

John Robbins
Automation Consultants, Inc.
4491 Foskuhl Rd. Floyds Knobs, IN. 47119
812 923 3718 812 923 5396 (fax)

Restoring the Memory on an Excel CNC with Fanuc 0 Control

About once or twice a month I get a telephone call from a shop owner asking what a RAM parity alarm is. He explains that "The alarm just showed up when I turned the machine on. I powered down and back up several times but it won't go away."

"Well", I explain, "Basically, it means that the memory area on the CNC is shot. All of the data is corrupted. You need to clear the memory and restart. Just like doing a format on a PC hard drive."

This is always followed by the same question. "Can I save my programs?"

"No", I explain, "The programs, along with all of the machine parameters, are already lost."

It's at this point that the shop owner realizes that the hundreds of hours he has spent programming a multitude of parts has just vanished.

Now there are two options at this point. The first option is for the owner to clear the memory area and reload all of the data from a backup. A typical machine can be brought back to original condition in less than one hour if there is a good backup. If there is no backup then he must use the second option. That is to reprogram all of his parts. This can only begin after the machine parameters have been reinstalled. Have you got a copy of these parameters handy?

It's fairly obvious that option number one would be the easiest. For this to be able to happen it is critical that a good backup of the machine information be available. There are a large number of machine controls out there. Each has its own procedure to backup and restore data. Over the next few issues I'll provide you with the procedures to complete this task on as many controls as possible. I'm going to start with the Fanuc 0 control. This is the most widely used control manufactured by Fanuc.

To begin you must have a PC connected to the RS232 port and able to transmit and receive programs. We'll start by backing up all of the data. After that I will show you how to reload everything. Please note that some machines may not have all of the options we discuss. It is also important to note that the Fanuc operating system has different functions depending upon the level of software. Think of this as the difference between the way Windows 3.1 and Windows 2000 differ. I'll try to cover as much as possible. If you have any additional questions or comments please contact me at my office.

4491 Foskuhl Rd.
Floyds Knobs, IN. 47119
812-923-3718

John Robbins
Automation Consultants, Inc.
4491 Foskuhl Rd. Floyds Knobs, IN. 47119
812 923 3718 812 923 5396 (fax)

Backing up the machine data

1. Connect the RS232 cable between the PC and the CNC. Start the communications package that you wish to use. Set the package up to receive the parameter file.
2. Place the machine in EDIT mode with memory protect off. Press the PARM key to display the parameter screen. Verify that the top left corner of the screen reads parameter. While holding the # key in press the green OUTPT key. This will start the parameter file transmitting from the CNC.
3. After the parameters are finished press the PARM key again to display the diagnostics page. Set the PC up to receive the diagnostic file. Hold the # key in and press the green OUTPT key. This will send all of the diagnostics out to the PC.

Some early versions of the 0 control did not allow for the transmitting of the diagnostics. If your machine will not allow this, record the diagnostics manually. Start with diagnostic 300 and record every fifth value. (300, 305, 310, etc.) Do this up to 395. Record all of the 400 and 500 level diagnostics.

4. Set the PC up to receive the programs. We will output all of the programs as one large file. If you are using any 9000 macro programs you must set parameter 10 bit 4 = 0 in order to output these programs. Press the PRGM key until the program page is displayed. Type in O-9999 (use the letter O not the number 0) and press the green OUTPT key. All of the programs in memory will be transmitted.

The newer versions of Fanuc software make use of an I/O soft key. This key is located under the CRT on the machine tool. If your machine has it press the I/O soft key then type in O-9999. After this press the PUNCH soft key. This will transmit the programs.

5. If you wish, you can also output the tool offset data. To do so set the PC up to receive the file. Display the tool offset page and press the green OUTPT key. Note that the work offsets will be recorded with the parameters when they are output.
6. There are several items that should be recorded manually. These include any macro variables in the 500 or above range. You should also record all of the 900 parameters. These will be needed when reinstalling the memory.

Now you have all of the data backed up your safe. Well, maybe not. You may wish to look at the data using a standard text editor. Make sure that all of the information looks proper without any unusual characters in it. It can be a royal pain if you go to restore the data only to find out that it wasn't saved properly on the PC. Copy the files you saved to a disk for additional protection. You don't want a hard drive crash on your computer causing you to redo everything.

4491 Foskuhl Rd.
Floyds Knobs, IN. 47119
812-923-3718

John Robbins
Automation Consultants, Inc.
4491 Foskuhl Rd. Floyds Knobs, IN. 47119
812 923 3718 812 923 5396 (fax)

Restoring Memory on Fanuc 0

1. Hold the RESET and DELETE key while powering up. **This will erase all memory in the machine!**
2. The machine will power up with a number of alarms. Press the e-stop button in.
3. Press the PARM key- go to parameter 900 and enter parameters 900 through 917. A "000" alarm appears after entering the first parameter. Just go back to parameters and enter the values for 901. After you enter parameter 901 a message will appear asking you to delete the memory area. Press the DELETE key.
4. Enter the remainder of the 900 parameters. When finished verify all parameters were entered correctly.
5. Power machine down for 10 seconds then power up
6. Go to parameter page #1 in MDI mode and set ISO = 1
7. Page down to parameter #2 Enter 0000 0000 for one stop bit. Use 0000 0001 for two stop bits.
8. Go to parameter # 552 set it to 10 for a baud rate of 4800. Set it to 11 for a baud rate of 9600.
9. Place machine in edit mode with the memory protect key off. With the parameter screen displayed push the INPUT key. LSK should appear in the lower right corner of the CRT.
10. From the PC transmit the parameter file to the CNC. The LSK flashing on the CRT should change to INPUT. After the file starts transmitting. It will take several minutes to read the file.
11. Power the machine down and up again. Verify parameter 2 bit 0 and parameter 552.
12. Go to the first page of the parameters and set the machine to the increment system you wish to use, inch or metric.
13. Go to diagnostic page- Verify Diagnostics is displayed in the upper left corner of the CRT.
14. Hold EOB in and push INPUT. Transfer the diagnostics file to the CNC.
15. Verify that parameter 10 bit 4 = 0

John Robbins
Automation Consultants, Inc.
4491 Foskuhl Rd. Floyds Knobs, IN. 47119
812 923 3718 812 923 5396 (fax)

16. Go to Edit mode and push the PGRM key. Push the Input key and LSK should start flashing on the CRT. Transfer the program file to the CNC. If the screen has the I/O soft key push it then the READ key. LSK will appear. Transfer the program file. When finished go back to parameters and set parameter 10 bit 4 = 1. After this set PWE=0. Push reset to clear the alarm (must be in MDI mode)
17. Verify the inch/metric setting on the first parameter page.
18. Push the OFFSET key until tool offsets are displayed. Press input. LSK will appear Transfer the offset file. Verify all offsets. (This will only work if you have the G10 option setting.)
19. Enter any extended work offsets P1-P8 by hand.
20. Power down Power up holding the LQP and CANCEL key. Move away form home position several inches and zero out at 25% rapid.
21. You should be ready to operate the machine. I know it sounds like a lot of steps but it sure beats programming all of those parts again.