

The purpose of this lab is to familiarize you with the Tree 310 CNC milling machine and the Dynapath controller. You will learn to setup the mill, load and edit programs, inspect programs graphically on the controller, set the part reference zero, and load tools. Come to lab prepared with relevant handouts, and appropriate shoes. Remember to sign in on the shop sign in sheet before beginning work. Plan to arrive 10 minutes early. The instructor will help you with the following exercises:

1. Familiarization with the machine layout, lubrication procedure, and the location of the power switches.
2. Clear the startup errors.
3. Move the tool to its home position.
4. Load the edge finder and set the part reference X&Y zeros at the top left hand corner of the **blank on the tooling plate**. Program "SPINDLE" can be used to start the spindle.
5. Set the part reference Z zero to the **top surface of the tooling plate**.
6. Load Tool 1, the 3/8" end mill.
7. Enter the example program listed below.
8. Check the example program using graphics.
9. Demonstrate the procedures for "proofing" a program by lowering the bed 1".
10. Run the example program in single step mode.
11. Run the example program in auto mode.
12. Review the procedures for loading programs from the PC to the Tree 310
13. Transfer program for the bike wrench from the computer to the Tree using MasterCAM.
14. Fabricate the part
15. Explain the requirements for cleaning the mill.

Sample program to practice editing programs (remember to use your initials in the name, e.g. DBEX1).

```
(EXAMPLE)$
N0001(T)SAMPLE PROGRAM           : Text event
N0002(9)M03S2500T04E01$         : spindle on
N0003(0)X2.0Y2.0Z1.0$           : rapid to 2,2,0
N0004(1)Z-.2/F5.$               : feed incrementally down 0.2" (use "incr" key)
N0005(1)Y-2.0R.5F10.0$          : first of four autoradius events
N0006(1)X-2.0R.5$
N0007(1)Y2.0R.5$
N0008(1)X2.0R.5$
N0009(1)Y-1.0$
N0010(1)Z1.0
N0011(0)X0Y0$
N0012(9)M30$
```

1. Turn machine on.
2. Lubricate the machine.
3. Clear errors.
4. Set machine reference zero.
5. Set part zero: X, Y, Z.
6. Enter program (keyboard or computer).
7. Check program using graphics.
8. Proof part with bed lowered.
9. Make part.
10. Set machine to Home position.
11. Turn machine off.
12. Clean machine.

1. Turn Power On

- a. Turn circuit breaker at the lower left rear of machine on.
- b. Turn toggle switch on right side of controller (orange box with keyboard) on.
- c. Wait for monitor to warm up.
- d. Check the lubrication plunger. It should be in the up position, if not pull it up.

2. Set Reference Position

- a. Clear error. **Mode Select/Fault/Status, Reset, Reset**
- b. Enter jog mode. **Mode select, 0**
- c. Make sure that the machine is at the home position (about -.050" from the reference marks on the X,Y, and Z axes. This is the position the machine should return to automatically at the end of a program. The X axis reference mark is an arrow on the front of the bed, the Y axis is an arrow on the right front corner of the saddle). If it is not move the axes using the jog commands (**Mode Select, 0 - Mode Select, 8**). When all axes are about .050" away set reference: **R, X+, Y+, Z+**. Each of the axes should jog to its reference position.

The reference mark for the x-axis is an arrow on the front of the bed. Move the bed about .050" to the right of the mark. The reference mark for the y-axis is an arrow fixed to the right front of the knee. Move the bed about .050 away from the arrow (-y direction). The z-axis is at the reference position when the bottom of the chromed portion of the spindle is flush with the bottom of the bearing housing. Move the spindle about .050" down from this position.

3. Set Part Zero

- a. Fix part on bed of mill.
- b. Clean the taper on the toolholder and quill.
- b. Place the edge finder in the spindle.
- c. Turn on the spindle:
 1. Go to the catalog: **Mode Select, C**
 2. Choose program "Spindle": **S, P, I, N, D, L, E, Enter**
 3. Run the program in single event mode: **Mode Select, 2, Cycle Start**
- d. Jog the edge finder to the x-axis edge of the part. (Don't crash!) Lift the edge finder above the part. Jog to compensate for the edge finder radius (.100") **Mode Select, 7**.
- e. Set X-axis part zero. **9, X+ switch,2**
- f. Repeat for Y axis.
- g. Turn spindle off: **Mode Select, 2, Cycle Start**.

4. Set Z-axis Reference
 - a. Remove edge finder and insert reference rod into spindle.
 - b. Manually raise or lower bed as necessary to provide about 0.5" of clearance between reference rod and the reference Z surface of the part. Use of very long tools such as drills may require a larger gap.
 - c. Slowly jog the spindle until the end of the reference rod is close, about .050", to the part surface.
 - d. Slowly raise the bed until the gap between the reference rod and the work piece is .010", as measured with a feeler gauge.
 - e. Raise the bed another .010".
 - f. Set Z-axis zero: **Mode Select, 0, 9, Z+**
 - g. Go to the home position: **H, Z+, X+, Y+**

5. Proof Program
 - a. Lower the bed enough to prevent the tool from touching the work piece the minimum programmed z depth. (e.g. if a depth of z=-2.1 inches lower the bed about 2.6 inches)
 - b. Load desired program: **Mode Select, C, program name, Enter**
 - c. Run the program in single event mode: **Mode Select, 2, Cycle Start**
 - d. Press cycle start to execute each block. Pay careful attention to -Z-axis moves to prevent crashes.
 - e. When completed, raise the bed back to the reference position.

6. Run the Program
 - a. **Mode Select, 1, Cycle Start**

7. Power Off

8. Clean the Machine

A. Prepare mill to receive file

Mode Select, 9

Check communication status.

4800 baud

Parity ON

Control Character OFF

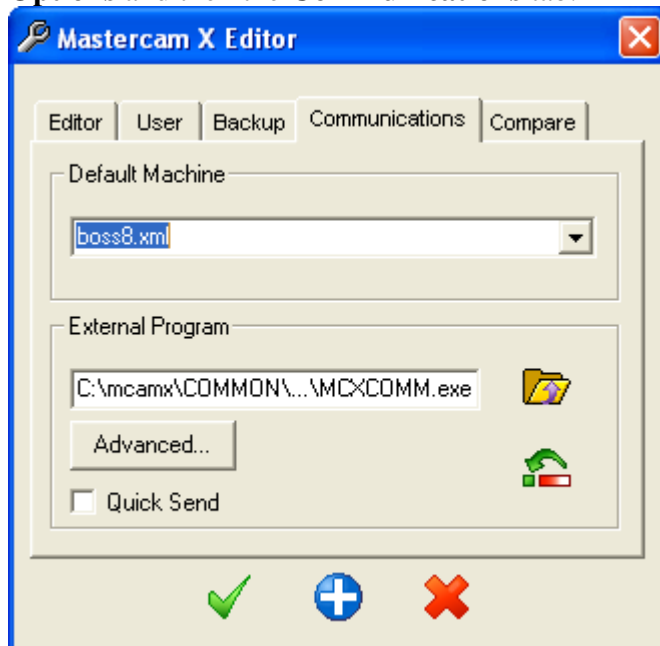
Echo OFF

Program Load (soft key)

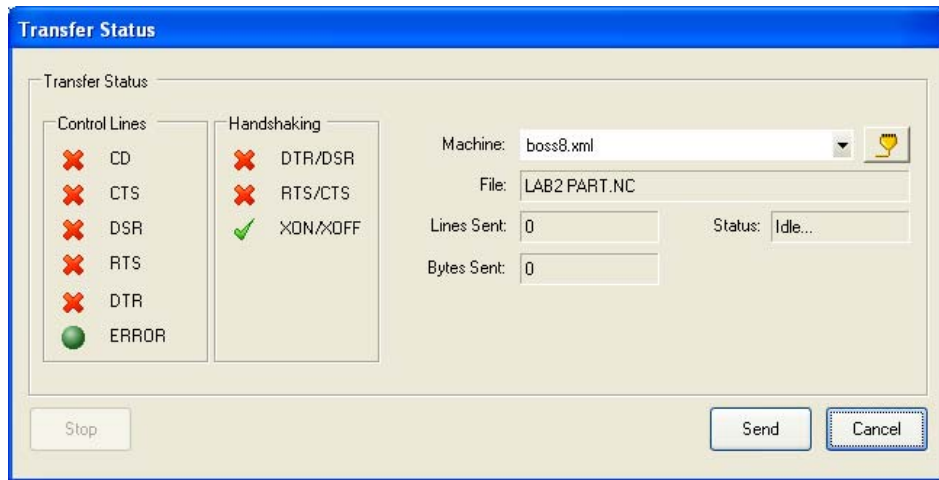
Display reads "Busy Loading"

B. Set up the computer to send the file

1. Open your .nc file in MasterCAM X's Editor. If it doesn't open automatically after posting your job, you can start the editor by selecting **File, Edit/Open External** in MasterCAM.
2. Choose the communication protocol. From MasterCAM X's Editor Menu select **Tools, Options** and then the **Communications** tab.



3. Select the **Boss 8** protocol from the default machine menu and click the green check.
4. Select **Communications, Send** or press the send icon. The following dialog box will open:



5. Press **Send**

- C. While loading the block numbers will flash in the lower right corner of the Dyanpath controller. After the entire program has been loaded the mill display will indicate "done". If no end statement was included as the last block of the program the Abort Load soft key must be pressed to end.
- D. After loading
Mode Select, G to use graphics, **Cycle Start** to start the program
Mode Select, 4 to view program
Mode Select, 3 to edit the program, press **Reset** when finished editing
- E. Common problems
1. Format Fault is usually caused by: spaces between words in a block (a typical block should read N0020X2.0Y1.0F10\$) or by blocks without \$ at the end. This error can also be caused by o's in place of zeros. The program name must be in capitals, with eight or fewer characters.
 2. Duplicate I.D. fault occurs when the name defined in the first line of the program is the same as one already in the catalog. If the offending file is your own it may be deleted (mode select, C, program name, delete program (soft key), or the name in the program must be changed.
 3. Communication or Parity errors are usually caused when communication parameters at the mill and computer are not the same. Check that the mill parameters are those described in section A. The computer parameters can be set by typing mode com1:1200,E,7,1 Return. The computer should respond "com1:1200,E,7,1,-". Sometimes these errors are caused by random glitches, so try loading the program. If it persists lower the baud rate to 300 baud at both the mill and the computer.