DAQFactory Express – U12 Tutorial

A step-by-step guide for using your new LabJack with DAQFactory-Express

Congratulations on the purchase of your new LabJack U12. Included on your installation CD is a fully licensed copy of DAQFactory-Express and a 25-day trial version of DAQFactory-Pro data acquisition software which will help you make the most of your LabJack without the need for any programming. 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 40 different screen components including knobs, switches, graphs, images, gauges and more. And if you prefer the flexibility of programming, DAQFactory includes a scripting language for performing just about any data acquisition task, while still taking advantage of all the rest of DAQFactory's features.

This detailed guide will walk you through taking data from your LabJack, displaying the data on the screen in various ways, graphing it, controlling outputs, and logging your data to a file with DAQFactory.

Starting DAQFactory Express:

Click on the **Start** button, select **Programs**, **DAQFactory Express** then **DAQFactory Express**.

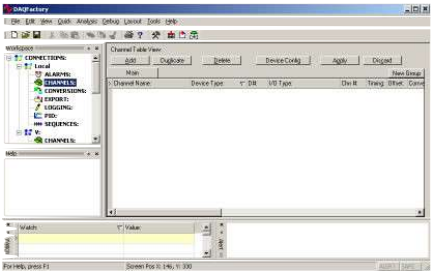
A splash screen will display and then you will be left with a blank DAQFactory document. For this tour, we will mostly be using the workspace, which is the area along the left of the screen with an Explorer like tree, and pages, which is currently the blank area that occupies most of the screen.

Creating Channels:

First we will add a channel in DAQFactory and start acquiring data.

1. 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.*



2. 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.

3. 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 "_". Like just about everything else in DAQFactory, Channel names are case sensitive.

4. In the second column, **Device Type**, select **LabJack_U12** from the drop down list.

   The device type determines which type of device you wish to communicate with. This can be a specific manufacturer, a specific device, or a generic communications method such as OPC or serial.

5. In the third column, **D #**, leave the cell set at 0.

   The device # is only used by some devices and has different meanings depending on the device. For the LabJack U12, the device # is either the ID # burned into the LabJack, or 0 which uses the first LabJack found.

6. In the fourth column, **I/O Type**, select **A to D (SE 0-7; Diff 8-11).**

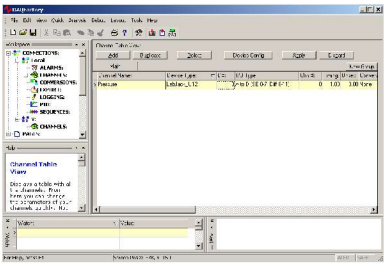
   Most devices have several different types of data coming in or going out. The I/O type determines which type of data you desire. The LabJack U12 contains A to D, D to A, Digital inputs and outputs and counter channels. The I/O Type determines which of these you wish to communicate with for this channel. I/O Types are not fixed types and different devices will have different choices available.

7. In the fifth column, **Chn #**, enter **0.**

   Most devices have several I/O points per I/O type. The channel number determines which I/O point this channel refers to. For the A to D channels of the LabJack, 0 through 7 are the single ended inputs, and 8 through 11 reference the differential inputs.

8. 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. Because the LabJack is a slower device, you should never set the Timing to a value less than 0.02, and you may not even be able to go that small. For high speed acquisition, use the streaming mode as described in the DAQFactory help.*



9. Click on the **Apply** button at the top of the screen.

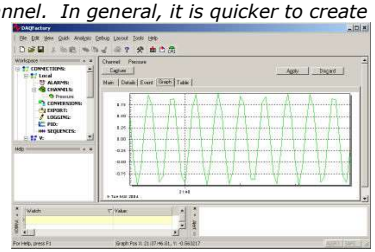
   *The changes you make are not implemented by DAQFactory until you click **Apply**. If you do not want to keep your changes, you can click **Discard**. Once you click **Apply**, DAQFactory will start taking data from your LabJack device, D#0, I/O Type A to D, Channel #0, and place that data in the "Pressure" channel. This assumes of course that you have the LabJack plugged in. If it is working, the LED on the LabJack will blink once a second.*

10. 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.*

11. 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.



12. In the **Channel View** that appears, click on the **Graph** tab.

    The channel view also provides a quick graph and 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. What you will see in the graph will depend on what you have wired into AI0 of your LabJack. Even without anything wired, you should see a horizontal line at around 1.4 to 1.7 volts. You have just confirmed that you are acquiring data on your pressure channel.

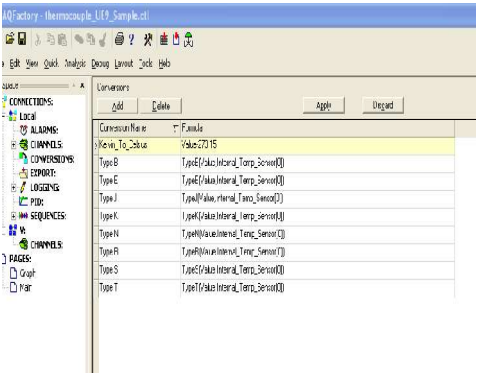
13. If you'd like to see the table, click on the **Table** tab.

You are now taking data. In the next section we'll display it on our own custom screens.

Conversions:

Conversions provide a way to quickly convert your voltage values read from the U12 into more useful units, such as Celsius, psi, etc. Another convenient use for a conversion is to quickly convert uncalibrated readings into more useful calibrated quantities through a derived equation. For example, you may have calculated a slope and offset to apply to each voltage data point to adjust the calibration. A Conversion can be used to quickly do this for each data point.

1. To create a Conversion click on **CONVERSION:** in the Workspace. This will bring up the Conversion table. Click on the **Add** button to add your first conversion equation. Give your conversion a name in the Conversion Name field and then type a formula for the conversion in the Formula field. The graphic shows an example of a conversion table with several thermocouple conversions.



2. An example of a simple conversion would be to offset each data point by 100. The formula for this would be written as: Value + 100. Where Value is used wherever you want to substitute the raw channel value in the formula.

3. After you have completed your conversion select the **Apply** button.

4. Next, go to the channel table by clicking on **CHANNELS:** in the Workspace. The eighth column from the left is labeled Conversion. Click in the appropriate row for the channel you wish to apply the conversion to and a drop down menu will appear. Select your conversion from the drop down menu and select the **Apply** button. Now each raw channel value will be automatically converted into a more useful format for your application.

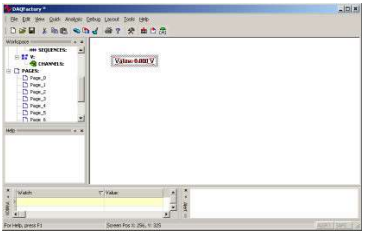
Displaying the Data:

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 some different components 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 DAQFactory-Express you are limited to these 2 pages. With other versions of DAQFactory as seen under the trial version 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...**

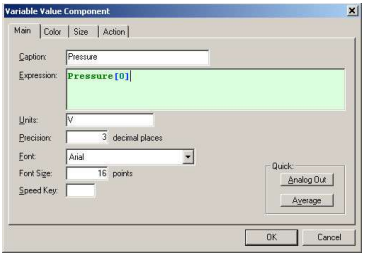
All of the screen components have different properties windows for configuring the look of the component and telling the component what data to display.

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.



6. Click on **OK**.

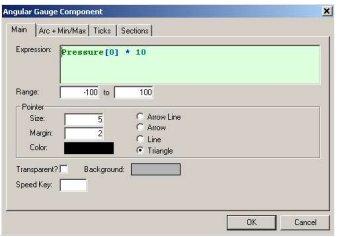
*Now the screen should display **Pressure: 1.412 V**. The value will depend on what you have wired into AI0. This is the current value in the pressure channel we created earlier. Feel free to open the properties window again and play with the units, font and other settings.*

7. While holding the **Control** key, click on your **Variable Value** component.

This will select the component, as indicated by the shaded rectangle surrounding it.

8. While holding the **Control** key, click and drag the **Variable Value** component to a new location on the screen.

This allows you to move the component around the screen.



You now know how to display your data using the Variable Value component. Next we will learn how to use the graph to display historical data.

In the mean time, 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**.

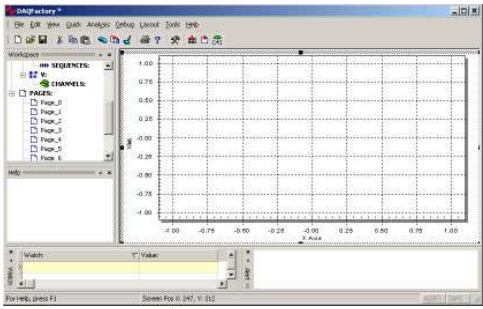
Graphing / Trending the data:

Displaying scalar values in text or gauge 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. The screen shots show a sine wave which is generated by the Test device. Your screens will look different depending on what you have wired into AI0.

1. If you are still displaying the page with the gauge 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. By default, the initial 2 pages are assigned the keys 0 and 1.

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 (see Display the Data if you forgot how to do this). Next, open the properties window for the graph.

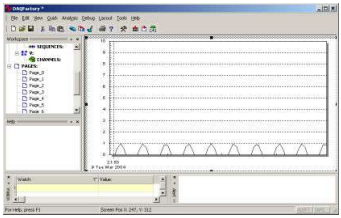


3. Next to **Y Expression:** type **Pressure**.

   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**.

   Depending on what signal you have wired into AI0, you may see a trace displayed in the graph. To display the entire trace, we need to change the Y axis scaling.



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

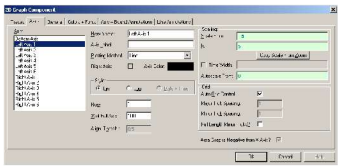
   Unlike other components, you can double click on different areas of the graph to open the properties window for the graph to different pages. Double clicking on the left axis brings up the axis page with the left axis selected.

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

   This will scale the graph from -5 to 5. Depending on what you have connected to AI0, you may want to select a different range.

7. Now we'll try another method to scale the graph. Open the graph properties box again and go to the **Traces** page. Change the Y Expression to **Pressure * 10** and hit **OK**.

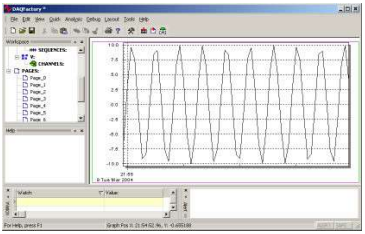
   Like the other screen components, the expressions in graphs can be calculations as well as simple channels. In this case, we are multiplying each of the values of Pressure's history by 10 and plotting them. This should take our Pressure value out of the current graph range.



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.

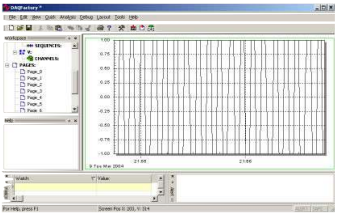
After autoscaling, you should see the graph properly scaled to view the entire range of AI0. Notice the rectangle around the graph. The left and right sides are green, while the top and bottom are purple. The purple lines indicate that the Y axis of the graph is "frozen". A frozen axis ignores the scaling 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 -5 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**.*

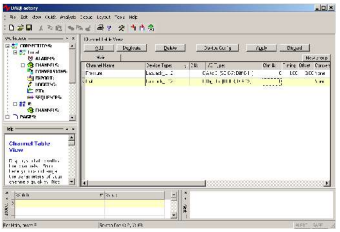
In a vs. time graph, the bottom axis does not have a fixed scale. It changes as new data comes in so that the new data always appears at the very right of the graph. The time width parameter determines how far back in time from the most recent data point is displayed. By changing it from the default 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. DAQFactory offers many advanced automated control features such as PID loop control and sequences, but for now we will just 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,**with a Device Type of **LabJack_U12**, Device # of **0**, an I/O type of **Dig Out (IO 0-3; D 4-19)** and a Channel # of **0**. Leave the rest in their defaults.

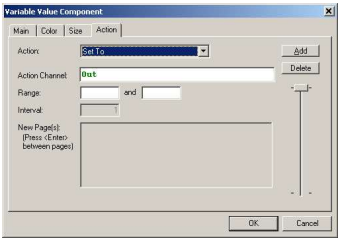


4. Click on **Apply** to save your changes.

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

5. Click on one of your pages such as **Page_1.**
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 later in the manual. 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.

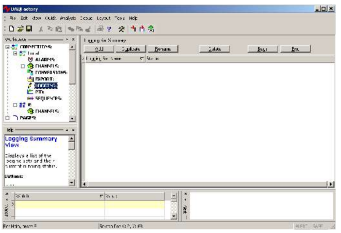
13. Click on the component. A new window will appear requesting a new value. Enter any number and hit **OK**.

    Out will now be set to the value you entered. The component will now display your new value without the big red X.

You now know how to set an output to value using several different screen components. Next we will log the incoming data to disk.

Logging Data

So far we have seen how to take data and display it on the screen. 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**.

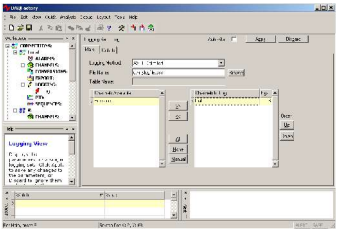
   *Like channels and all other DAQFactory names, the logging set name must start with a letter and only contain letters, numbers or the underscore. Once you click **OK**, the logging set view will be displayed for the new logging set. You can also get to this view by clicking on the logging set name listed 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 10,000 points per second (depending also on your computer / hard drive 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**. The file is still open for logging in DAQFactory, so you may have to quit DAQFactory first.

   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.

Included in the installation are four files that were created to work with your LabJack U12 and DAQFactory-Express. To view these files, open DAQFactory-Express and select File => Open from the main menu. Look under C:\DAQFactoryExpress\Samples\LabJackU12\. In particular the file StreamingAnalog.ctl will get you started with streaming data from your U12

For simpler applications, it is simply a matter of repeating most of these steps for additional channels. But even for simpler applications there are many tweaks available for fine tuning your application to work the way you want. These, along with many more examples are provided in the appropriate sections of the DAQFactory help file. At this point, we suggest playing with DAQFactory a bit, jumping to the appropriate sections of the help file for more detail on the areas of DAQFactory you may be interested in utilizing. Also in the help is more detail on using other features of the LabJack within DAQFactory.