read up more on *Sherline* and their products, please visit their website at: <http://sherline.com/>.
- You can find many different resources on the Internet by simply Goggle-ing what you are having trouble with.



Appendix A:  
G & M Code  
Reference Tables



## G Code List

<b>g00</b>	Rapid positioning.
<b>g01</b>	Linear interpolation.
<b>g02</b>	Circular/helical interpolation (clockwise).
<b>g03</b>	Circular/helical interpolation (counterclockwise).
<b>g04</b>	Dwell.
<b>g10</b>	Coordinate system origin setting.
<b>g17</b>	XY plane selection.
<b>g18</b>	XZ plane selection.
<b>g19</b>	YZ plane selection.
<b>g20</b>	Inch system selection.
<b>g21</b>	Millimeter system selection.
<b>g40</b>	Cancel cutter diameter compensation.
<b>g41</b>	Start cutter diameter comp. left.
<b>g42</b>	Start cutter diameter comp. right.
<b>g43</b>	Tool length offset (plus).
<b>g49</b>	Cancel tool length offset.
<b>g53</b>	Motion in machine coordinate system.
<b>g54</b>	Use preset work coordinate system 1.
<b>g55</b>	Use preset work coordinate system 2.
<b>g56</b>	Use preset work coordinate system 3.
<b>g57</b>	Use preset work coordinate system 4.
<b>g58</b>	Use preset work coordinate system 5.
<b>g59</b>	Use preset work coordinate system 6.
<b>g59.1</b>	Use preset work coordinate system 7.
<b>g59.2</b>	Use preset work coordinate system 8.
<b>g59.3</b>	Use preset work coordinate system 9.
<b>g80</b>	Cancel motion mode (includes canned).
<b>g81</b>	Drilling canned cycle.
<b>g82</b>	Drilling with dwell canned cycle.
<b>g83</b>	Chip-breaking drilling canned cycle.
<b>g84</b>	Right hand tapping canned cycle.
<b>g85</b>	Boring, no dwell, feed out canned cycle.
<b>g86</b>	Boring, spindle stop, rapid out canned.
<b>g87</b>	Back boring canned cycle.
<b>g88</b>	Boring, spindle stop, manual out canned.
<b>g89</b>	Boring, dwell, feed out canned cycle.
<b>g90</b>	Absolute distance mode.
<b>g91</b>	Incremental distance mode.
<b>g92</b>	Offset coordinate systems (RESET your home position values).
<b>g92.2</b>	Cancel offset coordinate systems.
<b>g93</b>	Inverse time feed mode.
<b>g94</b>	Feed per minute mode.
<b>g98</b>	Initial level return in canned cycles.

Table 2

<b>M Code List</b>	
<b>m00</b>	Optional program stop.
<b>m01</b>	Program stop.
<b>m02</b>	Program end.
<b>m03</b>	Turn spindle clockwise, e.g., s1000 m3 for 1000 rpm.
<b>m04</b>	Turn spindle counterclockwise.
<b>m05</b>	Turn spindle off.
<b>m06</b>	Tool change, e.g., t1 m6 for tool 1.
<b>m07</b>	Mist coolant on.
<b>m08</b>	Flood coolant on.
<b>m09</b>	All coolant off.
<b>m48</b>	Disable feed override.
<b>m49</b>	Disable spindle speed override.

**Table 3**

<b>G Code and M Code Modal Groups</b>
Group 1 = {g0, g1, g2, g3, g80, g81, g82, g83, g84, g85, g86, g87, g88, g89} —motion
Group 2 = {g17, g18, g19} — plane selection
Group 3 = {g90, g91} - distance mode
Group 5 = {g93, g94} - spindle speed mode (feed rate mode)
Group 6 = {g20, g21} - units
Group 7 = {g40, g41, g42} - cutter diameter compensation
Group 8 = {g43, g49} - tool length offset
Group 10 = {g98, g99} - return mode in canned cycles
Group 12 = {g54, g55, g56, g57, g58, g59, g59.1, g59.2, g59.3} coordinate system selection
Group 2 = {m26, m27} - axis clamping
Group 4 = {m0, m1, m2, m30, m60} - stopping
Group 6 = {m6} - tool change
Group 7 = {m3, m4, m5} - spindle turning
Group 8 = {m7, m8, m9} - coolant
Group 9 = {m48, m49} - feed and speed override bypass

**Table 4**



**SHERLINE**  
**PRODUCTS**  
INCORPORATED 1974

Appendix B:  
Sherline Accessories Shop Guide  
For the Mechatronics Lab





(A.) CNC Mill Z-axis Backlash Lock, P/N 4017Z



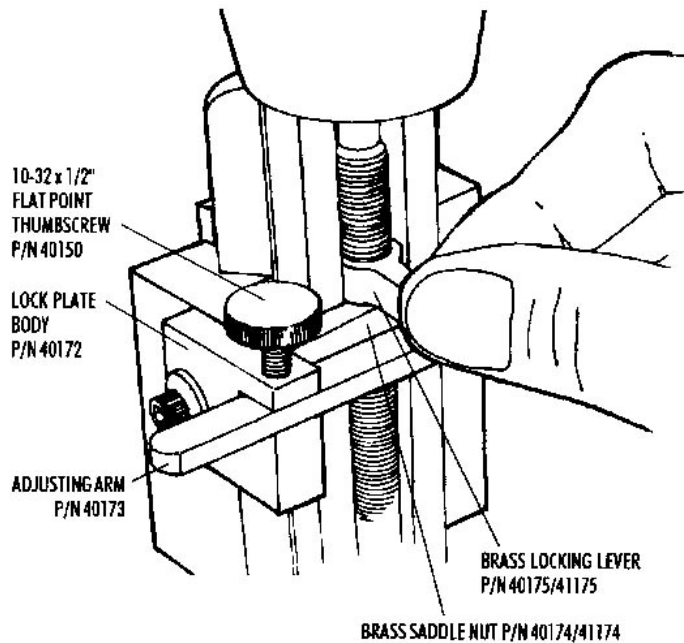
***Adjust brass locking lever to desired amount of backlash and then lock positioning arm in place with knurled thumbscrew. Side plate attaches using existing (Click on photo for larger image.)***

[The purpose of the Z-axis backlash lock](#)

CNC machining operations require precise Z-axis movement to producing a good part. The normal backlash of .003" to .005" can be unacceptable in these situations and a way was needed to reduce backlash to the .001" to .002" range. The problem would arise when the Z-axis gibs were set so tight to eliminate vibration that the headstock had to be physically pushed down. This would leave backlash between the cutter and the part. With the normal vibration of making a cut and the tendency of an end mill to pull itself down into the part, the headstock would slowly lower the amount of this backlash. *Sherline* owners found ways to work around this problem when working with manual machines, but it soon become apparent that this was a problem that was unacceptable for CNC machining. I'm happy to state that I solved this problem without adding excessive cost to *Sherline* mills and at the same time apologize for not finding a solution sooner. Because I designed this lever to lock the leadscrew a couple of years ago, I never thought of it as an option that could be used to control backlash; however, early one morning I finally realized how to use this lock could be used in this way, and we had a prototype working the next day and were into production within a week. I'm also pleased to report that the anti-backlash system can be added to every *Sherline* mill or vertical milling column ever built!

By locking this lever against the Z-axis saddle nut, it keeps the leadscrew from turning once the headstock is positioned at the proper height for the operation. This new lever positioning system will allow you to position the locking lever in a partially locked position, removing as much

backlash as you desire, and then it can be locked in that position. As wear occurs, the position of the locking lever can be adjusted.



*Figure 1—Components of the lever locking system as they are installed on new Sherline CNC mills.*

#### Using the Z-axis lock on a CNC mill

Move the brass locking lever to provide the desired amount of backlash. Tighten the thumbscrew to hold the plastic arm in position. Do not over tighten the thumbscrew. There is not a lot of force trying to move the arm. The arm serves only to hold the locking lever in position once you have adjusted it.

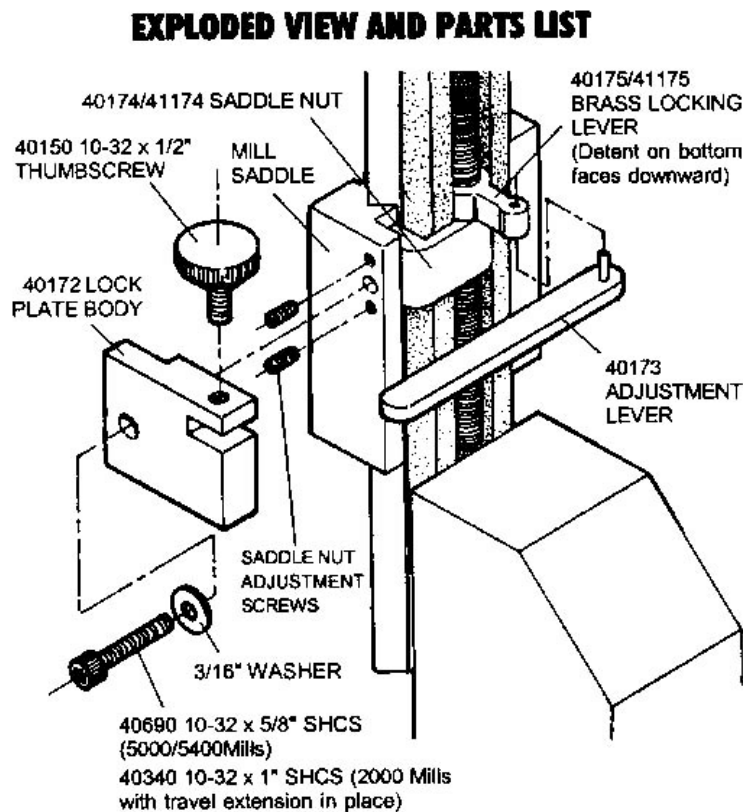
The brass locking lever is adjusted by hand as shown in Figure 1, and then the positioning arm is locked in place. Don't adjust the locking lever by moving the plastic arm directly. You will have much better feel for the amount of pressure needed by adjusting the locking lever itself.

#### Adjusting the saddle nut alignment

When you reinstalled the new saddle nut in place of the old one, the two adjusting set screws were left in their previous adjustment. If binding occurs in the new installation, it will be necessary to readjust the new saddle nut on the leadscrew using the two set screws on either side of the attachment screw. To do this you will first need to remove the locking plate to get access to these screws. Then, to adjust the saddle it should first be positioned at the end of its travel as close to the hand wheel as possible. Remove the lock plate and loosely install the original shorter screw that attached the saddle to the old saddle nut. Bring each set screw into light contact with the saddle nut and retighten the screw. If binding occurs, readjust the two set screws until the

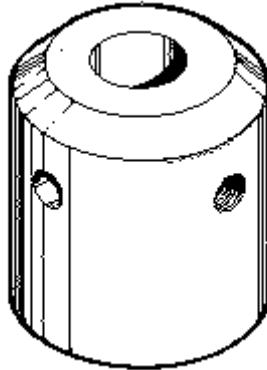
leadscrew moves freely. Then remove the shorter screw and reinstall the locking plate and new, longer screw and tighten.

What we are attempting to accomplish is to have the saddle nut ride on the leadscrew with the minimum amount of drag. You can check the drag by turning the leadscrew hand wheel. If you feel excess drag, tighten or loosen a single set screw while moving the saddle with the hand wheel until the hand wheel turns freely, but keep the saddle close to the hand wheel. If you adjust the saddle nut while it is in the center of the leadscrew, it may be slightly off center but will feel free until the saddle gets close to either end of its travel. Here, the leadscrew is supported and cannot deflect so it will bind the most. If you can't eliminate the binding, tap the saddle nut with a plastic hammer on the leadscrew side while the saddle nut is tightly attached to the saddle and readjust. Don't use the machine with a loose attachment screw, as this will cause excessive wear and backlash.



*Figure 2—Exploded view shows saddle nut and locking lever installed on the leadscrew with the other parts of the Z-axis lock pulled out. Adjustment to the two saddle nut adjustment screws must be made before installing the plate body. Use the original saddle nut screw to make the adjustment. Then remove it, install the plate and use the new longer*

**SHERLINE**  
**PRODUCTS**  
INCORPORATED 1974



**(B.) SHERLINE End Mill Holders**

P/N 3079 (3/8" End Mill Holder)

P/N 3075 (5/16" End Mill Holder)

P/N 3076 (6.0 mm End Mill Holder)

P/N 3077 (8.0 mm End Mill Holder)

P/N 3078 (10.0 mm End Mill Holder)

P/N 6079 (1/4" End Mill Holder)

P/N 6080 (3/16" End Mill Holder)

P/N 6081 (1/8" End Mill Holder)

---

**End Mill Holders for the Sherline Mill**

Most of the work accomplished on the *Sherline* milling machine can be accomplished with one of the miniature series cutters. These cutters have standard shank diameters of 3/16" (4.8mm) or 1/4" (6.4mm) and are normally held using *Sherline's* Milling Collets, P/N 3060. However, there may be times when you will want to use a standard end mill with a 3/8" (9.5mm) diameter shank. These cutters are more readily available than the miniature series of cutters and are actually less expensive in many cases. Also, special purpose cutters such as those designed for cutting key slots, dovetails and corner radii usually have 3/8" diameter shanks. The *Sherline* 3/8" end mill holder will increase the versatility of your milling machine by enabling you to hold these popular cutters. (CAUTION! Always consider the power and size limitations of your equipment when using larger size cutters. You may want to reduce cutter speed.)



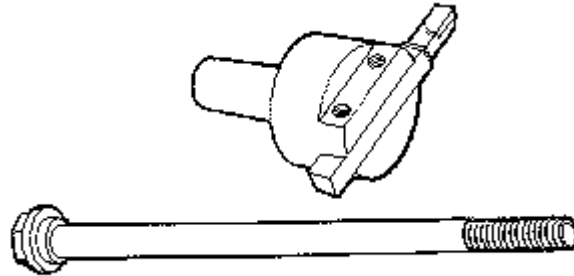
*FIGURE1--The flat area on a commercial double-ended end mill is for the set screw on the end mill holder to tighten against.*

In addition to the 3/8" holder, this same type of end mill holder is now also available to hold 5/16" (P/N 3075), 1/4" (P/N 6079), 3/16" (P/N 6080) and 1/8" (P/N 6081) end mills. The advantage of this type of holder is that it allows you to use smaller size double ended mills or tools with longer shanks which could not be held in a milling collet. We actually decided to produce this type of holder in the smaller sizes after many requests from *Sherline* users who simply prefer its ease of use even when using regular miniature end mills.

For those who use metric size cutting tools, the 10.0 mm (P/N 3078) is the most common size, but we also offer 6.0 mm (P/N 3076) and 8.0 mm (P/N 3077) sizes.

One of our *Sherline* machinists also pointed out that these holders make an excellent "quick-change" tool system for milling. Keep your favorite cutters chucked up in their own holders and simply swap holders to change tools. Since they just screw onto the spindle thread, changeover is a quick operation.

**SHERLINE**  
**PRODUCTS**  
INCORPORATED 1974



**(C.) SHERLINE Flycutter**  
P/N 3052

---

**Use of a Flycutter**

The Flycutter is held in the spindle with a drawbolt that pulls it up into the Morse #1 taper. This is a "sticking" taper and it has to be removed from the spindle by backing out the drawbolt a few turns (do not disengage) and giving the bolt a few light taps with a hammer.

A Flycutter is a great way to machine flat surfaces. It can be easily sharpened and is probably the most economical way to remove material on a mill. The cutter is basically a left handed lathe tool. We supply it with a carbide tip cutter, but there is no reason a piece of 1/4" square high speed steel wouldn't work.

As with all machining operations, it is imperative the work is securely held to the work table. A flycutter on the *SHERLINE* can cut a path 2" (50mm) wide by .010" (.25mm) deep in aluminum without even trying. Flycutters exert lower machining stresses on the machine than you may expect. This is because the cutter "peels" the material off with very little crushing action. If possible the cutter should swing a diameter larger than the part width. The cutter will usually take a second cut with the back side of the cutter even when the spindle is perfectly square with the table.

---

**CAUTION!**

**Chips thrown off by the flycutter are HOT.  
Long sleeve shirts are advisable and eye protection is a must!**

---

If you're machining aluminum, run the spindle at 1/2 speed, with steel use 1/4 speed, and use a feed rate that creates curly chips about .002" (.05mm) thick. You really should have some understanding of cutting speeds if you use high speed steel cutters on steels. It is very easy to exceed the cutting speed of high speed steel with a large cutter diameter. An example would be a H.S. cutter swinging a 2" (50mm) circle shouldn't exceed 200 RPM when cutting steel with a cutting speed of 100 surface Ft/Min.

$$4 \times \text{CUTTING SPEED (Ft./Min.)} \div \text{CUTTER DIAMETER (In.)}$$

Or

$$(4 \times 100) \div 2 = 200 \text{ RPM}$$

*Note: The factor 4 used in this equation has been rounded off to allow mental calculations, the actual number should be 3.8*

In metric a close approximation is:

$$300 \times \text{CUTTING SPEED (M/Min.)} \div \text{CUTTER DIAMETER (mm)}$$

*Note: The factor 300 has been rounded off to simplify calculations, the actual number is 318.*

**Flycutter Parts List**

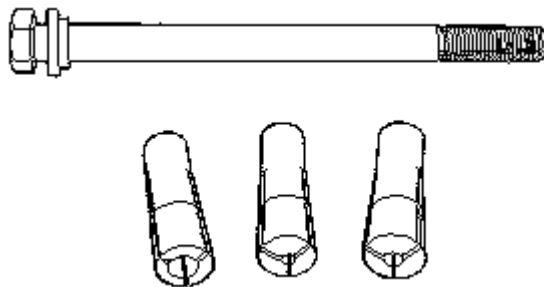
NO. REQ.	PART NO.	DESCRIPTION
1	1193	Carbide Tool, Left
1	4005	Hand wheel, Inch (P/N 4105 Metric)
1	3049	Flycutter Body)
1	3088	1/4-20 x 5-1/8" Drawbolt
2	3108	10-32 x 3/8" Set Screw

**ACCESSORIES AVAILABLE**

PART NO.	DESCRIPTION	PRICE
1196	High Speed Steel Tool, Left	\$5.00
3005	High Speed Steel Tool, Blank	\$3.00
3005B	5 High Speed Steel Tools, Blank	\$10.00



**SHERLINE**  
**PRODUCTS**  
INCORPORATED 1974



**(D.) SHERLINE Milling Collet Set**  
P/N 3060 (Metric P/N 3090)

---

**Milling Collets**

SHERLINE Milling Collets are designed to be used with the Morse #1 internal taper that is standard on both the Lathe and Vertical Mill. Because of the shallow angle of the Morse #1 taper when the drawbolt is tightened, greater clamping force can be applied when compared to the clamping pressure of WW collets; therefore, we recommend the use of these milling collets for holding miniature size end mills (3/16" or 1/4" shank), #1 and smaller center drills or Dremel bits (1/8" shank) and other small cutters. The set consists of a 1/8", 3/16" and 1/4" mill collet, drawbolt and thrust washer. Other size collets, including metric mill collets, are available through *SHERLINE*. The metric sets consists of 3mm, 4mm and 6mm collets.

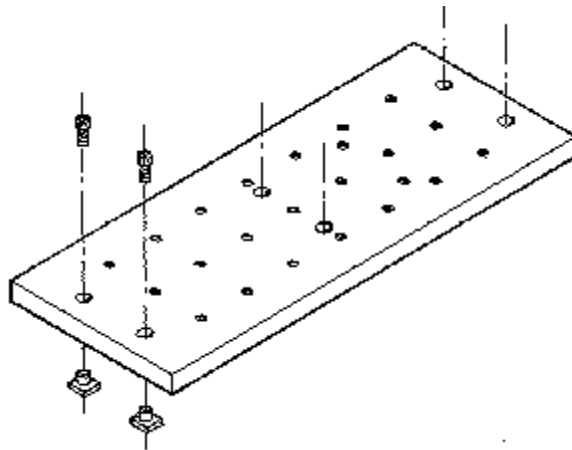
Cutting tools held by these collets **must** have a shank size within .001" of the manufactured size. Because of the shallow angle, these collets will not loosen from the spindle without backing the drawbolt off a couple of revolutions and tapping the drawbolt to free the collet.

End mills must run true in order to cut properly. Small end mills or miniature size end mills are very expensive and can be easily damaged if milling is attempted using a drill chuck to hold the end mill. Milling Collets are a **must** for proper operation of your mill! For holding other size end mills, see also P/N 3079, 3/8" End Mill Holder.

Milling Collets are available in the following sizes:

- P/N 3087 3/32" Mill Collet
- P/N 3089 5/32" Mill Collet
- P/N 3091 7/32" Mill Collet
- P/N 3092 3.0mm Mill Collet\*
- P/N 3093 4.0mm Mill Collet\*
- P/N 3094 6.0mm Mill Collet\*
- P/N 3095 1/8" Mill Collet\*\*
- P/N 3096 3/16" Mill Collet\*\*
- P/N 3097 1/4" Mill Collet\*\*
- \*Included with set P/N 3090
- \*\*Included with set P/N 3060

**SHERLINE**  
**PRODUCTS**  
INCORPORATED 1974



**(E.) SHERLINE Mill Tooling Plate**  
P/N 3560

---

The Mill Tooling Plate went into production after a request from a customer who had damaged his mill table. It was an intelligent request because we use Tooling Plates on many of our large shop machines. They not only protect the machine's table from damage, but they provide an inexpensive, modifiable surface for clamping work that is as flat as the machine's original table. We felt it was time the *SHERLINE* miniature machine tool line should be able to benefit from this same shop practice.

**Use of the Mill Tooling Plate**

Like a Lathe Faceplate, the Mill Tooling Plate should be looked at as somewhat disposable. It was manufactured more for making it easier to clamp down hard-to-hold parts than for protecting the table. It has a hole pattern predrilled to make it easy to clamp down the *SHERLINE Mill Vise*. Two additional holes were also added to accommodate the *SHERLINE Rotary Table*, but you should feel free to drill whatever additional holes are needed to meet your particular clamping and setup needs. We use 10-32 holes for the *SHERLINE* clamp screws. Unless you have a particular need for another size, we suggest that you stay with that size for additional holes so the same size screws can be used in all holes.

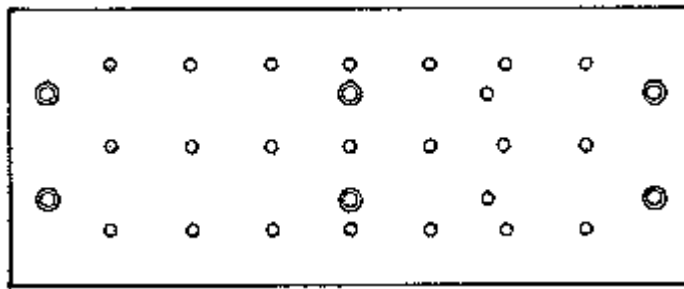
The material used for the plate is cast aluminum which is milled to 1/2" thickness. Though not quite as strong as extruded grades, it is very stable and free from internal stresses so it will not warp when you drill additional holes. The surface is not anodized as it is intended to be drilled and machined to suit your unique needs. The size of the plate is approximately 4" x 10".

### Clamping to the Tooling Plate

Use all six mounting screws and Tee Nuts provided to mount the Tooling Plate to your Mill Table. Do not over-tighten Tee Nuts or you can damage your table slots. The heads are countersunk into the Tooling Plate to leave your working surface free from obstructions.

### Clamping Parts to the Tooling Plate

It should always be your goal to try and hold a part as firmly as possible for milling. If a part moves during the cutting process it will probably be ruined. The heavier the cut you wish to take, the more important this becomes. The best way to hold a part more securely is by adding more clamping points. Do not over-tighten the points you already have. For example, on the mill vise use four clamping points rather than two.

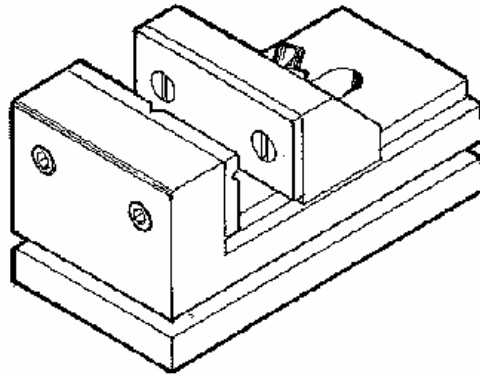


*Mill Tooling Plate hole pattern. Six countersunk holes are for mounting the plate to the Mill table with socket head screws and Tee Nuts provided. Other holes are for mounting Mill Vise or Rotary Table. Drill additional holes as needed for your particular jobs.*

### Mill Tooling Plate Parts List

NO. REQ.	PART NO.	DESCRIPTION
1	3561	Tooling Plate (4" x 10" x 1/2")
6	3562	10-32 x 7/16" Skt Hd Cap Screws
6	3056	Tee Nuts (Replacement set of 4)

**SHERLINE**  
**PRODUCTS**  
INCORPORATED 1974



**(F.) SHERLINE Mill Vise**  
P/N 3551

---

**Purpose of the Mill Vise**

The most convenient way to hold small parts for milling is the Milling Vise. Milling vises are different from other machine vises in that they are designed to hold the movable jaw down while clamping, eliminating any chance for the jaw to lift. A cheap drill press vise is almost useless on a milling machine because you can't accurately align the machine to it.

The SHERLINE Milling Vise is sized just right to be the perfect working companion to the SHERLINE Milling Machine. The jaws are 1.50" (38.07mm) wide by 1.00" (25.38mm) deep and open a full 2.00" (50.76mm). The fixed jaw has both a horizontal and vertical "V" groove to facilitate holding round bar stock. The vise comes complete with two clamps for use in attaching it to the milling machine. These clamps are the same as the 4-Jaw Hold Down Set (P/N 3058).

*NOTE: To keep the surface of your table from being marred, we recommend you take a file and slightly soften or deburr all the edges of the angle clamps that contact the table.*

**Instructions for Use**

The advantages of this vise are obvious when movement of the jaw is studied. (See Figure 1 and look at the bottom of your vise.) The tightening force (F1) produces not only a force against the part (F2), but also a force pulling the jaw downward (F3). Therefore, angle "A" must exceed 45° in order to make force F3 greater than F2. This keeps the movable jaw from "tipping" back. Also note that extreme clamping angles beyond 60° start to apply much downward pressure but not

much horizontal force is directed to holding the part. Moving the pull-down barrel to the proper slot keeps the adjustment within the most effective clamping range.

To clamp a part, place the jaw in approximate position and start tightening the adjustment screw at an angle of 45° or greater. (The back face of the moveable jaw is machined at a 45° angle for reference.) If the angle of the adjustment screw gets up to 60° or greater and you still haven't drawn down on the part, loosen up the screw a little and move the pull-down barrel to the next slot and retighten.

---

***CAUTION! Extreme vertical adjustment angles can allow the 10-32 x 1-3/4" adjustment screw to be driven into the surface of the table, damaging both screw and table. Therefore, the vice comes assembled with a 1-5/8" screw that is usable for most settings. To enhance adjustment to the longest ranges, change to the 1-3/4" screw provided.***

---

Figure 2 shows the proper way to hold a part in the vise. If the part cannot be centered, use a spacer to help keep the jaws parallel. This vise has been designed to accurately hold objects being machined. It is not recommended for use as a bench vise or for clamping parts in such a way and with such force as to adversely effect its accuracy.

### Exploded View and Figures 1 and 2

FIGURE 1-below left, is a cross-section which shows the pull-down bar located in the center detent of the vise body.

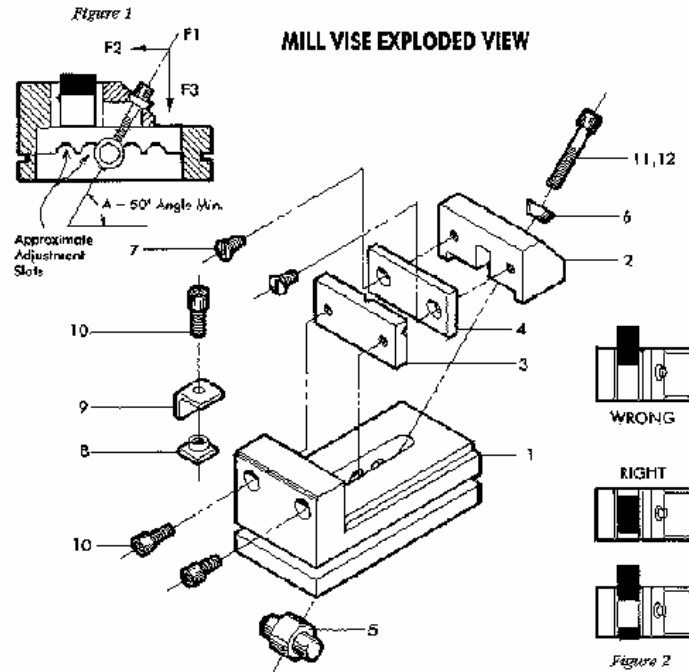
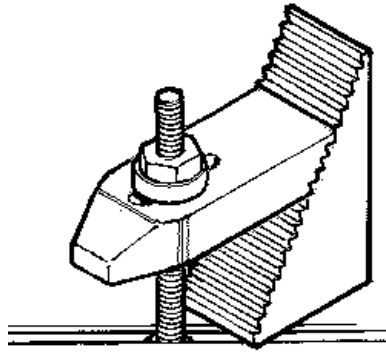


FIGURE 2- above right, shows the right and wrong way to hold a work piece in the vise.

### Mill Vise Parts List

REF. NO.	PART NO.	DESCRIPTION
1	35100	Vise Body
2	35020	Movable Jaw
3	35030	Fixed Jaw Insert
4	35040	Movable Jaw Insert
5	35120	Pull-Down Member
6	35060	Convex Washer
7	35070	Flat Head Screw, 6-32 x 3/8 (2)
8	30561	T-Nut, 10-32 (2)
9	35580	Hold Down Clamp (2)
10	40330	Skt Hd Cap Screw, 10-32 x 5/8" (4)
11	35130	Skt Hd Cap Screw, 10-32 x 1-5/8"
12	40700	Skt Hd Cap Screw, 10-32 x 1-3/4"



**(G.) Sherline Step Block Hold-down Set**  
P/N 3013

*See the end of this section for free plans to make a wooden block to hold and organize the pieces of your step block set.*

---

### **Sherline's step block hold-down set**

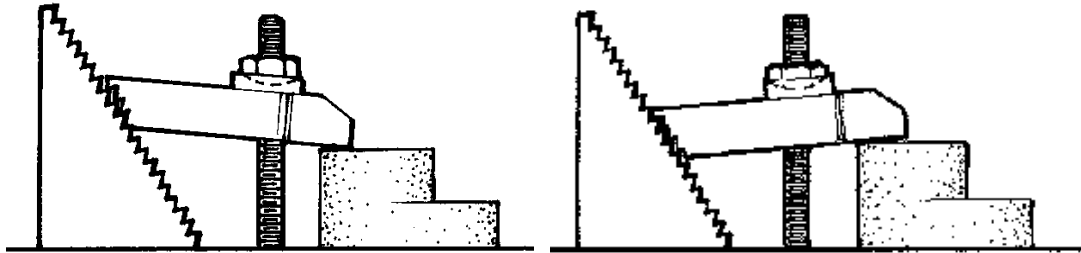
Look in any machine shop supply catalog and you will find hold-down sets that consist of step blocks with corresponding clamps that use T-nuts and studs. This is a very practical way to solve one of the more difficult problems that come up in the machine trades; that is, clamping a part to the table. I am sorry I didn't get around to manufacturing these sooner, but there always seemed to be a more pressing problem to solve until now.

### **Use of the step block hold-down set**

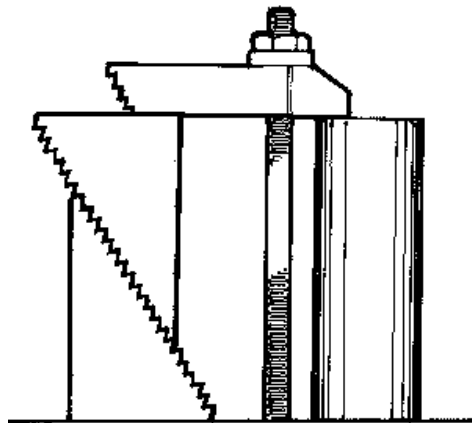
Not much is required in the way of instructions in order to use these clamps. A few illustrations are included to give you some tips. The biggest problem you can create for yourself is over-tightening the nut and damaging the T-slots on your mill table. You have to realize that you are stronger than your machine when you have both the lever (wrench) and the screw working to your advantage (or disadvantage).

In addition to the threaded studs we provide, 10-32 threaded stock can be purchased from most industrial suppliers and cut to length if you need a longer piece. We have also provided an extra, non-anodized section of step block extrusion for you to cut up into smaller step blocks and machine flat should you need them. (See Figure 3.)

A nice feature of this set that you won't find even in more expensive, full-size sets is the radiused nut and washer. They allow the nut to pull the washer down flat on the clamp even when it is slightly tilted. I appreciate this kind of detail and felt our customers would too.



*FIGURE 1-On the left is shown the **correct** way to hold a part. The tip of the clamp should pull down on the part. On the right is shown the **incorrect** way to hold a part, with the clamp angling upward and pushing down on the edge of the part. (The angles are exaggerated for illustration purposes. The many steps on the block will allow you to get your actual setup much closer to level.)*



*FIGURE 2-The blocks can also be stacked as shown to hold tall parts. (For parts taller than 3", a longer piece of 10-32 threaded stock will be required.)*



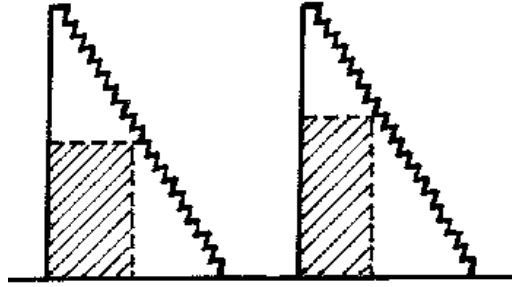
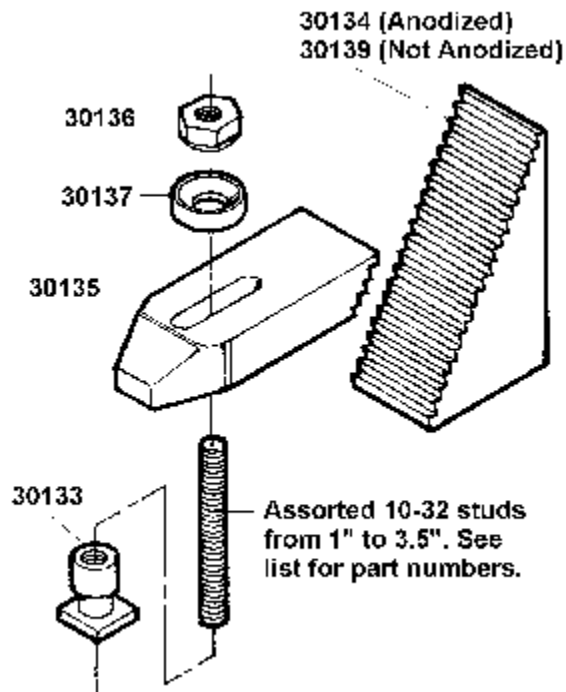


FIGURE 3-The unanodized block can be cut with a bandsaw or a hacksaw and milled into smaller blocks. On the left is shown a pattern for two equal 10-step blocks. On the right, the block is cut into a smaller 8-step and a larger 12-step block. The cross-hatched area of material is removed. The full block actually has 21 steps, one of which is lost to the horizontal cut. After sawing, mill the cut surfaces flat.

### EXPLODED VIEW PARTS DIAGRAM



### Step Block Hold-down Set Parts List

NO. REQ.	PART NO.	DESCRIPTION
2	30134	Step Block (anodized)
2	30133	Extended 10-32 T-nut
2	30135	Step Clamp
2	30136	Step Block Nut

2	30137	Step Block Nut Washer
2	30138	10-32 x 1" Stud
1	30139	Step Block (unanodized)
2	30141	10-32 x 1.5" Stud
2	30142	10-32 x 2" Stud
2	30143	10-32 x 2.5" Stud
2	30144	10-32 x 3" Stud
2	30145	10-32 x 3.5" Stud
2	30561	10-32 T-nut

***Make a wooden block to hold your step block set***

Full-size step block sets usually are held in a metal tray or box that keeps all the small parts together. You can make a block to hold the parts of your step block set as a simple afternoon project. Use a fine-grained hardwood such as maple, Honduras mahogany, birch or white oak. You could also use a block of aluminum if you have one available and want some milling practice. Cut the pockets and drill the holes using your milling machine. Hold the block with the step block set as you work on it. Finish the wood with Tung oil, varnish or polyurethane finish for a nice looking and practical addition to your shop.