



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

**Product Family: Directsoft**

**Number: AN-DS-003**

**Subject: Troubleshooting procedures for Directsoft**

**Date Issued: 2-2-04**

**Revision: Original**

This application is intended to give the Directsoft user instructions in the various tools for troubleshooting ladder logic programs using Directsoft. Topics covered will include the DSlaunch application launch screen, ladder logic view, and the dataview screen in Directsoft.

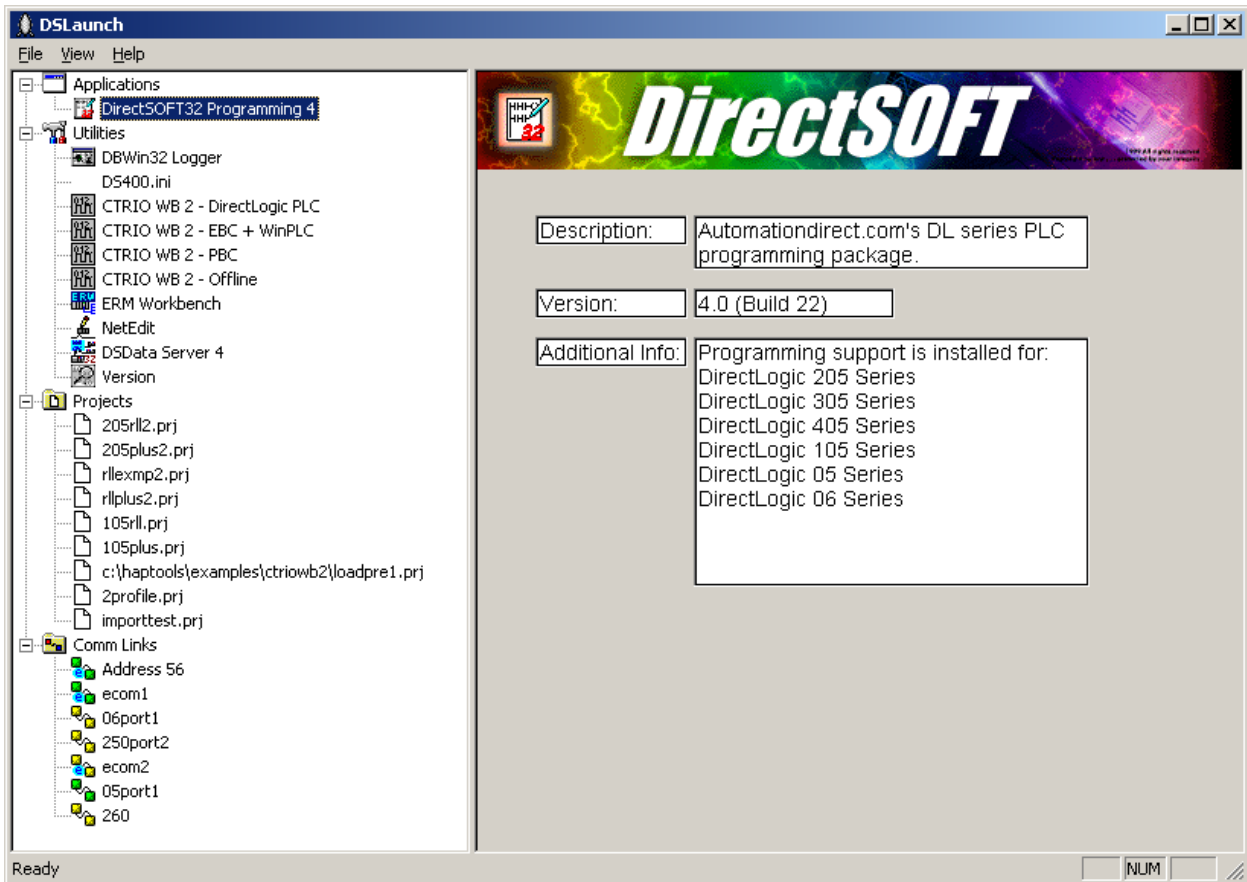
Next page please.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

First of all, anytime you are going to do any programming or monitoring you should always start by opening DSlaunch. This is the application that is used to launch all Directsoft related software.



Above we see a screenshot of DSlaunch. Notice here you can see the current version of Directsoft, what processors this version supports, which utility packages are installed, any projects that have been previously opened on this PC, and our existing communications links. As a bonus, DSlaunch checks to see if any of the links are available when it is opened. If you look closely you will see that the 'address 56', 'ecom1', and '05port1' links all have green boxes. This means that those links indicated a processor was present.

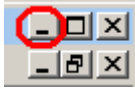
To open an existing link from here you can either just double-click it on the projects list, or, if it is not shown on the projects list you can right-click on the projects icon and select BROWSE to go find the project. Folks that open projects



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

directly without using DSlaunch may not realize this, but there is no reason you cannot open more than one project at a time. Try it, double click a project from the project list. Now minimize that project by selecting the underscore button in the upper right corner of the programming window.



You should see the DSlaunch menu again. Now open another project. You can now switch between the projects by selecting the appropriate window on your taskbar or using ALT-TAB to move between windows. This is very helpful if you need to copy/paste logic from project to project or simply want to visually compare projects. You can also select the windowed version of both open projects and view them side by side if you have a large enough monitor.

OK, go ahead and close any open projects and get back to the DSlaunch screen. For this next section you will need to have a PLC connected to your PC. If you have never made a link before you will have to create one. To do this from DSlaunch simply right-click the Comm Links icon and select add link. Here you'll be given a choice between various comm ports, modem, or ethernet.

Note: You should know that if you decide to use serial links, just because a comm port shows up on the list in the link wizard does not mean it exists as a resource on your PC. Directsoft is set up by default to look at comm 1 through 4, you still need to know which physical port on your PC is which comm port. You can expand the number of comm ports Directsoft will look for if you need to in the DS400.ini file in your Windows or Winnt directory. If you decide to use ethernet, you need to be sure you have added the IPX/SPX compatible transport protocol to the bindings for the network interface card on your PC before launching DSlaunch. IPX/SPX is the default protocol for both Directsoft and the Netedit utility.

So pick whichever connection type you want to use as well as the appropriate protocol and address. Directsoft will search for the appropriate baud rate and parity, or it will let you select the appropriate ethernet device from a list. Once the link has been successfully tested, Directsoft will ask you to name it. Click finish and the new link should show up on the DSlaunch screen under Comm Links.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

Now that you have a functional link, let's talk about links and projects. Links are just names for a given set of communication parameters. The link info contains the PLC type, baud rate, parity, PLC address, and protocol type. Ethernet links contain similar information for ethernet connections. But the link does not know which program is in the PLC. On the other hand, if you save a project while you are connected to a particular PLC, the project will remember that and try to connect to the same link if you open that project again in the future. For this reason we recommend you name links with names reflecting the parameters they have. '05port1' or 'ecomaddress1' are good names for links whereas 'filtergallery' might not be as good a choice. This is not to say you cannot have a link specific to each project you create, with the same name, but only to say that this is not necessary.

One of the neat things about the DSlaunch screen is that it gives you the choice of two different methods of connecting to the PLC. Let's say you have gone to a customer's site and they want you to make some changes to their code you wrote for them. The first thing you might want to do is make a copy of just the code that is on the PLC. One way to do this is to simply double-click the appropriate link on the DSlaunch screen. This will connect to the PLC and read whatever code is in the PLC and display that code on the ladder programming screen in Directsoft. The project name will be shown as UNTITLED and there will be no documentation associated with this project, but doing this lets you save the existing project as-is before making any changes.

Your other choice for connecting to the PLC is to open an offline project and then select PLC and CONNECT and point to the appropriate link. This can be accomplished all at once by double-clicking a project from the project list that was saved while online. In either case Directsoft will connect to the PLC, read the program from the PLC and compare it to the program saved on disk. If there are differences, the software will ask you which one you want to use. If you select USE PLC then you will have the PLC ladder logic displayed on your screen with the documentation from the project you opened from disk. Be aware that this project has the same project name as the one you opened, even though the code is now different. You might want to consider immediately saving this project using the SAVE AS feature and giving it a different project name to prevent accidentally overwriting what was on the disk before...unless of course that was your goal.



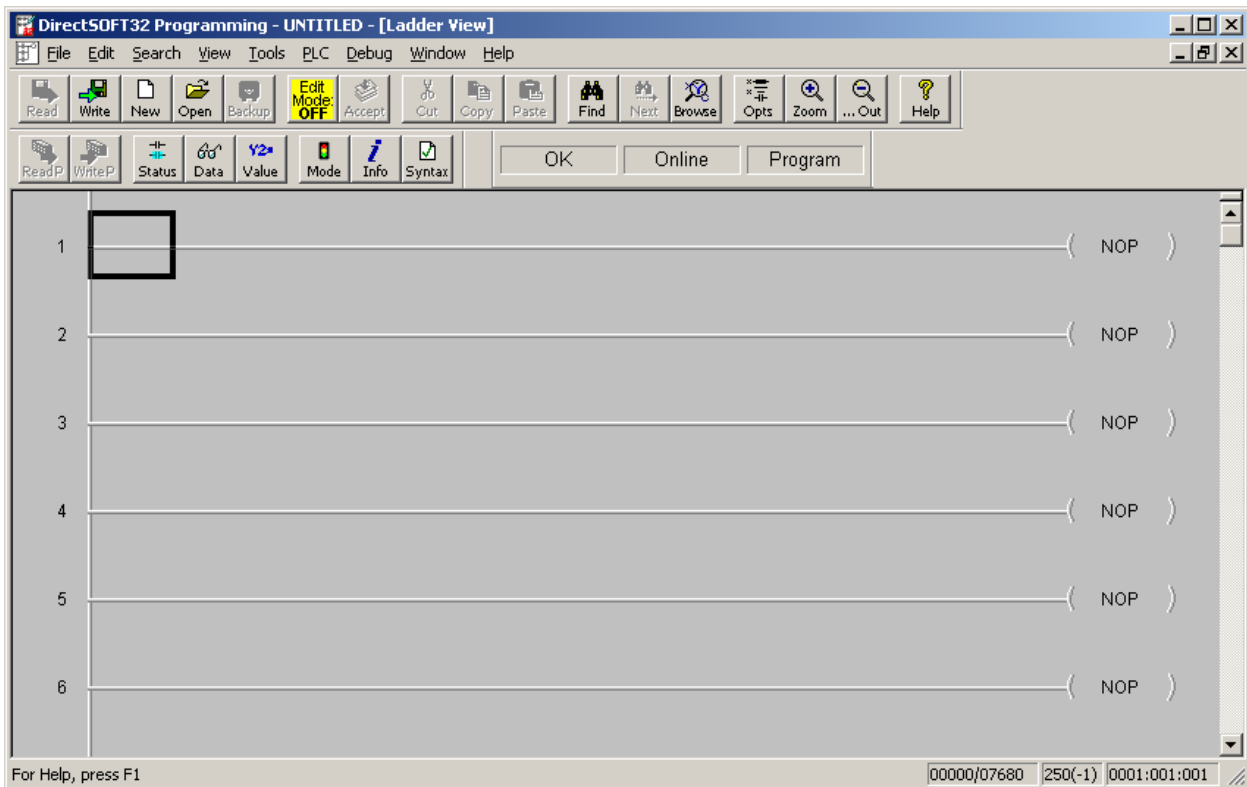
**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

If you select USE DISK then you will be looking at the code from the disk with its documentation, BUT, this code is not in the PLC at this point. You would need to select the SAVE PROGRAM or SAVE PROJECT TO PLC option to download the code/project information to the PLC.

So what is the difference between SAVE PROGRAM and SAVE PROJECT? In a nutshell, the SAVE PROGRAM option saves ONLY the ladder logic to either the PLC or the disk. The SAVE PROJECT option saves the ladder logic and all V location and setup information as well. If you want to be sure you get it all, use SAVE PROJECT.

OK, so now you have either double-clicked a link or opened a project that has connected you to the PLC. Your screen probably looks something like this.



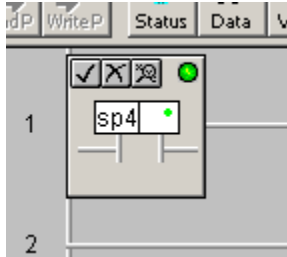
You might have code in there if this is an existing PLC application. This is the ladder logic programming screen, where most programmers spend their time. Let's add some code to this program and show off some features you might not be aware of yet at the same time. Most folks are aware of using the F2 key to enter normally open contacts, but you don't have to do this, you can simply place the



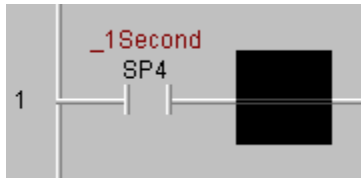
**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

edit cursor where you want the contact and enter the address. Here I am entering a special purpose relay that is on half a second and off half a second, so I simply type SP4 and hit ENTER.

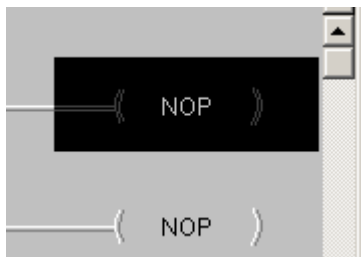


And I end up with this.



Now I'm going to enter an output coil in this rung, this output will be on half a second and off half a second. To enter the coil I select the NOP and then I could hit F7, select the coil tab, then select coil and press enter. But since I know I want an output coil, I can simply move the cursor to the NOP and type in OUT and press enter. Directsoft will open a window allowing me to enter the address I want this coil to have and then I press ENTER again.

Cursor to the NOP

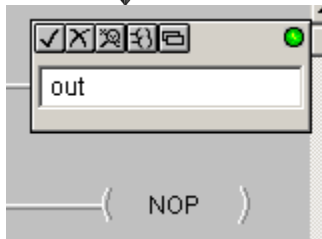


Type OUT

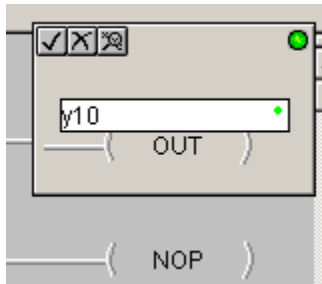


**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

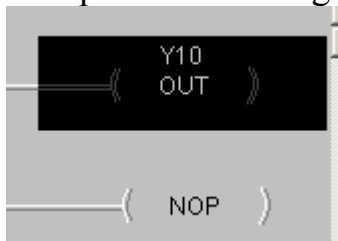
These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.



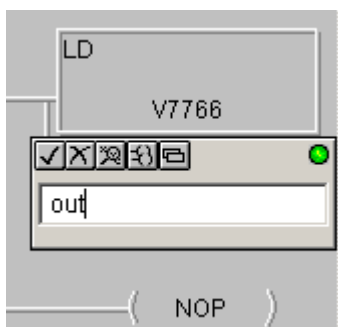
Press ENTER and enter the output address



And press ENTER again to finish.



This method of entering devices by typing their instruction name works for all instructions except one...the OUT box instruction. So if you need to enter an out box there is an extra step, let's go through an example. Let's say you want to move the value from V7766 (number of seconds on the clock) to V2000, and you want to do this every scan. We start the rung with SP1 and then add a LD V7766, press and hold the CTRL key and press the down arrow to add a vertical short, and then type OUT and press ENTER, just like we did with the coil.

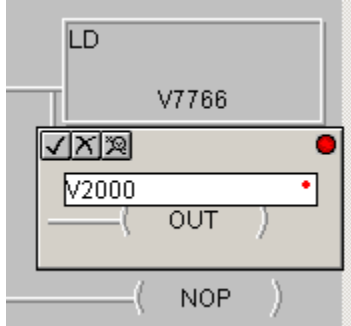




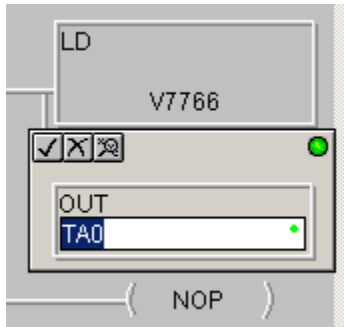
**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

If you try to enter a V location, like V2000, you get an error light.



This is because V locations aren't legitimate addresses for Coils. Now press the ? or / key on your keyboard. Don't hit the shift key, just press the button. You should see the instruction change to an OUT box instruction with a TA0 address. You can now enter the V2000 address and press ENTER.



So once you get used to the instruction names you use the most you can start entering them directly when programming. Pretty neat, huh? Now granted, some instructions like the Boolean compares (>, =, <=) are still easier to select from the toolbar, but for the common instructions this ought to speed things up.

Go ahead and add an END to this program if you have not already. Just go to a blank rung NOP and type END and press ENTER twice.

To accept the rungs you have edited (the ones with the yellow bar next to them) you can press the accept icon on the toolbar or just press the F8 key. You should then see a green and a blue line appear next to the accepted rungs. The blue bar indicates that these rungs have not been downloaded to the PLC yet, so we need to either press the Write P icon on the toolbar or select FILE and WRITE PROGRAM and TO PLC to download the program changes to the PLC.





**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

So now there is a simple program in the PLC and we need to troubleshoot it. The first and simplest thing to do is turn on status in the ladder view. You can do this by selecting the status icon on the toolbar, or by selecting **DEBUG** and **ALL STATUS ON**. This will cause the ladder view to display the current status of any contacts with contacts which are ON showing up with a light blue box around them.

Go ahead and place the PLC in RUN mode. This can be done using the traffic light **ICON (mode)** on the toolbar or by selecting **PLC** and **PLC MODES**. Once in RUN mode you should see the SP4 contact and the associated output blinking on and off. A common error that occurs at this point is a fatal error E401. If you get a fatal when going into RUN mode, it is likely this error. You can check the error type by selecting the blue info icon on the toolbar or by selecting **PLC** and **DIAGNOSTICS** and **SYSTEM INFORMATION**. E401 is a missing **END** instruction, if this happens and you have an **END** in your program, you probably forgot to accept and/or download your program to the PLC. Remember, what you see on the screen is not necessarily what is in the PLC.

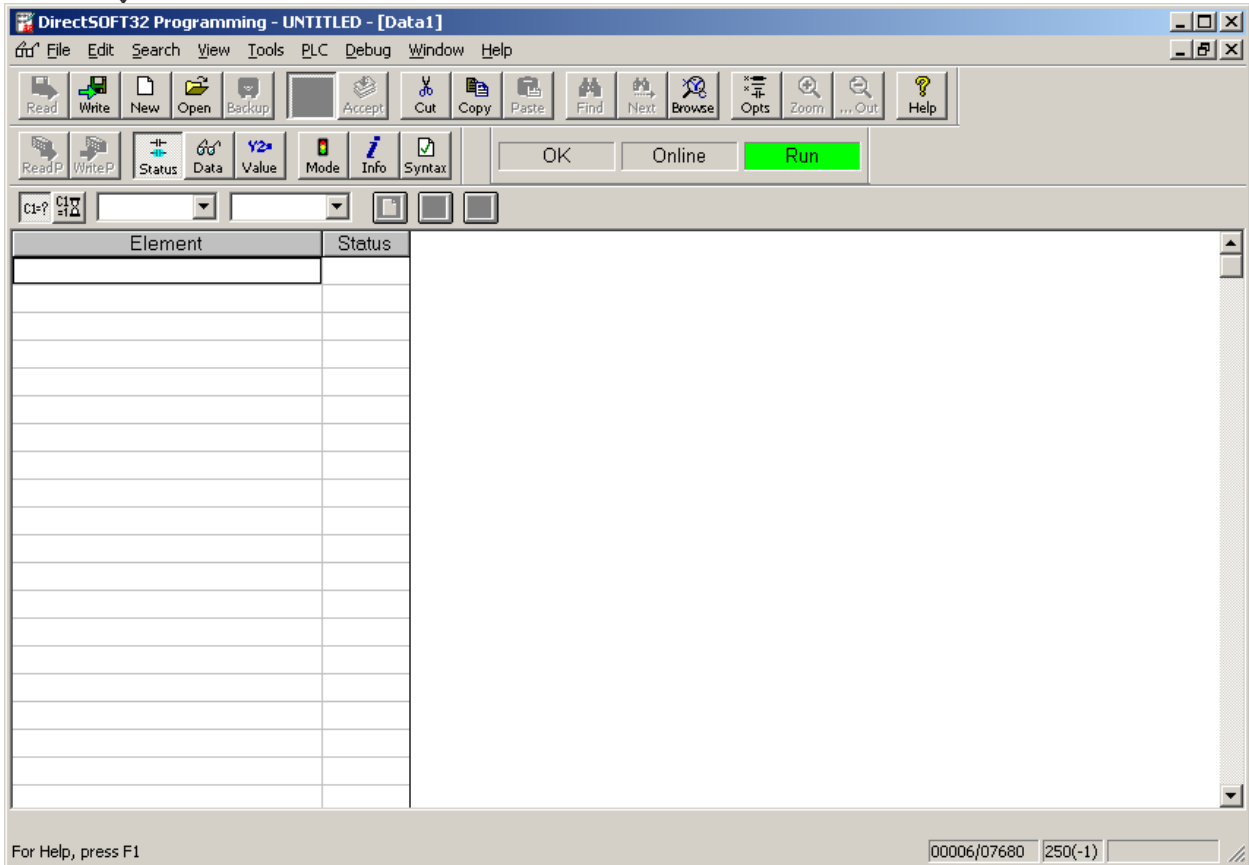
OK, so we can see bit-wise what is going on in this simple program, but what about those V locations? The ladder view doesn't show us their status. Now if we had any timers/counters, then we would see their values, but there is a better way.

Let's open up a dataview window. Select **DEBUG** and then **DATAVIEW** and then **NEW**. You should have a new dataview window open. The new window may be taking up the entire screen. This causes a lot of confusion where folks think they need to open a new dataview window each time they need one. You do not have to do that, and in fact it is a bad idea. One of the nice things about Directsoft being a native Windows application is that you can tile screens, so let's try that. Right now you probably have this.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

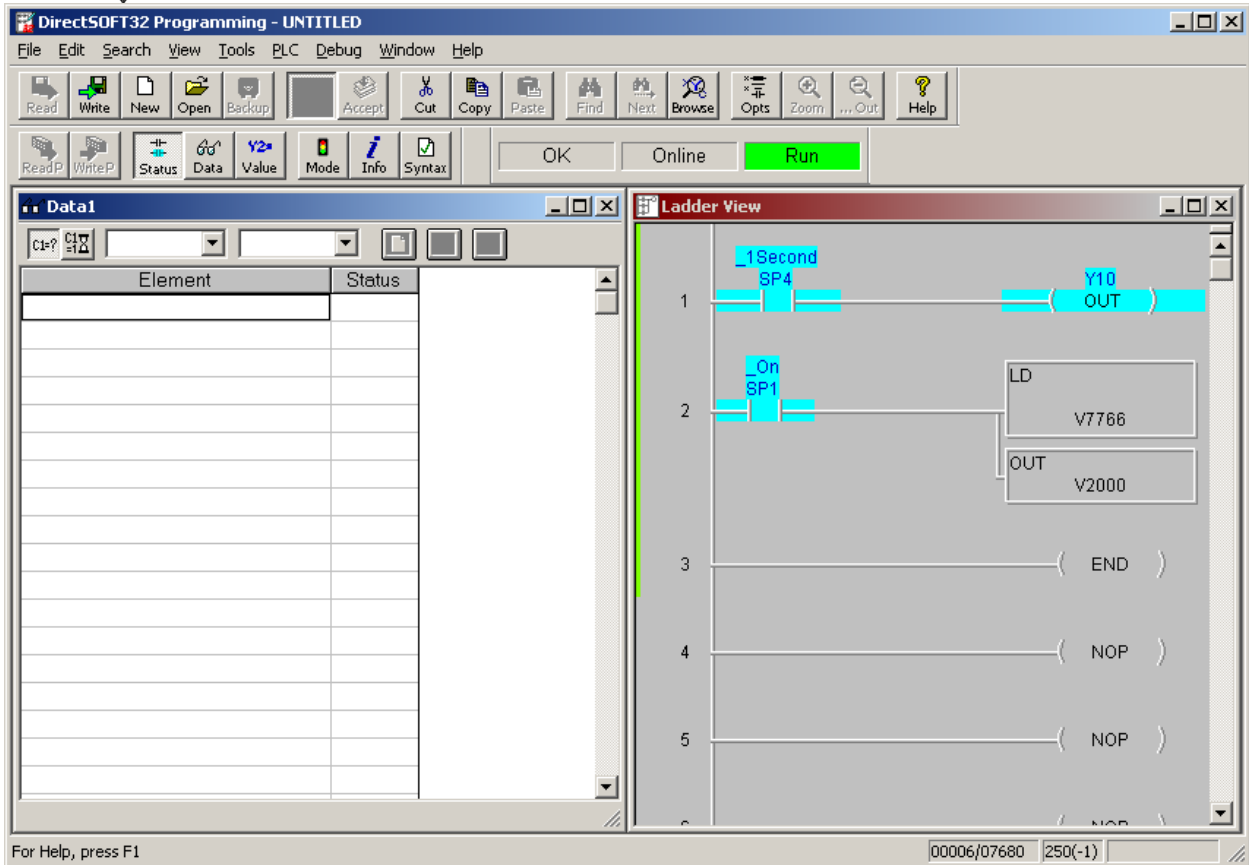


Select WINDOW and then select TILE VERTICALLY. Now you should have this.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

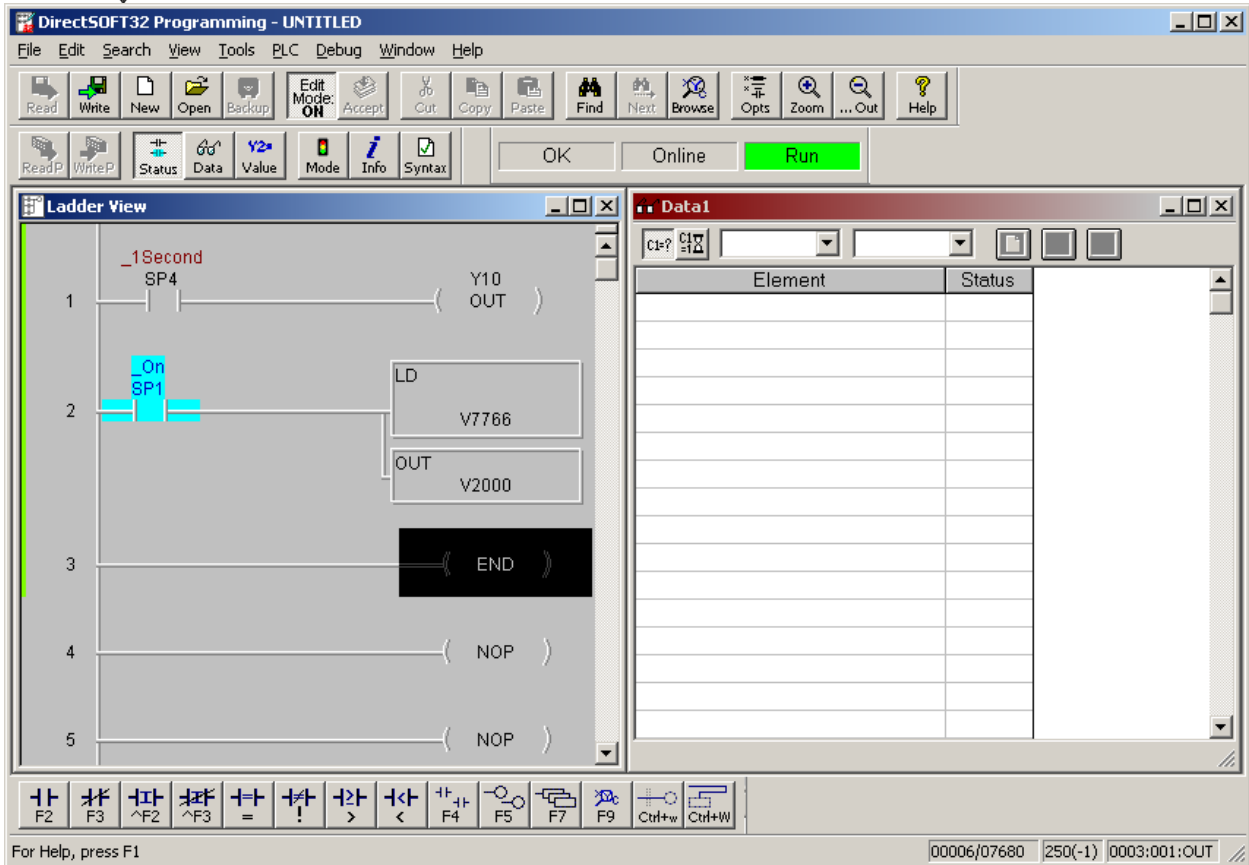


Which window gets put on the left is determined by which window is selected, so if you want the ladder view on the left, select it and select WINDOW and TILE VERTICALLY again.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.



Much better, OK now let's actually use that dataview window. Select one of the element boxes in dataview and enter `V2000` and hit enter. You should be able to see data changing from 0-59.

Element	Status
V2000	41

Notice that if you select the `V2000` box again, we get a display at the top of the screen showing the data format for this register. In this case it is in BCD/HEX (the default) and we are viewing a single word.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

Element	Status
V2000	40

Dataview also lets us view the same registers in different formats, so select the box under V2000 and enter V2000 again and hit enter. Now select the second V2000 box and then press the down arrow button next to BCD/HEX above and change the format to DECIMAL. We are now viewing the data in V2000 in both BCD format and Decimal format. This is a very useful tool when you are having problems with format conversions.

Element	Status
V2000	21
V2000	33

You can even view V locations as binary data. Select the next empty box under V2000 and enter V2000 again. Now change the data type for this entry to BINARY and we can see the individual bits changing state. You may need to widen the status column to see them all.

Element	Status
V2000	29
V2000	41
V2000	0000000000101001

Here's another trick you'll find useful. Select another empty element and enter V7766.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

Element	Status
V2000	31
V2000	49
V2000	00000000000110001
V7766	31

Now go back and select the V7766 element and press and hold the CTRL key and then hit the ENTER key a few times. You should see sequential registers being added. This works for any data type, V locations, C bits, timer values, anything. It also brings the same format as the element you started with along for the other registers. Pretty neat, eh?

Element	Status
V2000	40
V2000	64
V2000	0000000001000000
V7766	40
V7767	49
V7770	21
V7771	0
V7772	24

Of course you can also display bit data in data view. Go ahead and delete some of those V76xx locations. Now add C0 to an element. Now, notice that button with the C1=1 and the hourglass symbol on it up in the left corner of dataview? Press it. You should see a new column appear labeled EDITS. This column lets you change the value of bits or registers from data view, assuming the program isn't currently controlling them of course.



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

Element	Status	Edits
V2000	16	
V2000	22	
V2000	00000000000001	
V7766	16	
C0	OFF	ON OFF

Go ahead and double-click the ON button in the edits column next to C0, you'll see a window pop up that is asking if you want to write current edits, this will happen every time you try to change something if you don't click the little 'don't ask again for this view' box, select the box and hit YES. You should see the bit change from OFF to ON, unless something in the code is forcing it back off again. Intelligent modules or operator panels may also cause bits to not change, like H2-ERM and H2-CTRIO cards.

You can also send values to V locations this way. Go ahead and add another register we aren't using for anything, say V2002, to dataview. Now click in the edits field to the right of V2002 and enter a new number and press enter. Notice that two new buttons light up at the top of the dataview screen.

Element	Status	Edits
V2000	17	0
V2000	23	
V2000	00000000000001	
V7766	17	
C0	ON	ON OFF
V2002	0	50



**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

The single down arrow button allows you to write just the piece of data you currently have selected to the PLC. The multiple arrow down button writes all currently made edits to the PLC. Press the single arrow button and you should see the value in the PLC change, again, unless something in the PLC code is also writing to that location.

Element	Status	Edits
V2000	59	0
V2000	89	
V2000	000000000101	
V7766	59	
C0	ON	<input type="button" value="ON"/> <input type="button" value="OFF"/>
V2002	50	50

That's about it for this application note. Here are a few common problems and some simple programming tips to deal with them:

-Math not acting properly? Remember, when the PLC tries to perform math in a certain format and encounters data in a different format, the OUT after the math will output the same data you originally loaded with the LD. This is a clear indication that one of the numbers is not in the correct format.

-Have a large math rung with multiple math instructions that isn't solving correctly? Go in and add OUT box instructions after each math block to some currently unused V locations and pull those up in dataview to see where the math is failing.

-Having a problem writing external data to the PLC from a panel or other external device? First make sure the program mode switch is in TERM, it has to be. If that still doesn't solve it, add a new rung as the first rung and put just an END on it and download it and try again. If it works now that means something in the code is





**THIS INFORMATION PROVIDED BY AUTOMATIONDIRECT.COM TECHNICAL SUPPORT IS PROVIDED "AS IS" WITHOUT A GUARANTEE OF ANY KIND.**

These documents are provided by our technical support department to assist others. We do not guarantee that the data is suitable for your particular application, nor do we assume any responsibility for them in your application.

interfering, try moving the END rung down in the code until you find the offending logic.

-Analog values not working? Make sure you have used the correct addresses for the slot your card is in. The CPU slot is NOT slot 0...slot 0 is the slot next to the CPU. Also, be aware that changes made to rungs operated by the SP0 (first scan) bit will not be executed if you make a run time programming change. You will need to either power cycle the PLC or make a run-program-run mode change.

#### **Technical**

**Assistance:** If you have questions regarding this Application Note, please contact us at 770-844-4200 for further assistance.