DAQFactory – UE9 Tutorial
Getting Started with DAQFactory-Express and your LabJack UE9
11/2/06

Congratulations on the purchase of your new LabJack UE9. Included on your installation CD is a fully licensed copy of DAQFactory-Express and a 25-day trial of DAQFactory-Pro which will help you make the most of your UE9 without the need for programming. With DAQFactory-Express you can take data and control outputs, log data to files easily read by other programs like Excel, create your own screens for displaying your data using any combination of 10 screen components including buttons, graphs, images and more.   For more advanced applications, consider one of the other versions of DAQFactory which depending on the version include 42 screen components, email, networking, PID, alarming, a 3800 image library, unlimited channels, unlimited pages, unlimited scripting and much more.   All your DAQFactory-Express applications will work in the other versions of DAQFactory.

This document will start you on your way using your new UE9 with DAQFactory-Express and will also work in all the other versions of DAQFactory. Feel free to use DAQFactory-Express to its full capabilities. If you need a more powerful version such as demonstrated by the trial version, please visit www.azeotech.com or your LabJack reseller to purchase a license.

From your Start menu, select Programs => DAQFactory Express => DAQFactory Express. This document will show you how to start from scratch and quickly start taking data and controlling outputs. This document is essentially the same as the guided tour in the DAQFactory help adapted for the UE9.
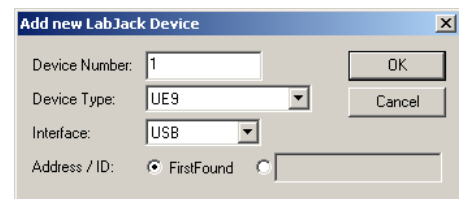
For updates to DAQFactory Express, sample documents and other DAQFactory Express specific information, please visit www.daqexpress.com.

Creating Channels for input and output

In this part, we will add a channel to the document and start acquiring data.

1. From the DAQFactory menu, select Quick – Device Configuration, or click on the hammer icon in the toolbar:

2. Select "LabJack" from the list of devices in the Device Configuration window.

3. The LabJack Device Configuration window will appear. Click on the Add button to add a new LabJack device. A new window will appear allowing you to specify which LabJack you want to use. For this sample, we will assume you only have one UE9, so select UE9 as the device type and click the button next to first found:
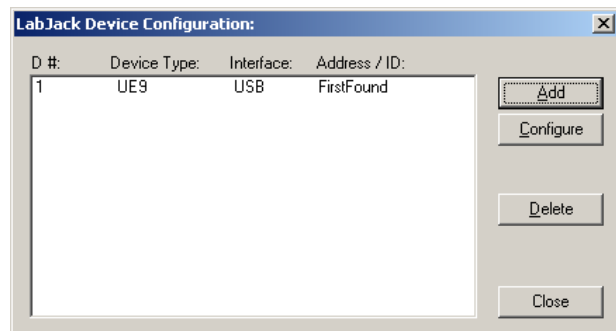
The device number is an arbitrary, non-zero number used to identify this LabJack within DAQFactory. This becomes important when you have multiple LabJacks. Currently, FirstFound can only be used with the USB connection. If you are connected to the UE9 over Ethernet, leave the button next to the address box checked and enter the IP address in that box.

4. Click OK to return back to the LabJack Device Configuration window and you will see your device listed:

Click on your device to highlight it, then click on the Configure button. This will open up the UE9 configuration window.

From here you can specify all the settings of the UE9. For now, just enable DAC0 by checking the box next to "Enable 0:" as shown. There are a number of other settings here you can play with as you get more familiar with the UE9 and DAQFactory. When done, click OK and then Close to close down all the windows and return to the main DAQFactory window.
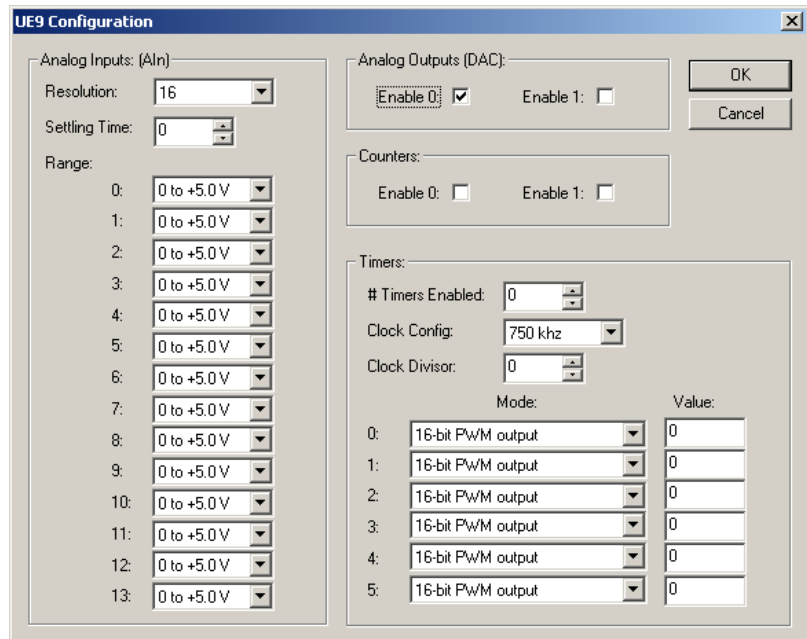
5. In the workspace (the tree area along the left of the screen), click on CHANNELS:.

The workspace provides an organized presentation of the different configuration screens for setting up your DAQFactory document. Clicking on almost every item in the tree will bring up a new view related to the item selected. Clicking on CHANNELS: displays the channel table where you can manipulate all the channels in your document.
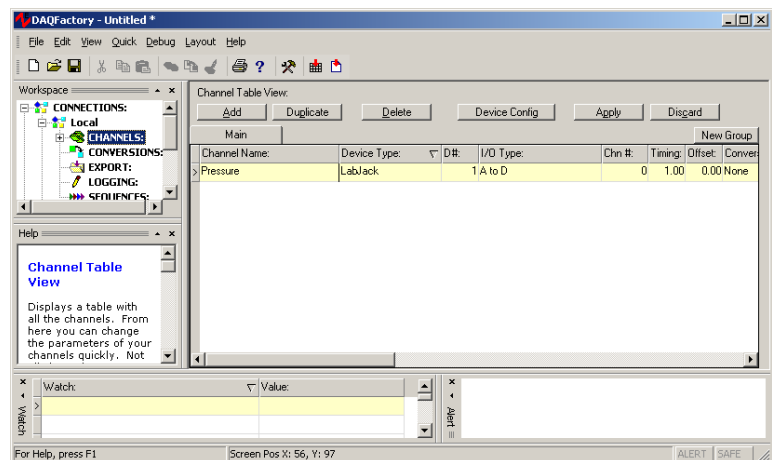
6. In the Channel Table View that appears, click on the Add button at the top.

This creates a new row where we will enter the information for our first channel.

7. In the new row of the table, enter Pressure in the Channel Name column.

All channels must have a name. You can assign any name you want to your channels as long as they are unique, start with a letter and only contain letters, numbers or the underscore "_".

8. In the second column, Device Type, select LabJack from the drop down list.

The device type determines which type of device you wish to communicate with. In Express, only LabJack and LabJack_U12 are available. In the other versions of DAQFactory you can communicate with a wide variety of devices from other manufacturers. The LabJack U12 device type applies to the U12 only. The LabJack device applies to all other LabJack devices (including the U3 and UE9)

9. In the third column, D #, set the cell to 1.

For the LabJack driver, the device # is that unique number you assigned the device in step 3 above. We left it at the default of 1.

10. In the fourth column, I/O Type, select A to D.

   The I/O type determines which type of data you desire. The UE9 has A to D (analog input), D to A, Digital In and Out, Counter and Timer. These are all the fundamental data types of this device. I/O Types are not fixed types and different devices will have different choices available.

11. In the fifth column, Chn #, enter 0.

   The channel number determines which I/O point this channel refers to with your device. For the UE9, this is the AIN pin # and the numbers correspond directly to the AIN number, so AIN0 is channel 0, AIN1 is channel 1, etc. For digital channels the mapping is:

0-7	FIO0-FIO7
8-15	EIO0-EIO7
16-19	CIO0-CIO3
20-22	MIO0-MIO2

12. Leave the rest of the columns in their default setting.

   One important item to mention is the sixth column, Timing. For input channels this determines how often the input is read from the device. The interval provided here is in seconds. For now, we are going to leave it set at 1, or one new reading per second. In general you will never want to make this number smaller than 0.01 as Windows needs a little time to redraw the screen and move the mouse. To acquire data at faster speeds requires hardware paced acquisition. To see how to do hardware paced acquisition in DAQFactory, check out the streaming sample in the DAQFactory UE9 samples directory. We recommend completing this tutorial first to ensure you understand all the basics.

13. Click on the Apply button at the top of the screen.

   The changes you make are not implemented by DAQFactory until you click Apply. If you do not want to keep your changes, you can click Discard. Once you click Apply, DAQFactory will start taking data from the test device, D#0, I/O Type A to D, Channel #0, and place that data in the "Pressure" channel.

14. Click on the + sign next to CHANNELS in the workspace.

   The + sign appeared when you hit Apply. Clicking on it displays the channels you've created as separate items in the workspace.

15. Click on Pressure listed under CHANNELS in the workspace.

   This will display the channel view for the channel Pressure. The channel view displays the configuration information for a single channel. In general, it is quicker to create your channels in the channel table, but the channel view provides space for some extra details.

16. In the Channel View that appears, click on the Table tab.

   The channel view also provides a quick table to view your incoming data. This can be used to confirm that data is being acquired and that you are reading the correct data. You should see numbers appearing in the table that correspond to the voltage on AIN0. You have just confirmed that you are acquiring data on your pressure channel.

Displaying the Data

The Channel View certainly provides a quick look at your data, but is not the most convenient or flexible way to display your data, especially once you have more than one channel. In this section we'll add a component to a page that will display data from the Pressure channel.

1. Click on Page_0 in the workspace under PAGES.

   This will display a blank white page. When DAQFactory Express starts, 2 blank pages are generated automatically with default names. With Express you are limited to 2 pages, but with all the other versions of DAQFactory you can create as many pages as you want.

2. Right-click somewhere in the middle of the blank area and select Displays-Variable Value.

    This will place a new page component on the screen. There are many different components available that allow you to create custom screens. This particular component simply displays a changing value in text form.

3. Right click on the new component and select Properties...

    All of the screen components have different properties windows for configuring the look of the component and telling the component what data to display.

4. Next to Caption:, enter Pressure.

    The caption is what is displayed in front of your value. A colon is automatically added when displayed.

5. Next to Expression:, enter Pressure[0].

    Expressions are actually formulas that you can enter to display calculated data. In this case, the formula simply requests the most recent data point of the channel pressure. The [0] indicates the most recent data point. If, for example, we wanted to convert the pressure voltage into actually pressure units, we could put something like Pressure[0] / 5 * 500. You can also use Conversions, as discussed in the help file, to convert the input into pressure units right at the channel and avoid voltages all together.
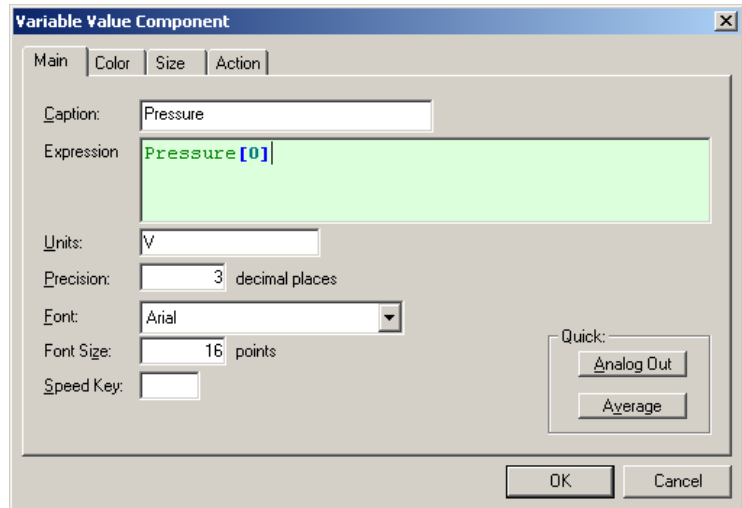
6. Click on OK.

    Now the screen should display Pressure: 0.324 V with a changing number. This is the current value in the pressure channel we created earlier. Feel free to open the properties window again and play with the units, font and other settings.

Feel free to play with other similar components if you'd like. When done, you can delete them by selecting the component (Control - Click) and then right clicking and selecting Delete Component. To move a component, hold the Control key and click and drag the component. All component manipulation uses the Control key, as a regular click will activate an action, such as a button press.

Graphing / Trending the data

Displaying scalar values in text form is certainly useful, but nothing is better than a graph to give you a handle on what is happening with your system. DAQFactory offers many advanced graphing capabilities to help you better understand your system. In this section we will make a simple Y vs time or trend graph.

1. If you are still displaying the page with the value on it, hit the 1 key to switch to Page_1. If you are in a different view, click on Page_1 in the workspace.

    The workspace is only one way to switch among pages. Another is using speed keys which can be assigned to a page.

2. Right click somewhere on the blank page and select Graphs - 2D Graph. Move and resize the graph so it takes up most of the screen by holding down the control key and dragging the component. Drag the black square boxes around the selected graph to resize. Next, open the properties window for the graph.

2D Graph Component

Traces | Axes | General | Colors + Fonts | Axis + Bound Annotations | Line Annotations

Traces:
Pressure

Y Expression: Pressure
X Expression: Time

Axis: Left Axis 1
Legend:
Trace Color:
Line Type: Medium Solid
Point Type: Circle
Name:

Wind Barbs:
Direction:
Speed:
Length:

Error Bars:
Y Positive:
Y Negative:
Y End Width: 5 pixels
X Positive:
X Negative:
X End Width: 5 pixels
XY Box?
End Thickness: 1 pixels
Line Thickness: 1 pixels

Add   Delete
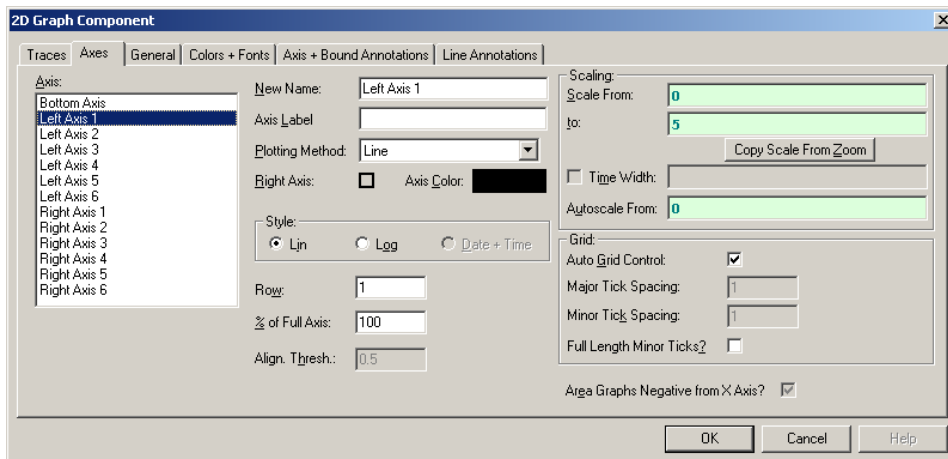
OK   Cancel   Help

3. Next to Y Expression: type Pressure.

The Y expression is an expression just like the others we have seen so far. The difference is that a graph expects a list (or array) of values to plot, where in the previous components we have only wanted the most recent value. By simply naming the channel in the Y expression and not putting a [0] after it, we are telling the graph we want the entire history of values stored in memory for the channel. The history length, which determines how many values are kept in memory and therefore how far back in time we can plot is one of the parameters of each channel that we left in its default setting of 3600.

4. Leave all the rest in their default settings and click OK.

You should now see your data displayed in the graph. The data will move from right to left as more data is acquired for the pressure channel. To better display the data, we need to change the Y axis scaling.

2D Graph Component

Traces | Axes | General | Colors + Fonts | Axis + Bound Annotations | Line Annotations

Axis:
Bottom Axis
Left Axis 1
Left Axis 2
Left Axis 3
Left Axis 4
Left Axis 5
Left Axis 6
Right Axis 1
Right Axis 2
Right Axis 3
Right Axis 4
Right Axis 5
Right Axis 6

New Name: Left Axis 1
Axis Label:
Plotting Method: Line
Right Axis: Axis Color:

Style:
Lin   Log   Date + Time

Row: 1
% of Full Axis: 100
Align. Thresh.: 0.5

Scaling:
Scale From: 0
to: 5
Copy Scale From Zoom
Time Width:
Autoscale From: 0

Grid:
Auto Grid Control:
Major Tick Spacing: 1
Minor Tick Spacing: 1
Full Length Minor Ticks?

Area Graphs Negative from X Axis?

OK   Cancel   Help

5. Double click on the left axis of the graph.

Unlike other components, you can double click on different areas of the graph to open the properties window for the graph to different pages. Double clicking on the left axis brings up the axis page with the left axis selected.

6. Next to Scale From enter 0, next to Scale To enter 5, then click OK.

This will scale the graph from 0 to 5 which will should show the full range of the input. You may want to adjust these numbers to your particular signal.

Now we'll try another method to scale the graph.

7. Open the graph properties box again and go to the Traces page. Change the Y Expression to Pressure * 10 and hit OK.

   Like the other screen components, the expressions in graphs can be calculations as well as simple channels. In this case, we are multiplying each of the values of Pressure's history by 10 and plotting them.

8. Deselect the graph by clicking on the page outside of the graph. The shaded rectangle should disappear. Next, right click on the graph and select AutoScale - Y Axis

   The graph component has two different right click popup menus. When selected, it displays the same popup menu as the rest of the components. When it is unselected, it displays a completely different popup for manipulating the special features of the graph.

   After autoscaling, you should see the graph properly scaled to display your signal zoomed in. Notice the rectangle around the graph. The left and right sides are green, while the top and bottom are purple. The purple lines indicate that the Y axis of the graph is "frozen". A frozen axis ignores the scaling parameters set in the properties window (like we did in step 6 above) and uses the scaling from an autoscale, pan, or zoom.

9. To "thaw" the frozen axis, right click on the graph and select Freeze/Thaw - Thaw Y Axis

   Once thawed, the graph will revert to the 0 to 5 scaling indicated in the properties box and the box surrounding the graph will be drawn entirely in green.

10. Double click on the bottom axis to open the properties to the axis page with the bottom axis selected. Next to Time Width:, enter 120 and hit OK.

    If the double click method does not work, you can always open the properties window for the graph using normal methods, select the Axes page and click on Bottom Axis.

In a vs. time graph, the bottom axis does not have a fixed scale. It changes as new data comes in so that the new data always appears at the very right of the graph. The time width parameter determines how far back in time from the most recent data point is displayed. By changing it from the default 60 to 120 we have told the graph to plot the last 2 minutes of data.

Once again, if you zoom, autoscale, or pan the graph along the x axis, it will freeze the x axis and the graph will no longer update with newly acquired values. You must then thaw the axis to have the graph display a marching trace as normal.

Controlling an output

In addition to taking data, most systems also control outputs. Here we will create an output channel and control it manually from a page component.

1. Click on CHANNELS: in the workspace to go to the channel table.

2. Click on Add to create a new row in the table.

3. In the new row, name the channel Out, Device Type LabJack of course, D# of 1, I/O type of D to A, and a Channel # of 0. Leave the rest in their defaults. This corresponds to DAC0.

4. Click on Apply to save your changes.

   The output channel has been created, but it hasn't been set to anything.

5. Click on Page_0 to go to our first page.

6. Right-click and select Displays-Variable Value to create another variable value component.

7. Right click on the new component and select Properties... to open the properties window.

8. For the expression, enter Out[0].

Feel free to set the caption as well. Like before, this will simply display the most recent value of the out channel.

9. Click on Action tab.

This tab exists on several different components including the static ones and works the same way with all of them.

10. From the Action drop down, select Set To.

There are many different options for the Action described in the DAQFactory help. The Set To action will prompt the user for a new value when the component is clicked.

11. Next go to Action Channel: type Out.

12. Leave the Range blank and click OK.

The range allows you to constrain the inputs to particular values. By leaving these properties blank, we are indicating that we do not want a range limitation. The page will now display your caption and 0 with a big red X through it. This indicates that out does not have a valid value yet. This is because we haven't set it to anything.
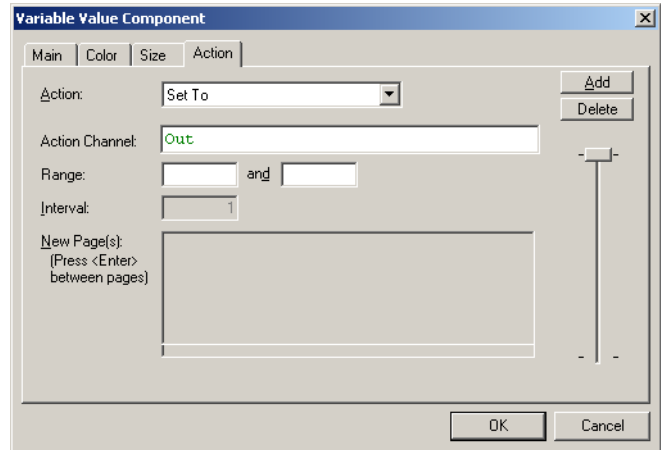
13. Click on the component. A new window will appear requesting a new value. Enter a voltage value within the DAC range, such as 2, and hit OK.

Out will now be set to the value you entered. The component will now display your new value without the big red X. If you tie a wire between DAC0 and AIN0 you will see the Pressure reading track with the DAC output. Note, if you'd like to be able to enter an output value in units other then volts, you need to create a Conversion and apply it to your output channel. This is described in the online help.

Logging Data

Next we will learn how to store this data to disk so that it can be opened with another program for analysis. In this example we will create a simple comma delimited file with our pressure data.

1. Click on LOGGING: in the workspace.

This will display the logging set summary view. Here you can see all your logging sets displayed and their running status. A logging set is a group of data values being logged in a particular way to a particular file. You can have as many logging sets as you need, and they can run concurrently.

2. Click on the Add button in the logging set and when prompted for the new name, type in Log and click OK.
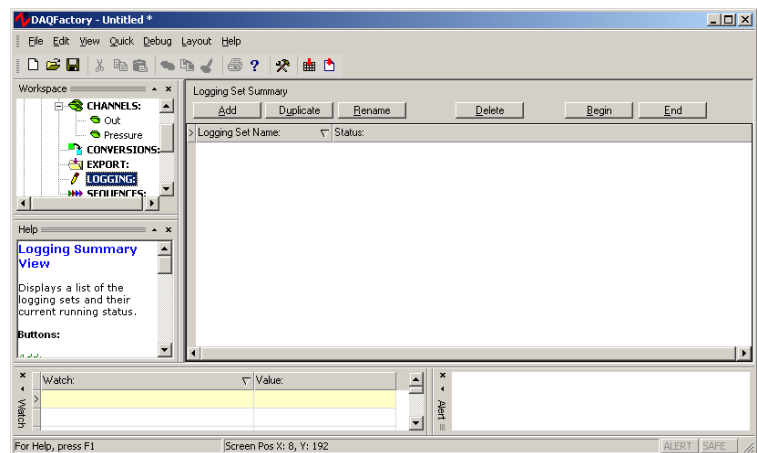
Like channels and all other DAQFactory names, the logging set name must start with a letter and only contain letters, numbers or the underscore. Once you click OK, the logging set view will be displayed for the new logging set. You can also get to this view by clicking on the logging set name listed under LOGGING: in the workspace. You may have to click on the + sign next to LOGGING to see your logging sets listed in the workspace.

3. Next to Logging Method, select ASCII Delimited

    ASCII Delimited is probably the most common method for data logging as it can be read by most other programs such as Excel. Unfortunately, it is not as space efficient or fast as the binary methods. But unless you have strict space constraints or you are logging more than about 50,000 points per second (depending also on your computer / harddrive speed), we suggest ASCII.

4. Next to File Name enter c:\mylogfile.csv

    It is usually best to fully specify the path to your file, otherwise the data will be stored in your DAQFactory directory. The .csv is a windows standard designation for comma delimited values.

5. In the Channels Available table, click on the row with Pressure, then click on the >> button to move it to the Channels to Log table.

    Each logging set can log any combination of channels. In this case, we will just log the input channel.

6. Click on Apply to save the changes.
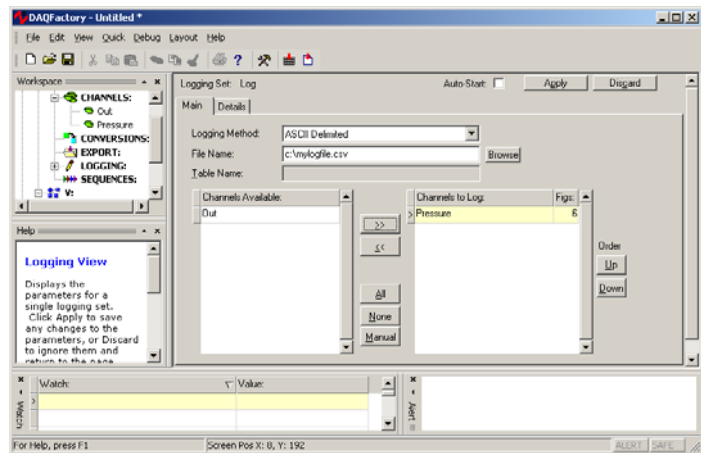
7. To start logging, click on the + next to LOGGING to display the new logging set, then right click on the logging set Log and select Begin Logging Set.

    Once started, the icon next to Log will change and the red stop sign will be removed to indicate that this logging set is running.

8. Wait at least 10 or 15 seconds to log some data and then right click on the logging set again and select End Logging Set to stop logging.

    There are other ways to start and stop logging sets, including the action page of some components such as the variable value component that we used earlier.

9. Now start Excel or Notepad and open the file c:\mylogfile.csv.

    You will see two columns, time and data. By default the time is given in Excel / Access format which is decimal days since 1900. You can also have DAQFactory log in its standard format of seconds since 1970. If you are using Excel and logging the time in Excel format, you can format the first column as date / time and the numbers displayed will be properly displayed as date and time.

Hint: if you cannot find the mylogfile.csv file, check to make sure that you selected Pressure to log and not Out. Since Out is an output channel, it only gets new values to log when you actually set it to something.

You have now acquired, displayed and logged your data.

For basic applications, it is simply a matter of repeating most of these steps for additional channels. But even for the most basic applications DAQFactory offers many adjustments for fine tuning your application to work the way you want. These, along with many more walk-throughs and examples are provided in the appropriate sections of the DAQFactory help. At this point we suggest playing with DAQFactory a bit, jumping to the appropriate sections of the manual for more detail on the areas of DAQFactory you may be interested in utilizing, and checking out the samples installed with DAQFactory.