# ADS Quick Start Guide

This guide assumes you already have ADS installed, licensed, and the analog libraries connected to your workspace. If you haven't done these, consult the appropriate guide.

Step 1)

Assuming you're in the same state that Mr. Chally's document left you off at, you should see a screen similar to the one below. If not, click the **Schematic** button to create a new circuit schematic.
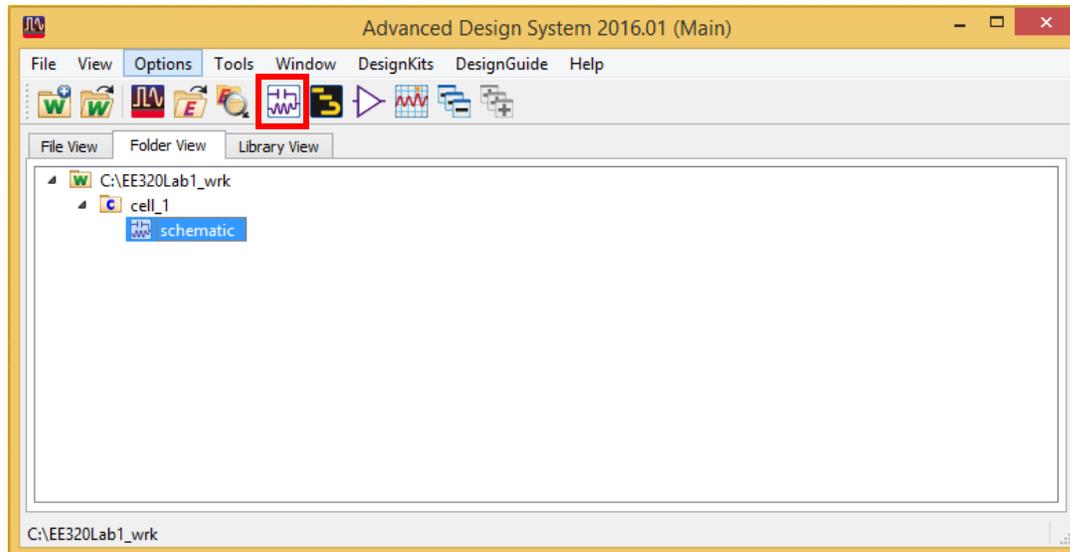

Step 2)

Let's say you want to simulate a simple circuit – maybe some low pass filter or something similar. On the schematic page, select the **Libraries** button to insert the components that you want. (Note: Make sure you select the correct part number! For EE320 labs, this is of **<u>critical</u>** importance in ensuring correct results)
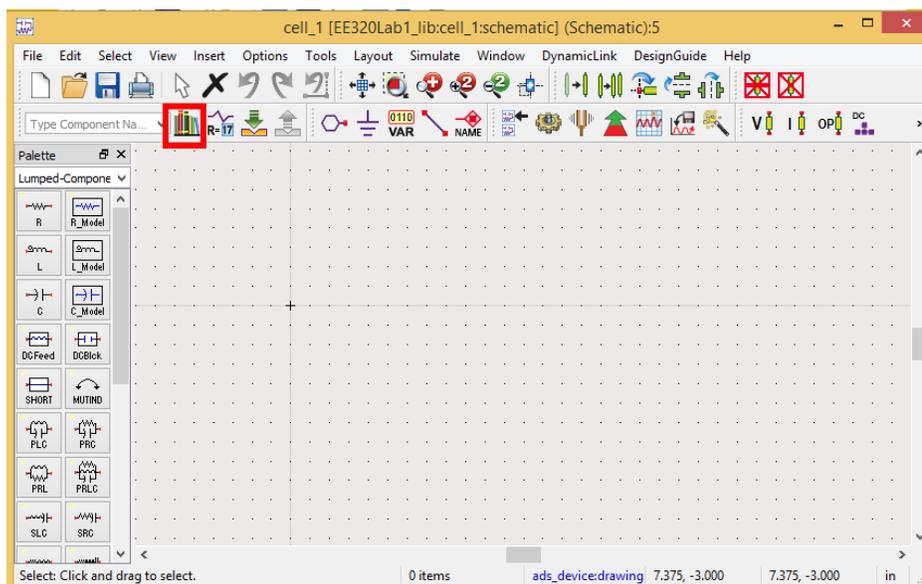
Step 3)

Shown below is my simple circuit. You can see it's composed of an AC source, a resistor, and a capacitor. Highly complex, I know.
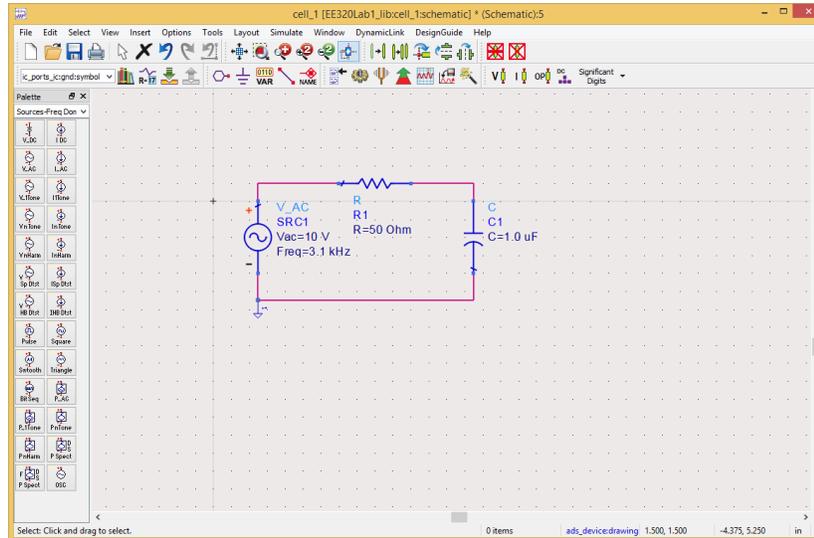

Step 4)

As is probably apparent, this is indeed a simple low-pass RC filter. Math tells us that it should have a cutoff frequency of ~3.1 kHz, so let's check! However, it's a bit more complicated than PSpice in that we need a **Simulation Controller.** Clicking Simulate without adding a controller will lead to an error message similar to the one shown.
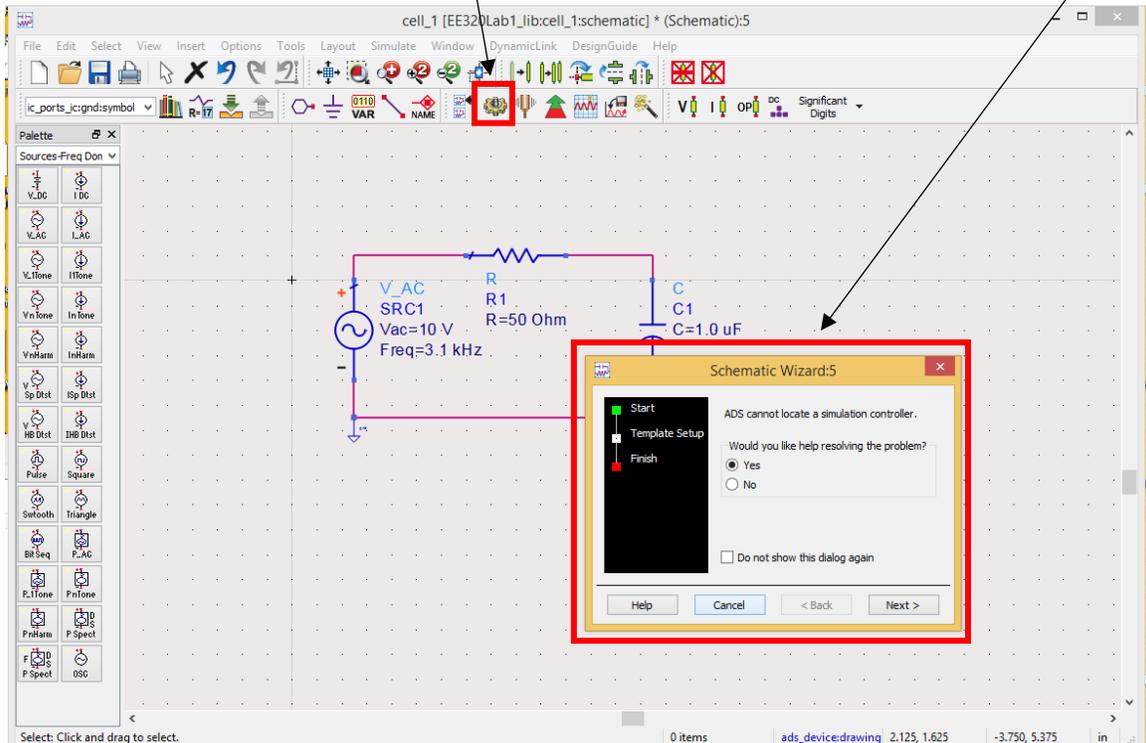
Step 5)

We want a simple transient analysis, so let's go add that controller. This can be found under the palette to the left of the circuit schematic – under **Simulation-Transient**. Click the box labeled **Trans**, and then click onto your circuit.
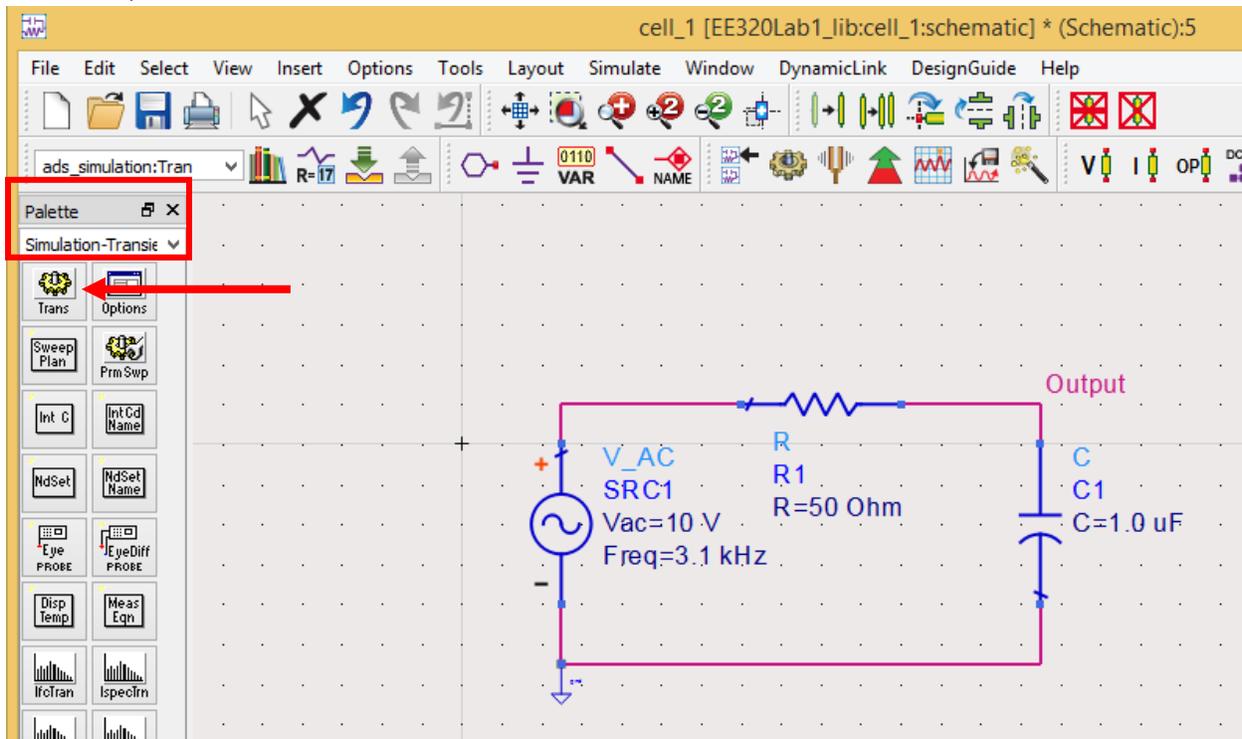
*In the case of wanting to characterize a filter, one would likely want an AC sweep across frequencies. In this case, we're using a transient analysis so we can see the output waveform – something we'll be needing often in EE320.
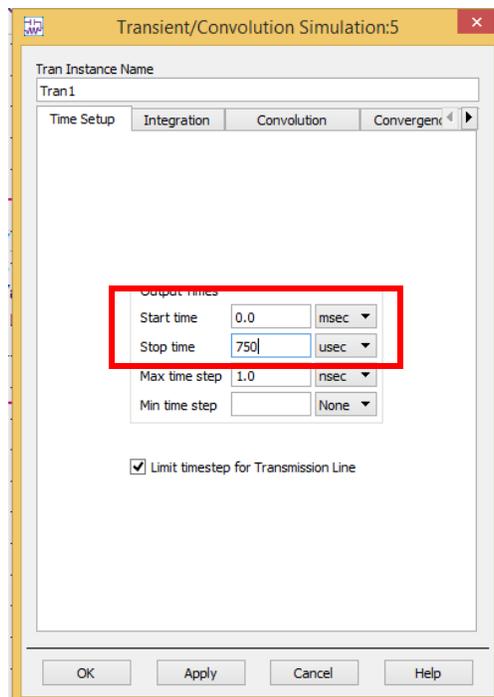
Step 6)

The parameters of the simulation can be changed by double-clicking on the **Settings** box of the controller. In this case, we want to edit the time duration of the simulation.
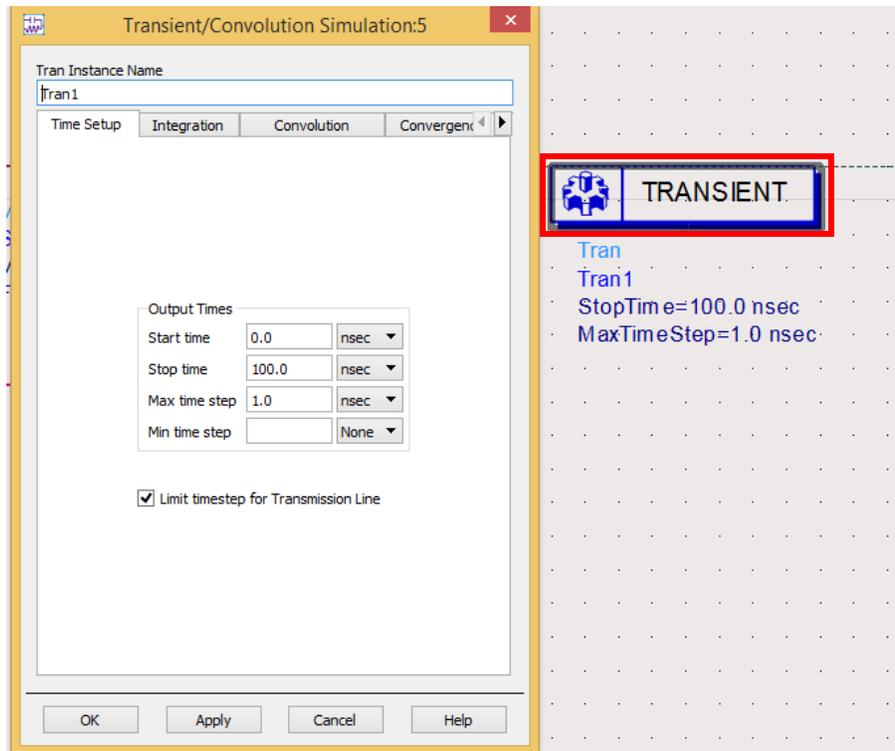

Step 7)

We want to test our filter at 3.1 kHz, it's supposed cutoff frequency. Setting a stop time of 750 us should give us two nice waveform periods to observe.

Step 8)
Now that we have everything set up, we need a place to take our measurements! This is most easily done by clicking the **Name** button on the toolbar, giving the net a descriptive name, and clicking on the circuit where we want to measure the waveform.
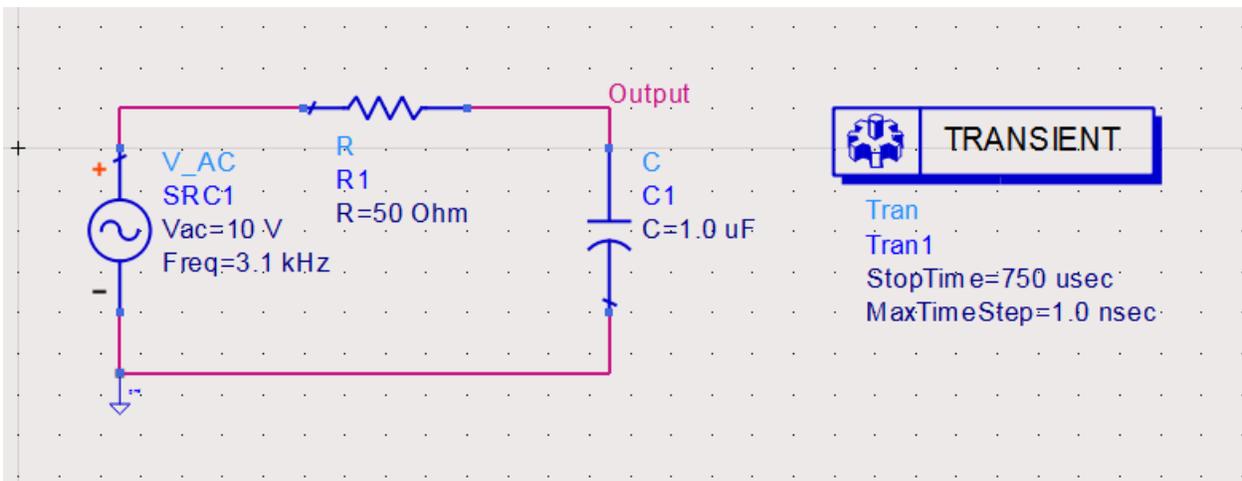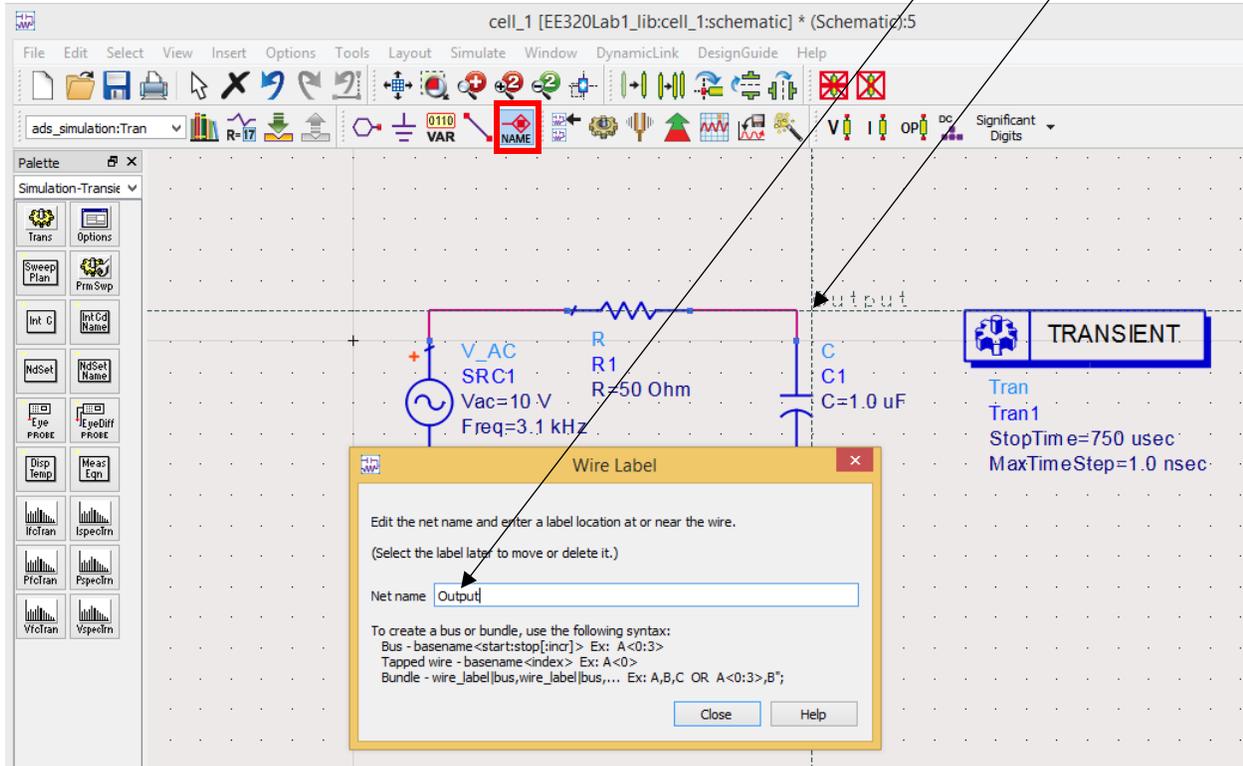
Step 9)

We're now ready to simulate this circuit and see what our output waveform looks like. Click again on **Simulate**, and now we should see the simulation window appear without any errors.
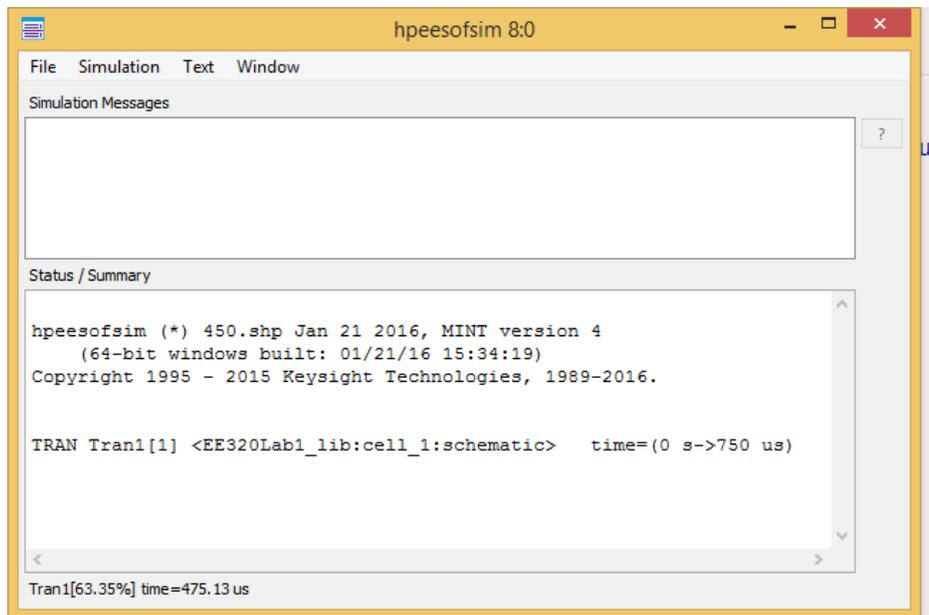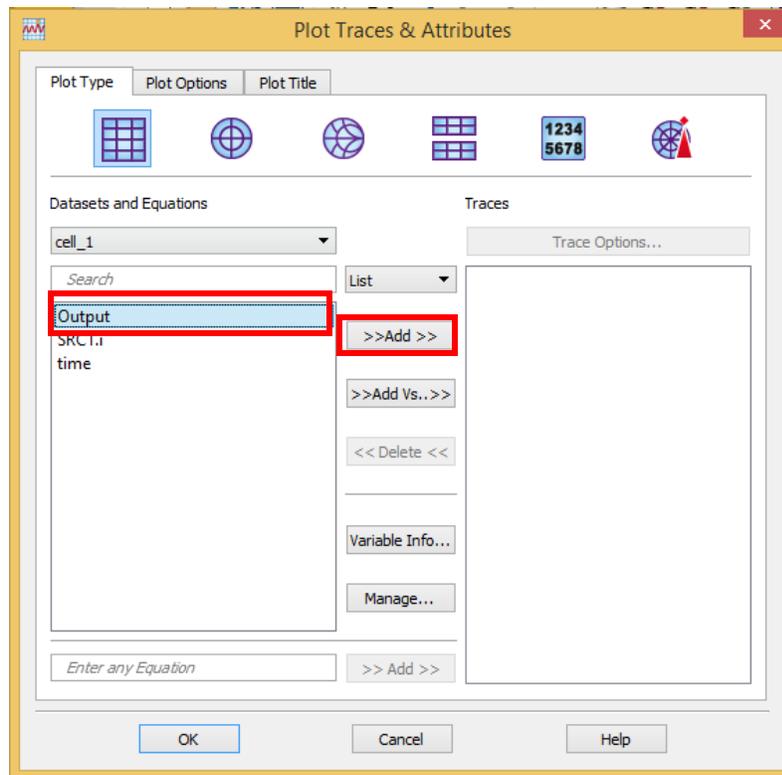

Step 10)

Once the simulation finishes, a blank window should display, looking suspiciously like it could contain some graphs. Let's add some graphs. Click **Insert** -> **Plot**, and click somewhere in the blank window.
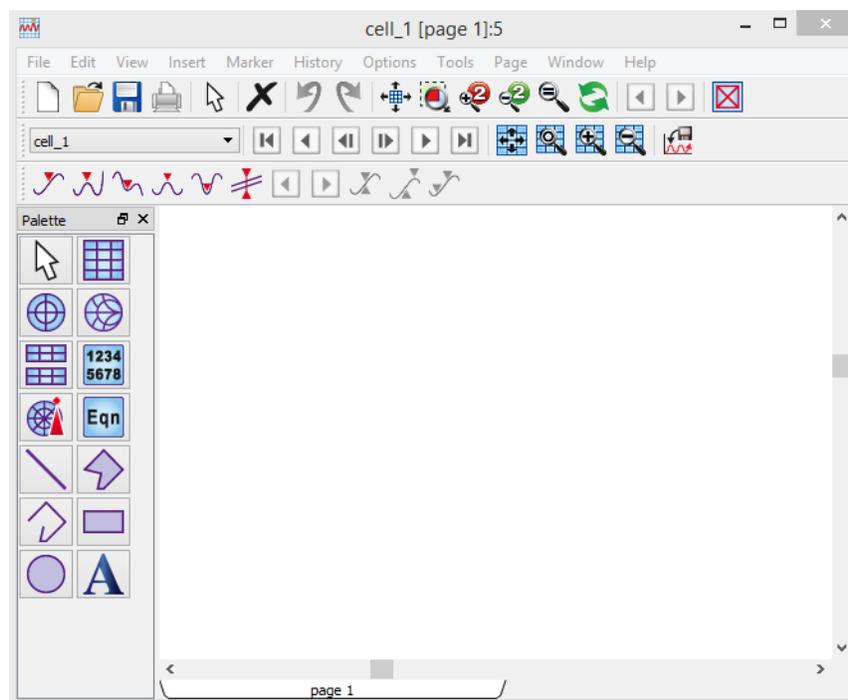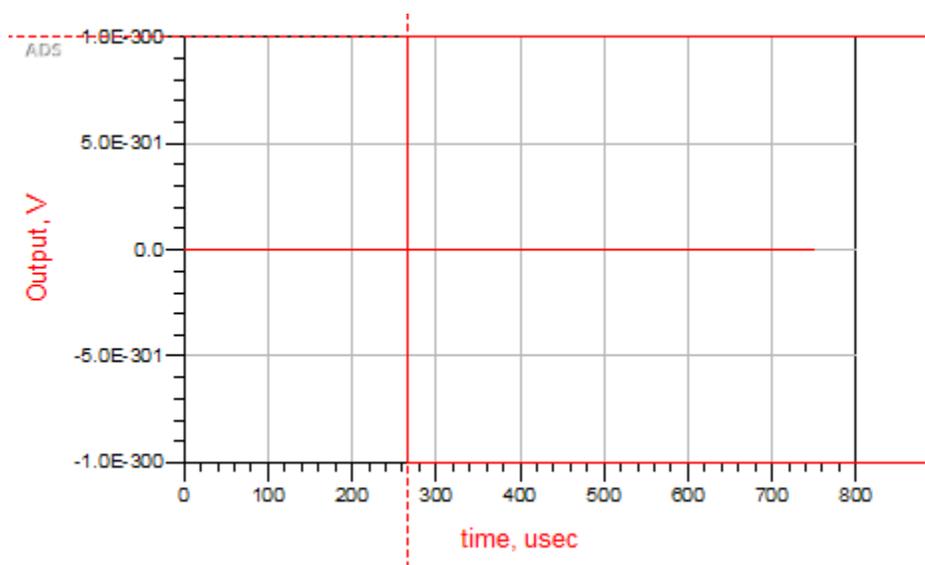
Step 11)
A window similar to the one shown should immediately appear, prompting us to add data to our plot. We care about the node that we named **Output**, so let's add it to the "traces" window.
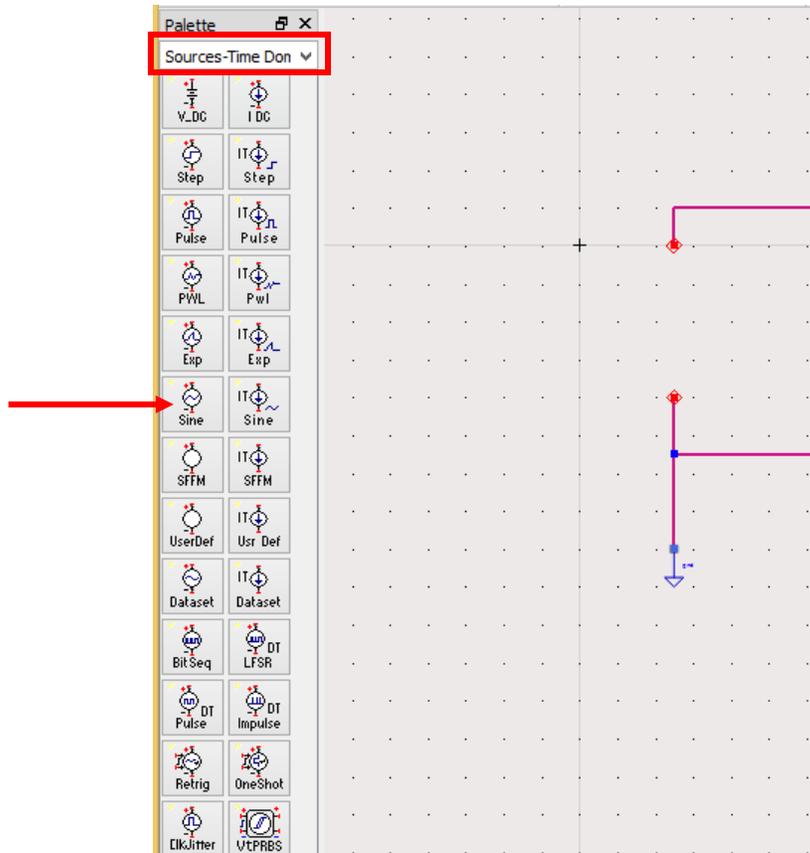

Step 12)
Click **Ok**, and you'll be shocked to see we have no data. Nothing. No signal. How can this be?

An incredibly common pitfall for first-time ADS users is using an **AC source** instead of a **Time-Domain sine-wave** when simulating their circuit. This will lead to the exact behavior that we just saw, giving us no data. Hopefully, now that you've made this mistake once, it'll save you significant headache in the future.

Step 13)

We want to swap our AC source to a Time-Domain sine-wave as rerun the analysis. This source can be found under **Sources-Time Domain**.

Step 14)

Now, if we redo our simulation and plot the output waveform, we should see something very similar to the window shown. This window contains some tools to help you analyze this waveform, determine its max/min values, etc, which I've demonstrated very briefly below.