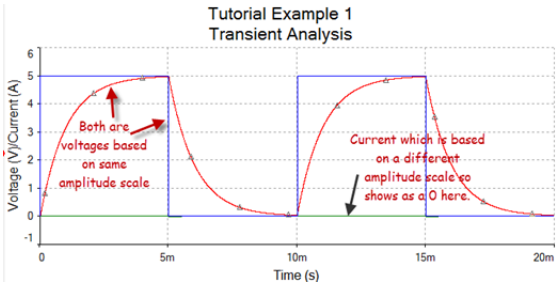
Multiple Graphs Tutorial

Introduction:

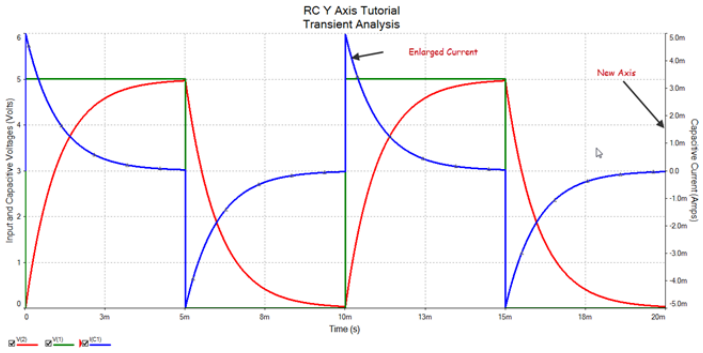
This tutorial demonstrates how to use **Multisim's Postprocessor** to create several graphs on the same page (but not in the same graph) which share the same **x-axis** scale. This is very valuable when you have more than two graphs which need to be compared and different points in time **(t)** or (X).

Note the three traces in the graph to the right. All three are based on the same time scale but one of them is shown as a 0 line when compared with the other two. This is because it is measured in **mA** while the other two are measured in **Volts**.



As shown in a previous tutorial, one way to solve this issue is to add in a second **y-axis** with a separate scale. When we do this all three waveforms will be able to be seen on the screen (and measured), but there will be a few disadvantages:

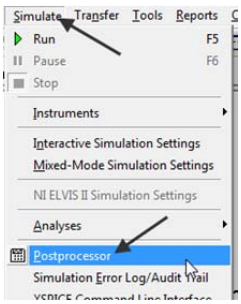
- The Grid is associated with the left axis, so it will not correspond with the new right axis.
- The graph as a whole will be very 'busy'. It will be difficult to understand what is happening fully because the data will tend to get 'lost in the forest'. In general, I don't believe in more than two waveforms on a graph.



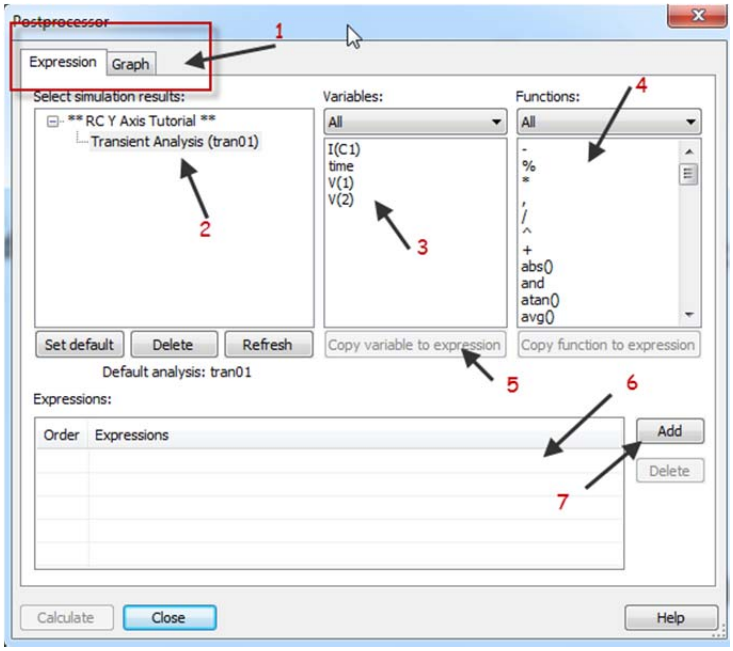
The Postprocessor

There is a much better way to do this. If you bring up the simulation menu, you will see a selection for the '**Postprocessor**'. This is a tool that works in conjunction with Multisim's GRAPHER to graph the results of the simulations, and to perform mathematical operations on the simulated data and graph those results as well. You will not that there are a large number of possible math operations, including the Integral and the Derivative. However, we will stick with just graphing the actual, unmodified simulation results.

So, let's bring up the Postprocessor and play with it.



Go to the '**Simulation**' menu and scroll down to the Postprocessor selection and open it up.



Note that there are two tabs, **Expression and Graph (1)**. We will 1st work with the **Expression Tab**.

In order to use the Postprocessor **you MUST have already performed the desired simulation.** When this has been done, the results name will be in the left window **(2)**.

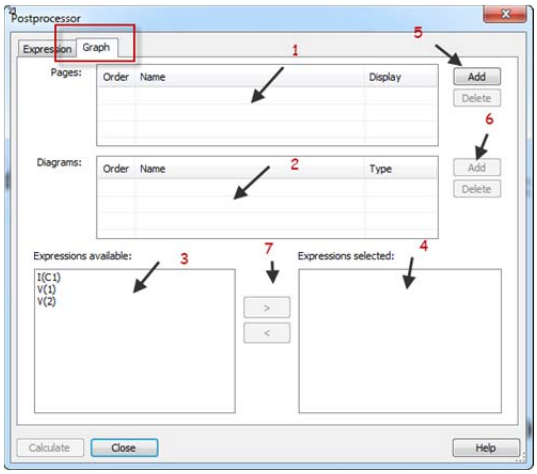
The variables that were in the **OUTPUT** of the simulation will be listed in the middle window **(3)**.

The functions that are available are shown in the right hand window **(4)**. In order to use them, you select the desired function and then copy it into the expression **(6)**. We will not be using this feature here.

- 1st, select the **Add button (7)**. This will create an expression line in the **Expressions** window **(6)**. Then select **I(C1)** in **(3)** and click on the '**Copy variable to expression**' button **(5)**. The **Expressions** window **(5)** will now look as shown below.

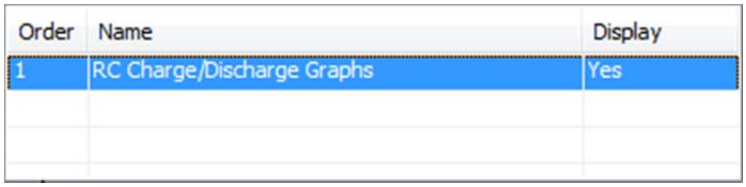
- Select Add again, and then select V(1), the input voltage, and add it to the new expression line. Repeat this for V(2), the capacitor voltage. The Expressions window now looks as shown below.

Now that the variables that are desired for the graphs have been identified, click on the Graph tab **(1).** The Graph Screen consists of the following windows:

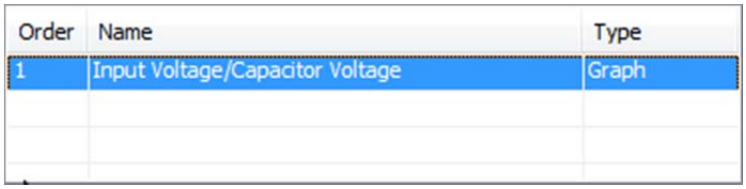


- The **Pages** window **(1).** This is the section where you create and name the page which the graph(s) will appear. Each time the **Add** button **(5)** is clicked, a new page will be created. You will name each page in the selection.
- The **Diagrams** window **(2).** This is where you create and name the **Graphs (diagrams)** which will be created on the created page. Each time the **Add (6)** button is clicked, a new graph will be created. You will name the each graph in the selection.
- The **Expressions available** window **(3).** This is where the items which you selected in the Expressions tab are shown.
- The **Expressions selected** window **(4).** This is where the traces required for a selected graph are placed.

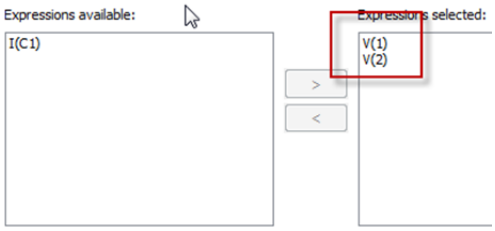
1. Create a page called '**RC Charge/Discharge Graphs**'. Click on **Add (5)** and then type the **new name** in place of the '**Post_processor_page_1**' which has now appeared. The **Pages (1)** window resembles the figure below. You are now finished with the **Pages** window **(1).**



2. Create your 1st graph. Click on the **Add** button **(6)** and a diagram line will appear. For this graph we will include both Voltage waveforms so rename the diagram "**Input Voltage/Capacitor Voltage**".

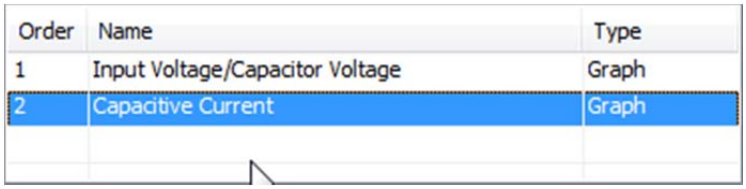


3. Select **V(1)** in the **Expressions available** window **(3)** and click on the **Add** button **(7)**. **V(1)** will move to the **Expressions selected** window **(7)**. Repeat the process for **V(2)**.



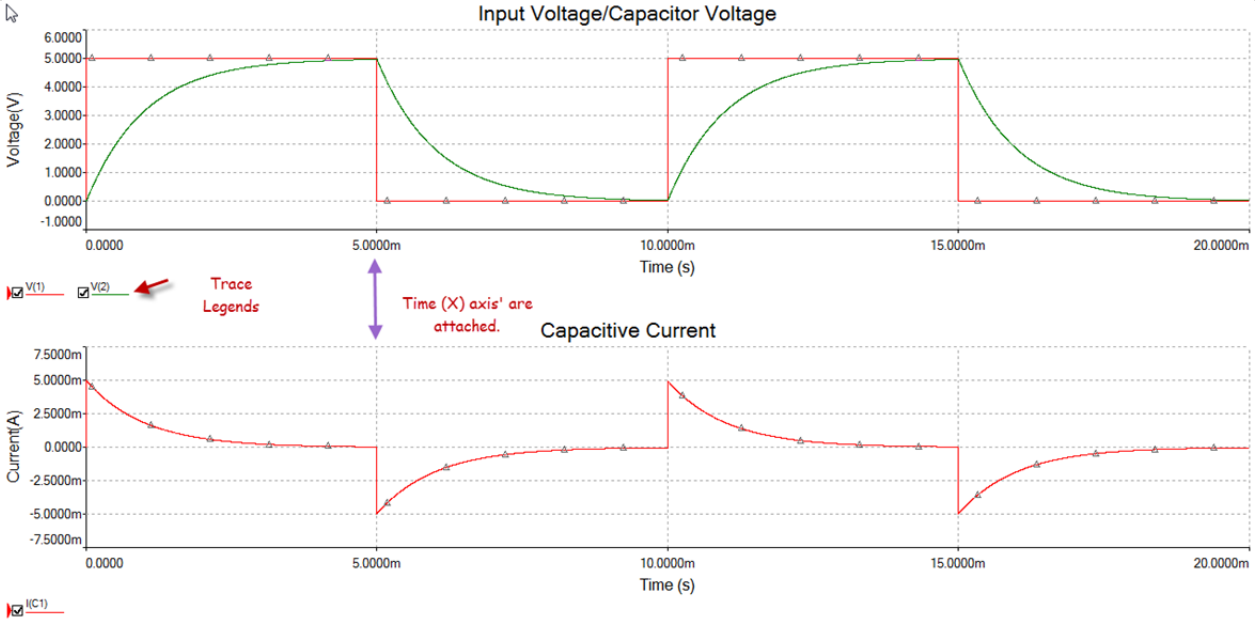
Note that the two selected variables have MOVED from the left window to the right window.

4. Go back to the **Diagrams** window and click on the **Add** button **(6)** to create a new diagram. Rename this diagram **Capacitive Current**.



MAKE SURE THAT THIS DIAGRAM LINE REMAINS HIGHLIGHTED so that the expressions can be selected for it instead of the 1st diagram.

5. Repeat step 3 for **I(C1)**. Once you are finished, click on **Calculate** in the bottom left corner of the window. The following page will appear.



Note that there is a legend for each graph. Also note that the two curves are aligned such that the Time axis for both graphs, are coordinated with each other. If for some reason they are not; then adjust the alignment by modifying the Precision values for one or both Y-Axis.

You can now format each graph individually as you have learned previously. You can also open up measurement windows in each graph as shown here.