
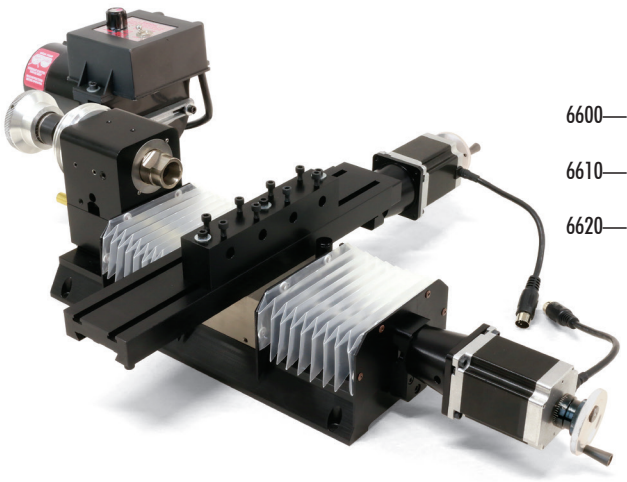

**PRODUCT DESCRIPTION**

6600—CNC Chucker w/3C headstock, ball screws, high-torque stepper motors & PC w/4-axis driver & software

6610—CNC Chucker w/3C headstock, ball screws & 2 high-torque stepper motors

6620—CNC Chucker w/3C headstock, ball screws, 2 high-torques stepper motors & 4-axis driver box (no PC)

SHERLINE INDUSTRIAL PRODUCTS DIVISION

# CNC Chucker Lathe

## P/N 6600, 6610, and 6620

### User Instructions for CNC Chucker Lathe with Ball Screws

**Notes:**

1. Do not get any chips on your ball leadscrew. If chips get into the ball nut, they will ruin the ball nut. On the headstock side of the table, the ball screw is protected with a brass tube. This makes it virtually impossible for chips to get onto the ball screw. On the backside of the table, the ball screw is protected by the accordion way cover. If this way cover gets filled with chips, it will become a "chip compactor" and two things will happen. One is that you will start to lose table travel in the Z-axis, because the chips are making the "closed cover thickness" of the cover thicker. Secondly, you will tear the cover and expose the ball screw to chips.
2. Do not blow the chips off your machine, use a small brush and brush them off. Blowing chips increases the chances of getting chips on your ball screw.

### Machine Accuracy

Your machine has been checked at the factory for alignment and backlash.

### Alignment

We indicate in the 3/8" hole on the gang tool post with a .0001" test indicator. We shim the headstock as needed to get the spindle centerline within .001" of the tool post.

### Backlash

We check the mechanical backlash on the X and Z-axis with a .00005" test indicator. The mechanical backlash on both axis is less than .001".

### Operating and Set-up Tips

1. Lubricate your machine with a light oil such as 3-in-1 oil. For the Z-axis, fill the oiler cup once a day (see Figure 1). This is a gravity feed system and the oil will flow down the sides of the base after a while. Mounting your machine inside a shallow basin (such as a cookie sheet) is recommended. For the X-axis, put a few drops of oil on your finger and wipe it onto the exposed dovetail surfaces underneath the table (see Figure 2). Then jog the table back and forth to spread the oil on both the X and Z-axis before running the machine.

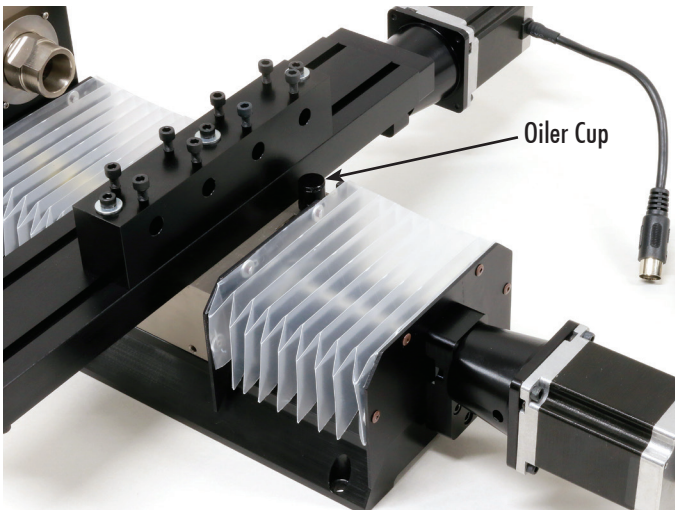


*FIGURE 1—Z-axis oiler cup*

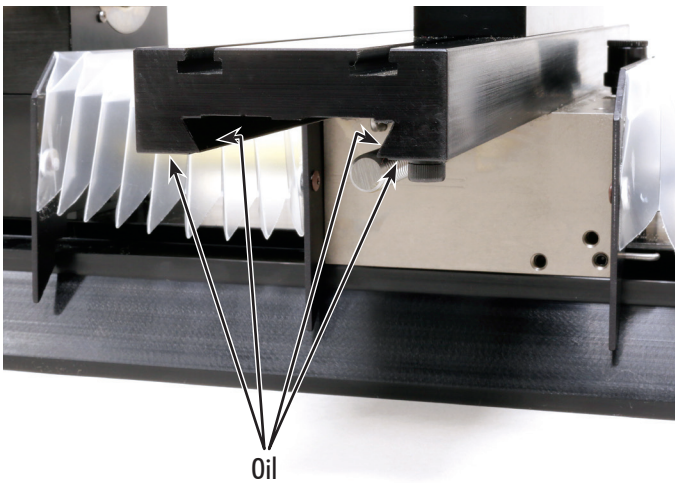


*FIGURE 2—The arrows indicate the surfaces underneath the table where oil should be applied.*

**SHERLINE PRODUCTS INC., Industrial Products Division** • 3235 Executive Ridge • Vista • CA 92081-8527 • Tel: (760) 727-5857 or (800) 541-0735 • Fax: (760) 727-7857
Internet: www.sherlineipd.com (Industrial Products Division) or www.sherline.com (machine tools and accessories)
12/22/21

2. Gang tool post: Your gang tool post will come from the factory mounted on the rear T-slot. This will give you more travel for the tools in the Z-axis. It will also offset the amount of usable space that is taken up by chucks and other holders that are mounted in the tool post. We center the gang tool post so you can add holders on either side of it if desired. Indicate the face of the gang tool post so it is perpendicular to the centerline of the spindle.
3. If you mount our rear cutoff tool post (P/N 3018) on the stepper motor side of the table (see Figure 3), you can use the cutoff holder as a programmable stop for your material. You can finish your program with a part-off cycle. Then you are already in position to start your next part.

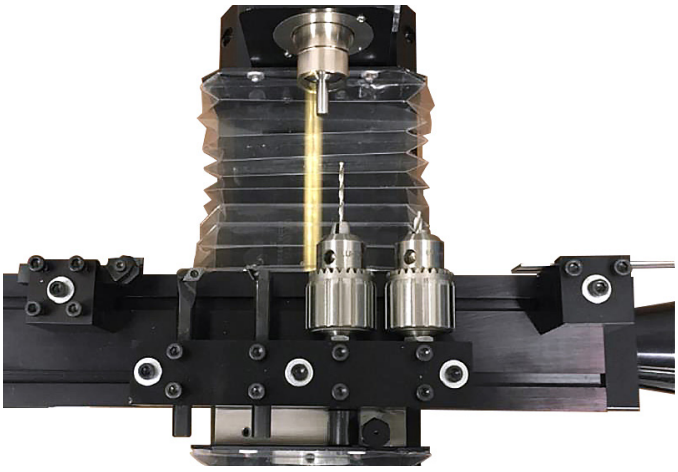

*FIGURE 3*

4. If you are going to be using the gang tooling tool post for most of your parts, it decrease your set up time if you indicate in each of the 3/8" holes. Move your machine until it hits the hard stops. Then move one full revolution (2.0mm) away from the hard stop. Zero out your X-axis. Place a test indicator in a collet in the headstock. Then use the jog mode and jog the first hole over to the spindle centerline. Using the Jog Increment function move the X and Z-axis until the hole is indicated in. Write down the X-axis position for the first hole. NOTE: Make sure to write "Rad" or "Dia" for your X-axis position depending on how you are going to program your parts. The control page shows both values (see Figure 4).

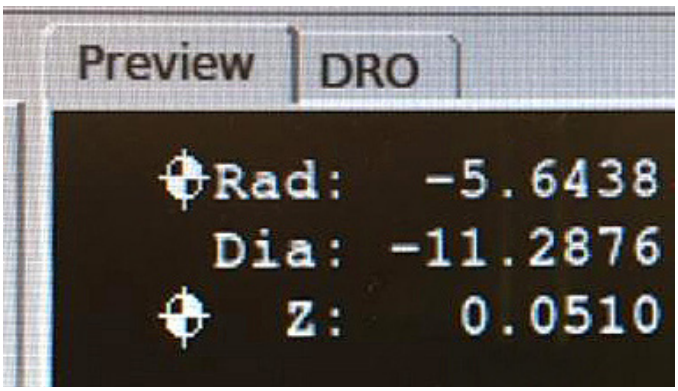


*FIGURE 4*

5. The more tools that you are using for your part, the longer and more complex the program becomes. In general, you cannot restart in the middle of the program. This means that if you have already dialed in the first four tools and you are working on the fifth tool, you can't just restart the program on the fifth tool. You have to run through the entire program again until it gets to the fifth tool. This is very tedious and time consuming. A better way is to write several short programs, one for each tool using the same home position and each tools individual tool offset values from the tooling page.

   Now run each individual short tool specific program for each tool. Once you have run all of the tool specific programs and dialed in each tool, you can do one of the following choices.

   a. Merge all of the small tool specific programs into one large continuous program.
   b. Turn each of the tool specific programs into "Sub Programs." Then make a "Main / Master" program that calls up each of the sub programs in sequence.

### The Lathe Uses a GUI Control Page and Tooling Page

1. Start up procedure:
   a. Click on the icon that represents your machine (Lathe Inch or Lathe Metric).
   b. Home out your machine.
2. If you already have a home position, turn your X and Z handwheels to the correct handwheel setting. Make sure the stepper motor power switch on the side of the PC is in the "OFF" position.
3. If you don't already have a home position, then you are just going to zero your machine at its current location.

   Zero your axis.
4. Click on the "Toggle Emergency Stop" icon (See Figure 5).

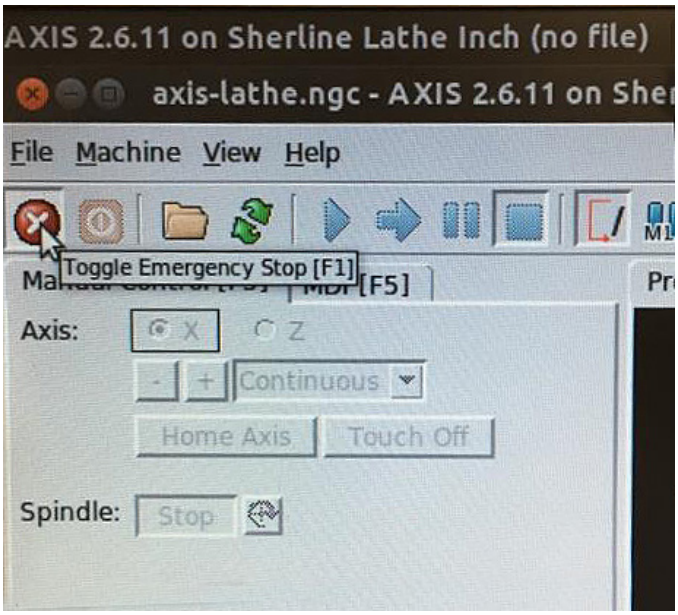


*FIGURE 5*

5. Click on the “Toggle Machine Power” icon.

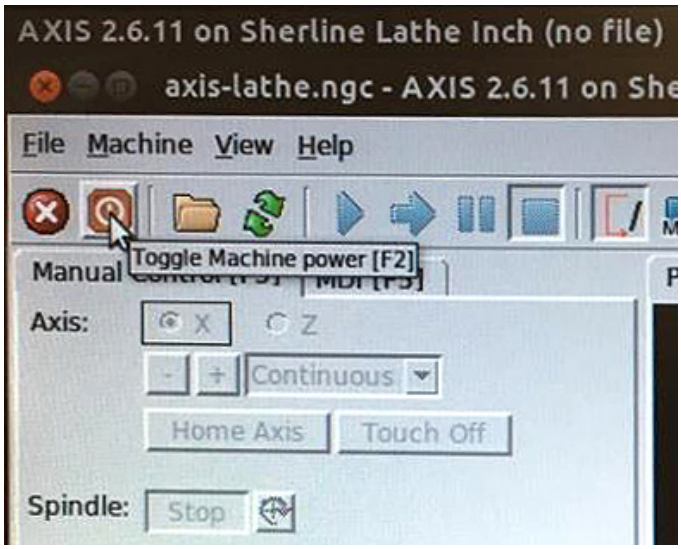


*FIGURE 6*

6. Click on the axis that you wish to Zero out. The dot will turn black when that axis is activated. Now click on the “Home Axis” icon.

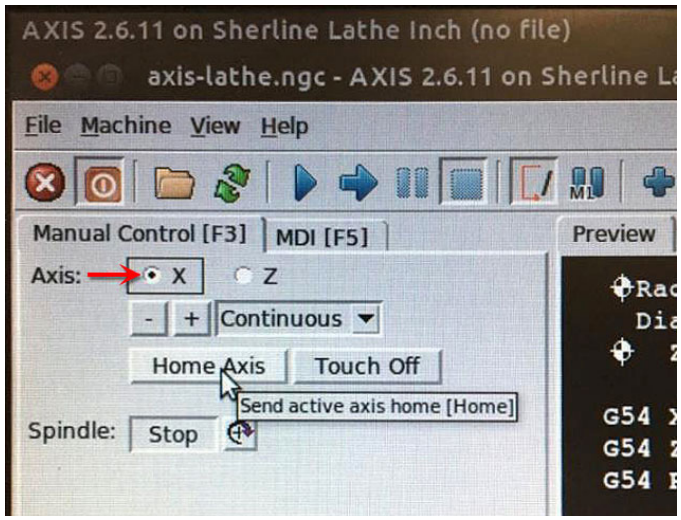


*FIGURE 7—Red arrow is pointing to the X-axis radial button.*

7. Choose the next axis, and then home it too.

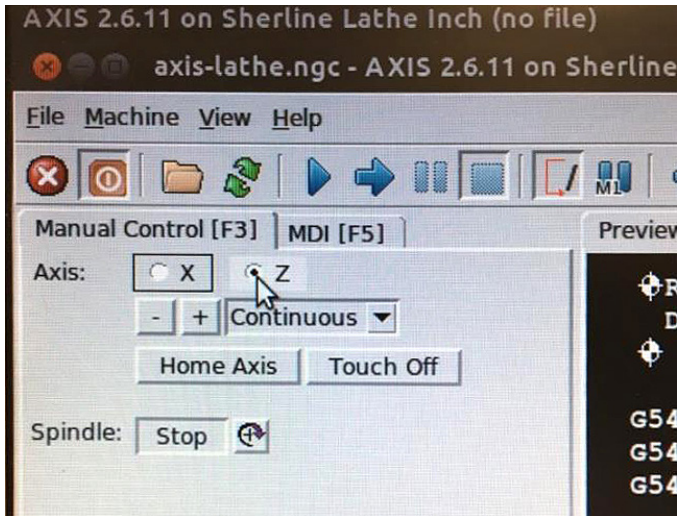


*FIGURE 8—Click the Z-axis radial button.*

5. Now your position page will show Zero for your radius, diameter, and Z-axis.

### To Access the Tooling Page

1. Click on “File” and then click on “Edit Tool Table.”

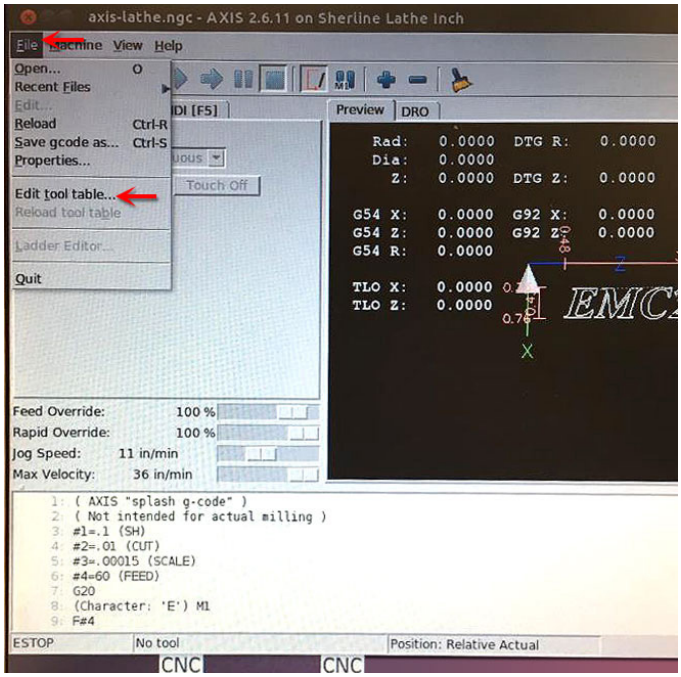


*FIGURE 9—Please note the red arrows indicating the “File” and “Edit tool table...” menus.*

2. This is what your tooling page will look like:
   a. The “Tool” and “Pocket” should be the same tool number.
   b. Your “X” is the distance from your X Home Position, to the “Centerline” of the part. You will acquire this number by making a cut on the OD of the stock, Writing down the X position shown on the position screen, Measure the diameter of the turned surface, and then adding the radius of that diameter to the X value that is shown on the position page. The sum of those two values is what you enter for your X value on the tool table. See example below.

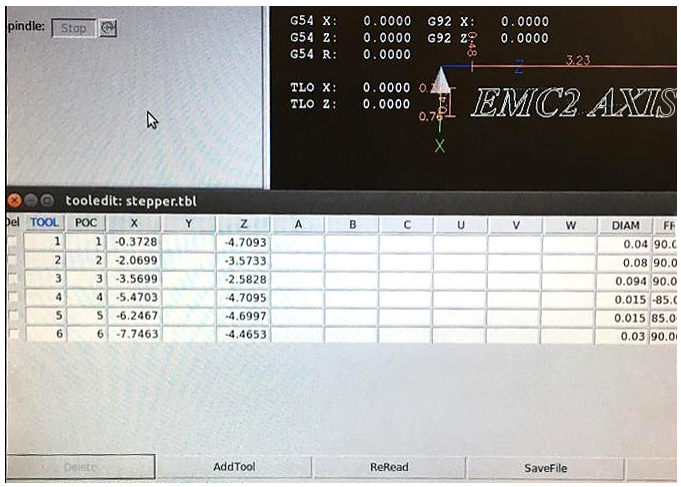


*FIGURE 10—Here is a link to more information on LinuxCNC Tool Tables (http://wiki.linuxcnc.org/cgi-bin/wiki.pl?ToolTable)*

For this example, we are using a ¼" dowel pin and just touching off on the side of it.

*FIGURE 11—Notice the carbide tip on the left just touching the side of the dowel*

Write down the X Radius value from the position screen (X-5.6438)

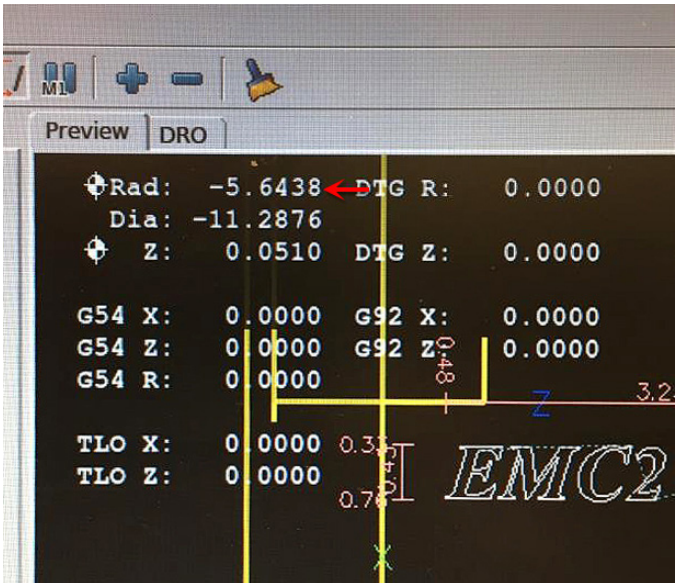


*FIGURE 12*

Measure the diameter of the part (.2502)


*FIGURE 13*

Add the radius of the part to the radius amount from the position page and enter that in the X value on the tool page (-5.6438 + -.1251 = -5.7689)

c. To get your Z-axis offset numbers:

Take the longest tool, touch it off on the front of the stock, then move in the Z+ direction by the amount of clearance that you want (Ex 1.00"). Now Zero out the Z-axis (this position will now be your Z Home Position.

The Z offset for the longest tool would then be Z-1.00".

Now touch off the rest of the tools and write down their Z position from the position page.

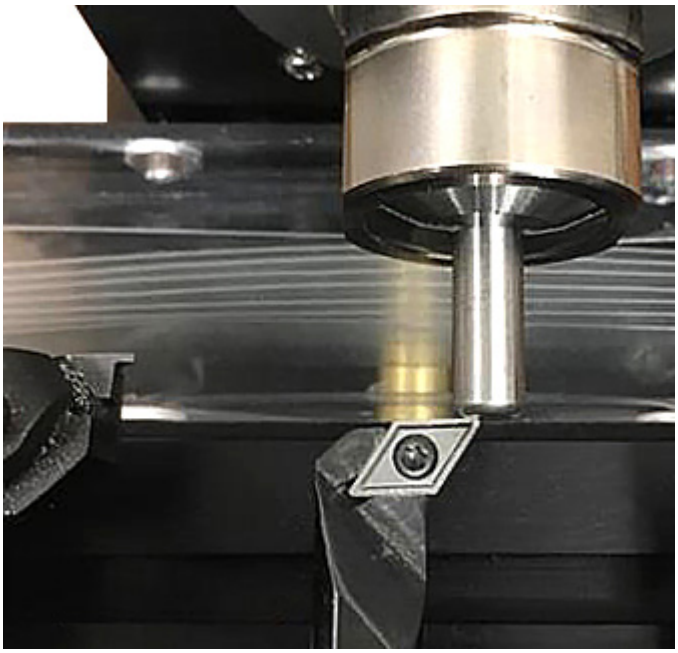

*FIGURE 14—Notice the carbide tip on the left just touching the front of the dowel*

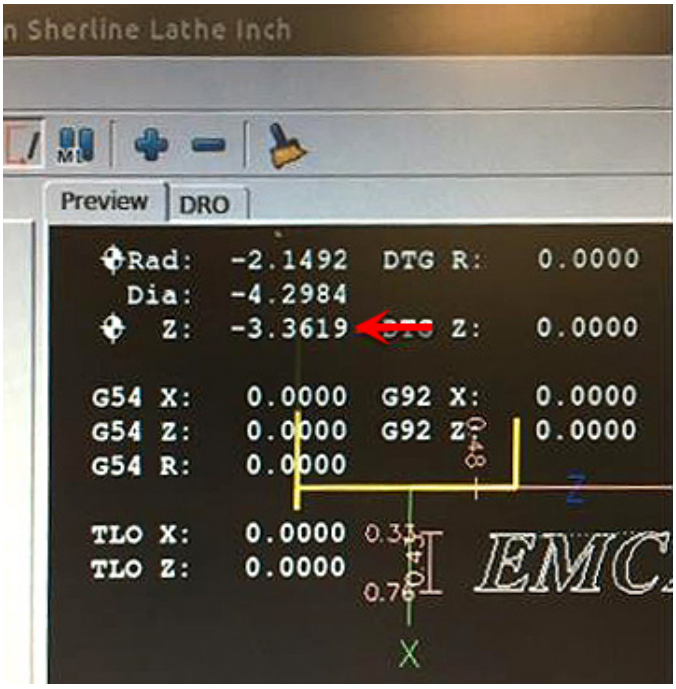


*FIGURE 15*

**Front and Back Angle for Your Insert or other Cutting Tool**

Below is a diagram showing how the front and back angles are defined. The insert shown is a 55-degree insert, which is 5 degree off square. The back angle is 30 degrees. The front angle is the sum of the back angle and the included angle of the insert (30 + 55 = 85).

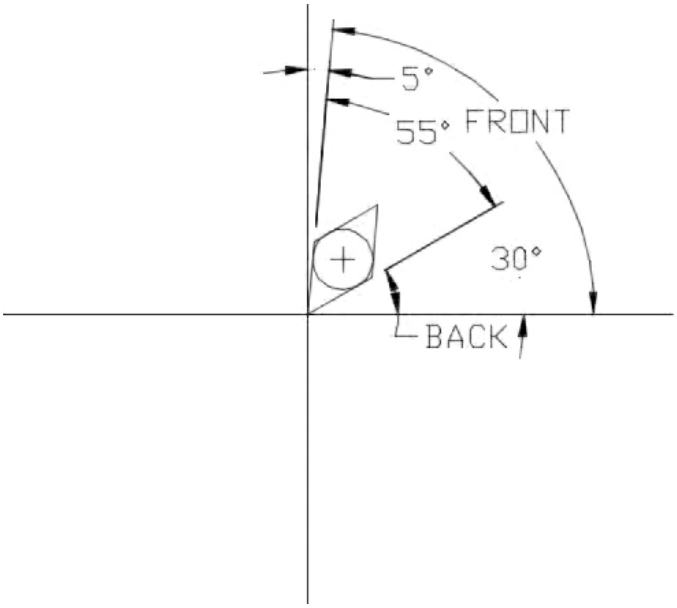


*FIGURE 16—The tool above is defined below on the tool page as tool #4 (fourth down from the top of the list in Figure 17).*

The tool #4 illustrated in Figure 16 above is a LH tool which is why the front and back numbers are negative in Figure 17 below.

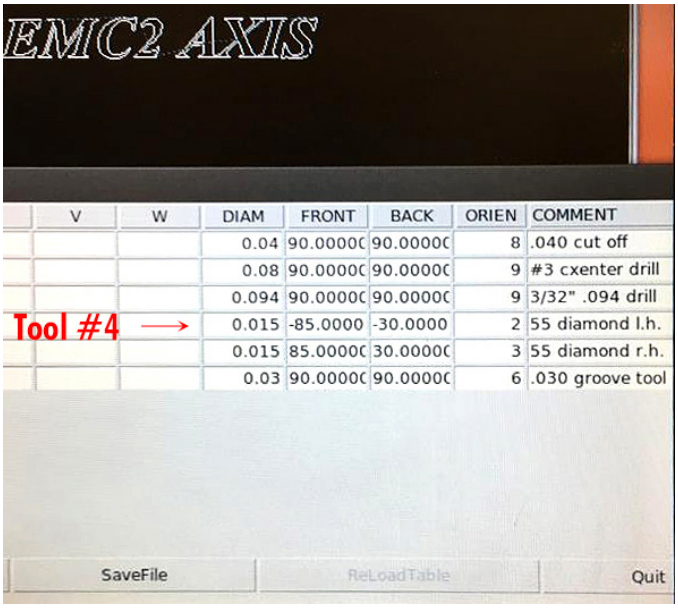


*FIGURE 17*

Any time that you make a change to the tool page, click on "Reload Table" and "Save File."

**Lathe Program**

1. This program was set up for a part on our CNC Chucker Lathe, which has gang tooling (see picture below).

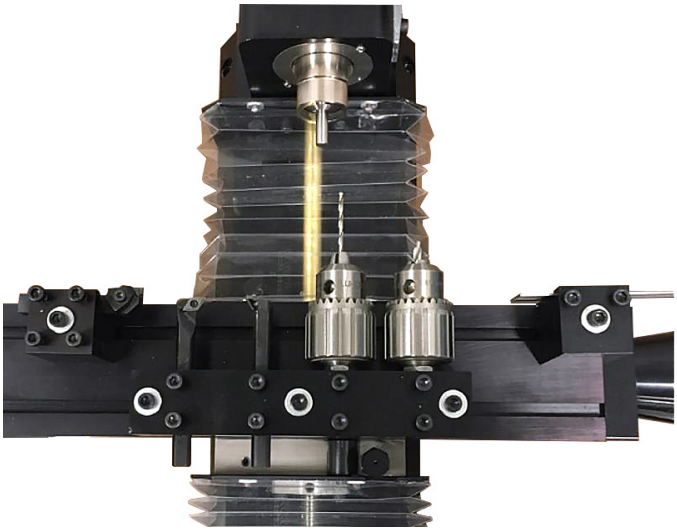

*FIGURE 19*

2. For this part we are using the following tools:
   a. .040 Cut Off Tool (which is also used as a stop to set the stock distance from the front of the collet)
   b. #3 Center Drill
   c. 3/32" .094" Jobber Drill
   d. L.H. 55 degree insert tool
   e. R.H. 55 degree insert tool
   f. .030 wide groove tool

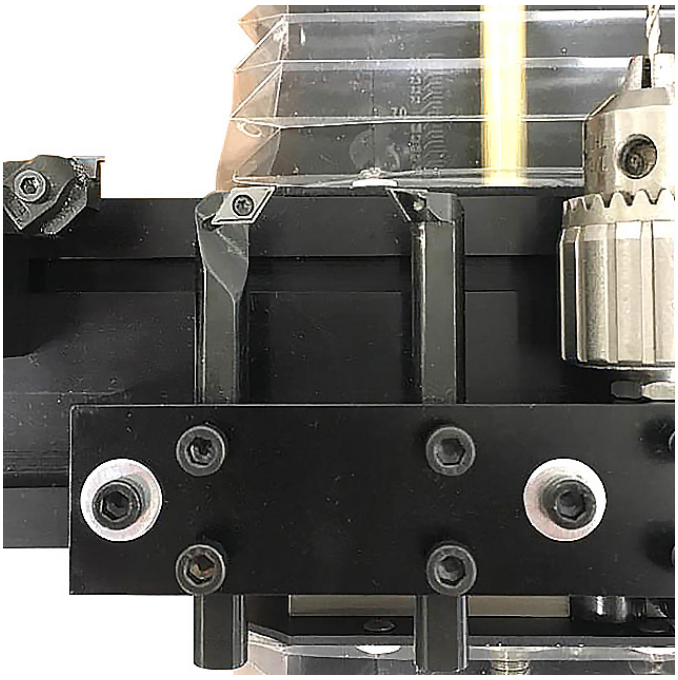

*FIGURE 19—Note: the LH and RH 55 degree tools are set up to cut on opposite sides of the part.*

3. Here is a print of the air fitting that we are making.

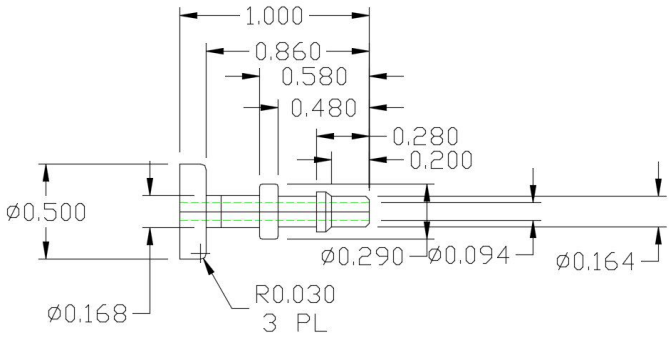


*FIGURE 20*

4. Here is a copy of the program.

```
%
(Machine Setup - 1 .040 wide Cutoff )
(TOOL #1     )
G80 G94 G7 G40 G20 G18 G90 G54
T1 M6 G43
G00 X-.6 Z.1
G00 Z-.820
G00 X-.208
G01 X-.118 F2.
G00 X-.208
G00 Z-.800
G01 X-.128 Z-.820 F1.0
```

G01 X.020 F3.0
G00 X0.0
G00 Z.100
G00 X0.0
M01
G01 Z0.0
M01
G00 Z1.
G49
G90G00G54 X0 Z0
(TOOL #2  2 CENTER DRILL )
N200 G00 G94 G7 G40 G20 G18 G90 G54
T2 M6 G43
G90 G00 Z.1
G00 X0
G01 Z-.100 F4.0
G90 G00 Z1.2
G49

(TOOL #3  0.0940 Dia.118.0000 Deg. 1.0000 CL )
N300 G00 G94 G7 G40 G20 G18 G90 G54
T3 M6 G43
G00 Z.1
G00 X0.0
G01 Z-.088 F2.0
G00 Z.1
G00 Z.060
G01 Z-.37
G00 Z.1
G00 Z-.350
G01 Z-.464
G00 Z.1
G00 Z-.440
G01 Z-.558
G00 Z.1
G00 Z-.530
G01 Z-.580
G00 Z.1
G00 Z-.560
G01 Z-.630
G00 Z.1

```
G00 Z-.610
G01 Z-.680
G00 Z.1
G00 Z-.660
G01 Z-.730
G00 Z.1
G00 Z-.710
G01 Z-.785
G00 Z.1
G00 Z-.76
G01 Z-.840
G00 Z.2
G49
(TOOL #4    55 DIAMNOD BACKSIDE  )
N400 G80 G94 G7 G40 G20 G18 G90 G54
T4 M6 G43
G00 X-.7
G00 Z0
G01 X0.0 F3.0

G00 X-.7 Z.052
G00 X-.57
G01 X-.47 Z.002 F6.
G01 Z-.8639
G02 X-.5095 Z-.884 I.0232 K-.0426
G02 X-.5197 Z-.9024 I.0533 K-.0247
G01 X-.52 Z-1.
G01 X-.62 Z-.95
G00 Z.052
G00 X-.52
G01 X-.42 Z.002
G01 Z-.8579
G02 X-.47 Z-.8639 I-.0005 K-.0532
G01 X-.57 Z-.8139
G00 Z.052
G00 X-.47
G01 X-.37 Z.002
G01 Z-.8578
G01 X-.42
G01 X-.52 Z-.8079
G00 Z.052
G00 X-.42
G01 X-.32 Z.002
```

G01 Z-.8578
G01 X-.37
G01 X-.47 Z-.8078
G00 Z.052
G00 X-.37
G01 X-.27 Z.002
G01 Z-.4807
G02 X-.3044 Z-.498 I.0124 K-.0296
G02 X-.3098 Z-.5083 I.0351 K-.0146
G01 X-.31 Z-.5744
G01 Z-.8578
G01 X-.32
G01 X-.42 Z-.8078
G00 Z.052
G00 X-.32
G01 X-.22 Z.002
G01 Z-.2335
G02 X-.23 Z-.2474 I.0165 K-.0138
G01 Z-.4778
G02 X-.27 Z-.4807 I-.0033 K-.0475
G01 X-.37 Z-.4307
G00 Z.052
G00 X-.27
G01 X-.17 Z.002
G01 Z-.0214
G02 X-.1823 Z-.0313 I.0136 K-.0153
G02 X-.184 Z-.0494 I.0705 K-.0122
G01 Z-.2097
G01 X-.22 Z-.2335
G01 X-.32 Z-.1835
G00 Z.052
G00 X-.22
G01 X-.12 Z.002
G01 Z.0006
G01 X-.1298 Z-.0017
G01 X-.1687 Z-.0207
G01 X-.17 Z-.0214
G01 X-.27 Z.0286
G00 Z.052
G00 X-.1867
G01 X-.0867 Z.002
G01 X-.1122 Z.0017
G01 X-.12 Z.0006

G01 X-.22 Z.0506
G00 X-.53
G00 Z-.2356
G01 X-.33
G01 X-.23 Z-.2856
G01 X-.2282 Z-.3002
G01 X-.1984 Z-.3275
G01 X-.184 Z-.3404
G01 Z-.3571
G01 Z-.4777
G01 X-.23 Z-.4778
G01 X-.33 Z-.4278
G00 X-.61
G00 Z-.5244
G01 X-.41
G01 X-.31 Z-.5744
G02 X-.3019 Z-.5959 I.0383 K-.004
G01 X-.26 Z-.6326
G01 Z-.8578
G01 X-.31
G01 X-.41 Z-.8078
G00 Z-.5826
G00 X-.36
G01 X-.26 Z-.6326
G01 X-.21 Z-.6759
G01 Z-.8577
G01 X-.26
G01 X-.36 Z-.8078
G00 Z-.6259
G00 X-.31
G01 X-.21 Z-.6759
G01 X-.194 Z-.6899
G03 X-.188 Z-.7022 I-.0114 K-.0093
G01 Z-.8577
G01 X-.21
G01 X-.31 Z-.8077
G00 X-.6
G00 Z.2
G49

(TOOL #5  55 DIAMOND FRONTSIDE )
N500 G80 G94 G7 G40 G20 G18 G90 G54
T5 M6 G43

G00 X.7 Z.0497
G00 X.1038
G01 X.0038 Z-.0003 F4.
G01 X.0942
G03 X.1131 Z-.0041 I-.0006 K-.0153
G01 X.1503 Z-.0223
G03 X.164 Z-.0395 I-.0127 K-.015
G01 Z-.2103
G01 X.2071 Z-.2391
G01 X.21 Z-.2464
G01 X.2092 Z-.2977
G01 X.2023 Z-.3063
G01 X.1659 Z-.3378
G01 X.164 Z-.3419
G01 Z-.4797
G01 X.2308 Z-.4799
G01 X.2424 Z-.481
G03 X.2664 Z-.4869 I-.0074 K-.03
G03 X.2892 Z-.5077 I-.0177 K-.0232
G03 X.2884 Z-.5865 I-1.3457 K-.0332
G03 X.2761 Z-.6006 I-.0428 K.0102
G01 X.1744 Z-.6887
G02 X.168 Z-.6965 I.0063 K-.0071
G01 Z-.8597
G01 X.4104 Z-.8599
G03 X.4786 Z-.8766 I-.0016 K-.0463
G03 X.4973 Z-.8955 I-.0308 K-.027
G03 X.5 Z-.9172 I-.0777 K-.0157
G01 Z-1.
G01 X.6 Z-.95
G00 X.8
G00 Z.5
G00 G49
(TOOL #6  OD GROOVE .030 WIDE )
N600 G80 G94 G7 G40 G20 G18 G90 G54
T6 M6 G43
G00 X.5
G00 Z-.3149
G00 X.274
G01 X.214 F4.
G00 X.254
G00 X.224
G01 X.184

G00 X.274
G00 Z-.3349
G01 X.214
G00 X.254
G00 X.224
G01 X.184
G01 X.204 Z-.3299
G00 X.274
G00 Z-.3447
G01 X.214
G00 X.254
G00 X.224
G01 X.184
G01 X.204 Z-.3397
G00 X.274
G00 Z-.3453
G01 X.23
G01 X.25 Z-.3403
G00 X.254
G00 X.184
G00 X.2348
(Machine Setup - 1  Turn Groove Finish )
(TOOL #6    OD GROOVE .030 WIDE )
G00 X.5 Z-.3703
G00 X.25
G01 X.21 Z-.3503 F4.
G01 X.164 Z-.3497
G01 Z-.3103
G00 X.254
G00 Z-.2899
G01 X.214 Z-.3099
G01 X.164
G01 Z-.3103
G00 X.6
G00 Z-.3
(Machine Setup - 1  Turn Groove Rough )
(TOOL #6    OD GROOVE .030 WIDE )
G00 X.5 Z-.5982
G00 X.35
G01 X.3082 F3.
G01 X.3282 Z-.5882
G00 X.35
G00 Z-.6149

G01 X.29
G00 X.33
G00 X.3
G01 X.256
G00 X.296
G00 X.266
G01 X.222
G00 X.262
G00 X.232
G01 X.188
G01 X.35
G00 Z-.6349
G01 X.29
G00 X.33
G00 X.3
G01 X.256
G00 X.296
G00 X.266
G01 X.222
G00 X.262
G00 X.232
G01 X.188
G01 X.208 Z-.6249
G00 X.35
G00 Z-.6549
G01 X.29
G00 X.33
G00 X.3
G01 X.256
G00 X.296
G00 X.266
G01 X.222
G00 X.262
G00 X.232
G01 X.188
G01 X.208 Z-.6449
G00 X.35
G00 Z-.6749
G01 X.29
G00 X.33
G00 X.3
G01 X.256
G00 X.296

G00 X.266
G01 X.222
G00 X.262
G00 X.232
G01 X.188
G01 X.208 Z-.6649
G00 X.35
G00 Z-.6949
G01 X.29
G00 X.33
G00 X.3
G01 X.256
G00 X.296
G00 X.266
G01 X.222
G00 X.262
G00 X.232
G01 X.188
G01 X.208 Z-.6849
G00 X.35
G00 Z-.7149
G01 X.29
G00 X.33
G00 X.3
G01 X.256
G00 X.296
G00 X.266
G01 X.222
G00 X.262
G00 X.232
G01 X.188
G01 X.208 Z-.7049
G00 X.35
G00 Z-.7349
G01 X.29
G00 X.33
G00 X.3
G01 X.256
G00 X.296
G00 X.266
G01 X.222
G00 X.262
G00 X.232

G01 X.188

G00 X.6

G00 Z-.6

(Machine Setup - 1  Turn Groove Finish )

(TOOL #6  OD GROOVE .030 WIDE )

G00 X.5 Z-.790

G00 X.33

G01 X.29 Z-.770 F3.

G01 X.168

G01 Z-.6103

G00 X.33

G00 Z-.5724

G01 X.29 Z-.5924

G03 X.2579 Z-.6097 I-.0172 K-.0002

G01 X.168 Z-.6099

G01 Z-.6103

G00 X1.0

G00 Z1.0

G49

G90G00G54Z0

G00 X0

M30

%

(Machine Setup - 1  Turn Groove Finish )

(TOOL #6  OD GROOVE .030 WIDE )

T6 M6 G43

G00 X.6

g00 Z.1

G00 x.380 z.05

G01 Z-.6 F4.

G00 X1.0

G00 Z1.0

G49

G90G00G54Z0

M30

%

**Program Code with Explanations:**

G80 G94 G7 G40 G20 G18 G90 G54

(G80 cancels drill can cycles, G94 feed in In/Min, G7 lathe program in Diameter Mode, G40 cancels cutter comp, G20 program is in Inch, G18 program is for ZX-axis, G90 absolute mode, G54 is the base coordinate system)

T1 M6 G43 (this implements the length offsets from the tool data page for tool #1 without a move)

G00 Z.1 (this moves the distance in Z that is on the tool data page for tool #1 -.100)

G00 X0 (this moves the distance in X that is on the tool data page for tool #1)

G00 Z0

G00 Z.1 (move to clearance point in the Z-axis .100 in front of the part.)

G49 (This clears the length amounts for tool #1 without move)

G90G00G54Z0  (This moves the Z-axis by the amount in the tooling page to Z Home)

X0  (This moves the X-axis by the amount in the tooling page to X Home)

5. To run your program:
    a. Click on “File” and “Open”

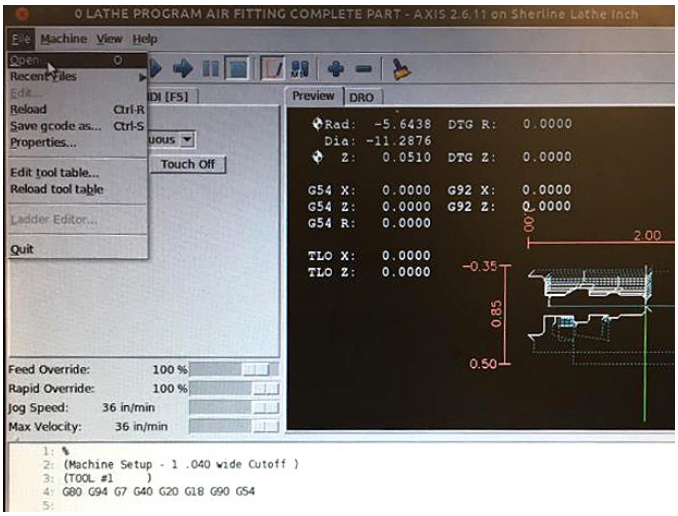


*FIGURE 21*

    b. Pick your program. If your program does not show, it may have been saved as a (.NC file) instead of a (.NGC File). If this is the case, choose “All Files.”

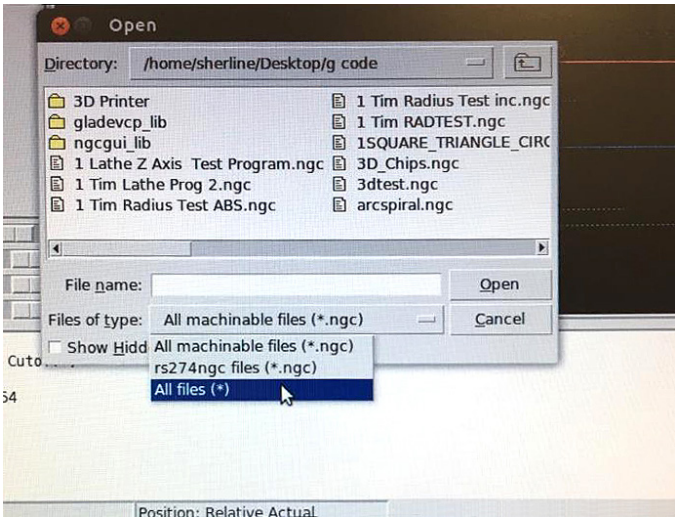


*FIGURE 22*

c. Run your program.

Hit the desired icon at the top (Start, Single Step, Pause, Stop) and run your program.

**Start Button:** This button starts your program.

**Single Step Button:** This button will run your program one line at a time and stop after each move.

**Pause Button:** When you click the Pause button, it will stop the program right where it is (it will not finish the move dictated by the line of code that is running). In order to resume the program, click the Pause Button again. Your program will now resume from the point where it stopped.

**Stop Button:** When you click the Stop button, the program will stop. However, you cannot resume the program. The Stop button will reset the program to the beginning. Then you will need to click on the Start button, and the program will Restart from the beginning.

***NOTE:** We "Highly Suggest" that you use the "Pause" button, instead of the "Stop" button.*

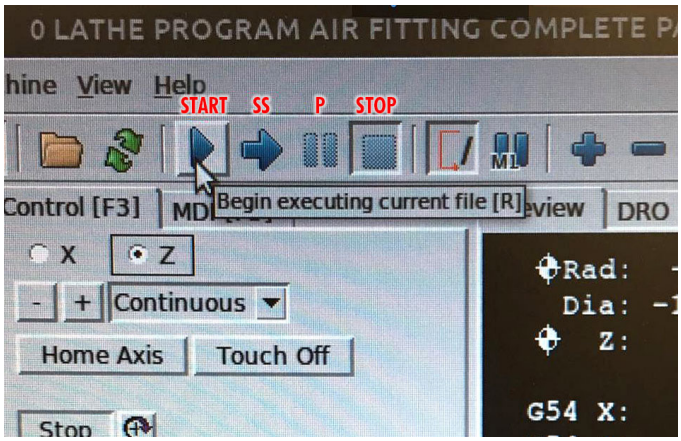


*FIGURE 23—The red words/abbreviations indicate their respective icons.*

Thank you,
Sherline Products Inc.

# Sherline 6600 CNC Chucker Lathe

## Exploded View and Part Numbers