



## Expanding Memory on your Fadal CNC

### Step 1:

Send the machine to the home position by one of the 2 ways listed below.

- From the “Enter Next Command” prompt type “SETCS”, press [ENTER] then type “HO”, and press [ENTER].  
*Or*
- Navigate to the “Functions” menu, select Option “4” (Home Axes), then select Option “2” (Return for Power Off).

Then you will need to go into MDI mode and make sure Tool #1 is in the spindle. If it is not, type “T1M6”, press [ENTER], then press the green [START] button.

### Step 2:

Use our CNCPros-DNC Software to backup the parameters, backlash values, parts programs, and tool offsets if your software will permit. (*Earlier versions did not have this feature, so write down the values on the included information sheet if it does not.*)

View the parameters:

- At the “Enter Next Command” line type “SETP”, and press [ENTER]. This brings you into the parameters screen. Press the [P] button to go to page 2, then [P] again for page 3.
- After you write everything down then press the [MANUAL] button to escape.

View the backlash values:

- From the “Enter Next Command”, type “BL”, then press [ENTER] to display the backlash values. There are 3 values for X, 3 for Y, and only 1 each for Z, A, or B. Record on worksheet provided.

Now you are ready to backup your programs, parameters and backlash, and tooling offsets.

- Under the “Quick Keys” menu, press Option “9” (Punch)
- Press Option “2” (Options)
- Select the [RECEIVE] button in the CNCPros-DNC Software
- On the CNC, select “0” (Program and Tooling)
- Press “0” again, and press “0” one more time.
- Save the file to your computer

(Repeat the previous steps to backup your parameters and backlash. Instead of selecting “0” (Programs and Tooling), you will need to select Option “4” (Parameters and Backlash.)



After verifying that your programs and offsets are saved onto your PC or worksheet, you have recorded your 2<sup>nd</sup> home values (if used), parameters and backlash values, and you have Tool #1 in the spindle, you are safe to power the machine down and install the module onto the CPU card in Slot #5. The module is a “piggy-back” board and will only fit one direction onto the board.

At power up, you will most likely receive an alert to the effect:

**YOU HAVE A MEMORY UPGRADE OR MEMORY CORRUPTION. PLEASE ZERO THE MEMORY.**

**With the machine power off, take a photo of your card rack and all wires connected to the PC boards. One by one, remove the cards, disconnect the cabling, and use the CNCPros.com supplied pink eraser to clean the gold tabs on the edge of the cards. This is the electrical contact for the PC boards and how each board communicates with the other boards and machine devices. Cleaning the card edge connectors eliminates most of the phantom faults and alarms seen with thermal changes in your shop. After the edge is clean and shiny, replace the card. Do this to all cards in your rack. The photo you took when you started will serve as a guide if you forget where the wires are connected.**

### Step 3:

Now you are ready to perform the Zero Memory Procedure.

- From the “Enter Next Command” line type “DI” press [ENTER] then type “G0 3000” (digit after G is the numeral 0, not the letter O) then press [ENTER]. (You are now in diagnostic mode on Page 1.)
- Press the [SPACEBAR] one time to advance to Page 2. Then press Option “5” (Zero Memory). Press [Y] to answer yes, press [Y] to answer yes a second time, wait several seconds, then press Option “2” (Start CNC).
- Power down the machine, wait about 3 seconds, then power the machine back up.

### Step 4:

Reload your CNC machine with all the programs from the CNCPros-DNC software.

- First, you will need to change the Baud rate to the same as you had before. Choose Option “9” and press [ENTER], then cycle the power on your machine and perform the cold start routine.
- On the CNC, type “TA,1” and press [ENTER].
- In the CNCPros-DNC software, press [SEND] to transfer the information back to your CNC.
- Double check the parameters and backlash sent to your CNC from the software by typing “SETP” on your CNC, then press [ENTER].



If you manually input the parameters and backlash in your CNC machine follow the instructions below:

- Each time you enter a value and press [ENTER], the value will be changed and the cursor will advance to the next question. Most questions have the potential answers listed on the bottom half of the screen. You will input the appropriate numeric value associated with each answer. When all parameters have been entered, press the [MANUAL] button and continue below:
- To insert the backlash values, use this format:
  - BL,1,value,value,value (for X)
  - BL,2,value,value,value (for Y)
  - BL,3,value (for Z)
  - BL,4,value (for A)
  - BL,5,value (for B)
- Power your machine off for 10 seconds again. On power-up, it will now recognize your parameters. If you have a 4<sup>th</sup> or 4<sup>th</sup>/5<sup>th</sup> axis, you will now have to enter the BL,4,value and the BL,5,value.
- After cold start, at the “Enter Next Command” line type “SETTO”, then press [ENTER] to reset the ATC to Tool #1. Jog any axis to its 2<sup>nd</sup> home location, then use the command SETX/SETY/SETZ accordingly.
- Follow the instructions in step 4 to upload all saved programs and tooling.

### Step 5:

Set your tool holder to Pocket #1.

- At the “Enter Next Command” line, type “SETTO” and press [ENTER].
- Go into “MDI” mode and you will see that Tool #1 is in the spindle.
- Type the “ME” command and your screen will now show the new expanded memory.