



BME 212 Electronics Laboratory

Experiment #9 Multisim Tutorial



Objective



The objective of this experiment is to introduce the Multisim program where the next experiments will be designed.

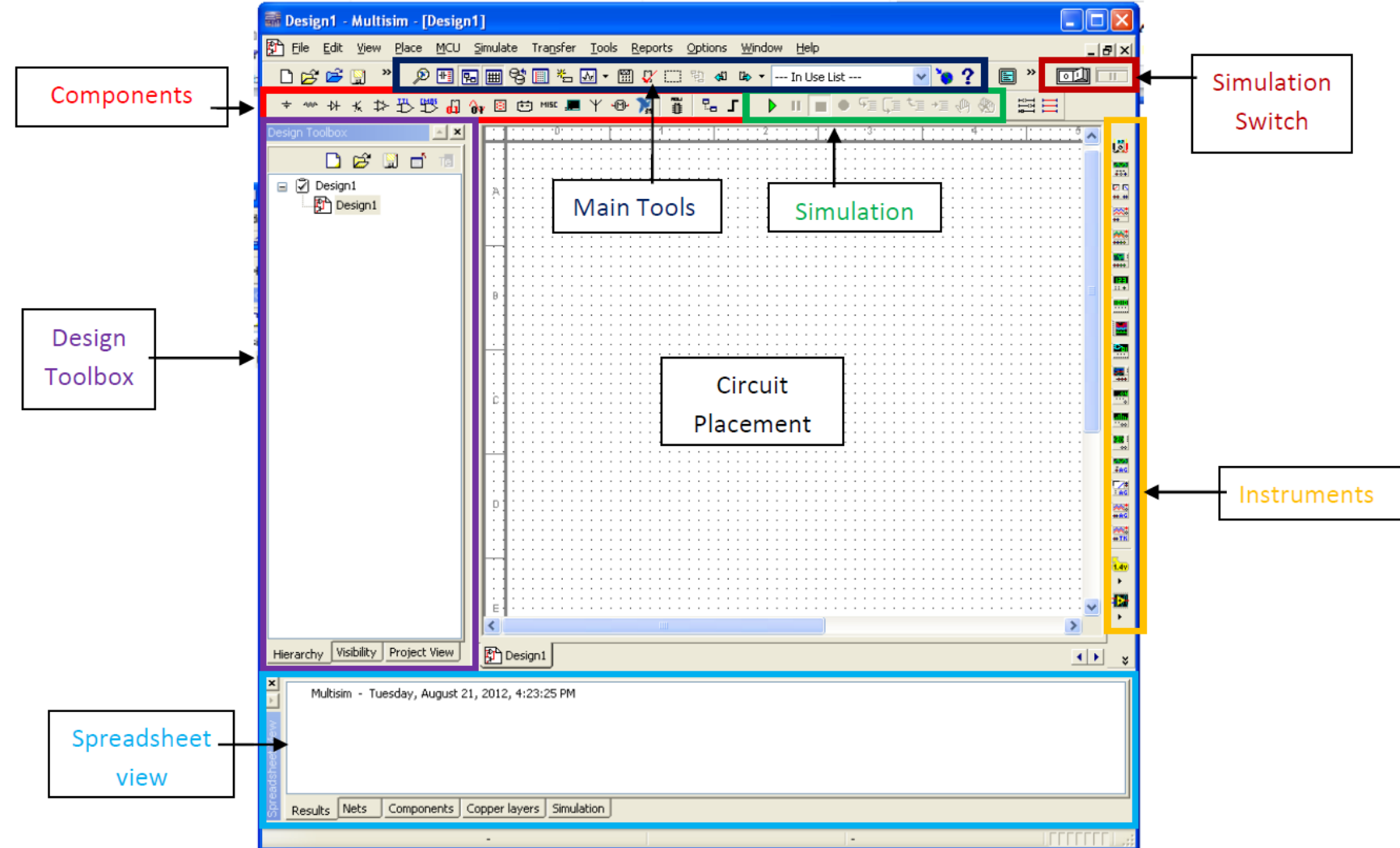


Multisim 11.0 Tutorial

Start:

Click Start ->

- ➔ Programs
- ➔ National Instruments
- ➔ Circuit Design Suite 11.0
- ➔ Multisim 11.0



Placing Components

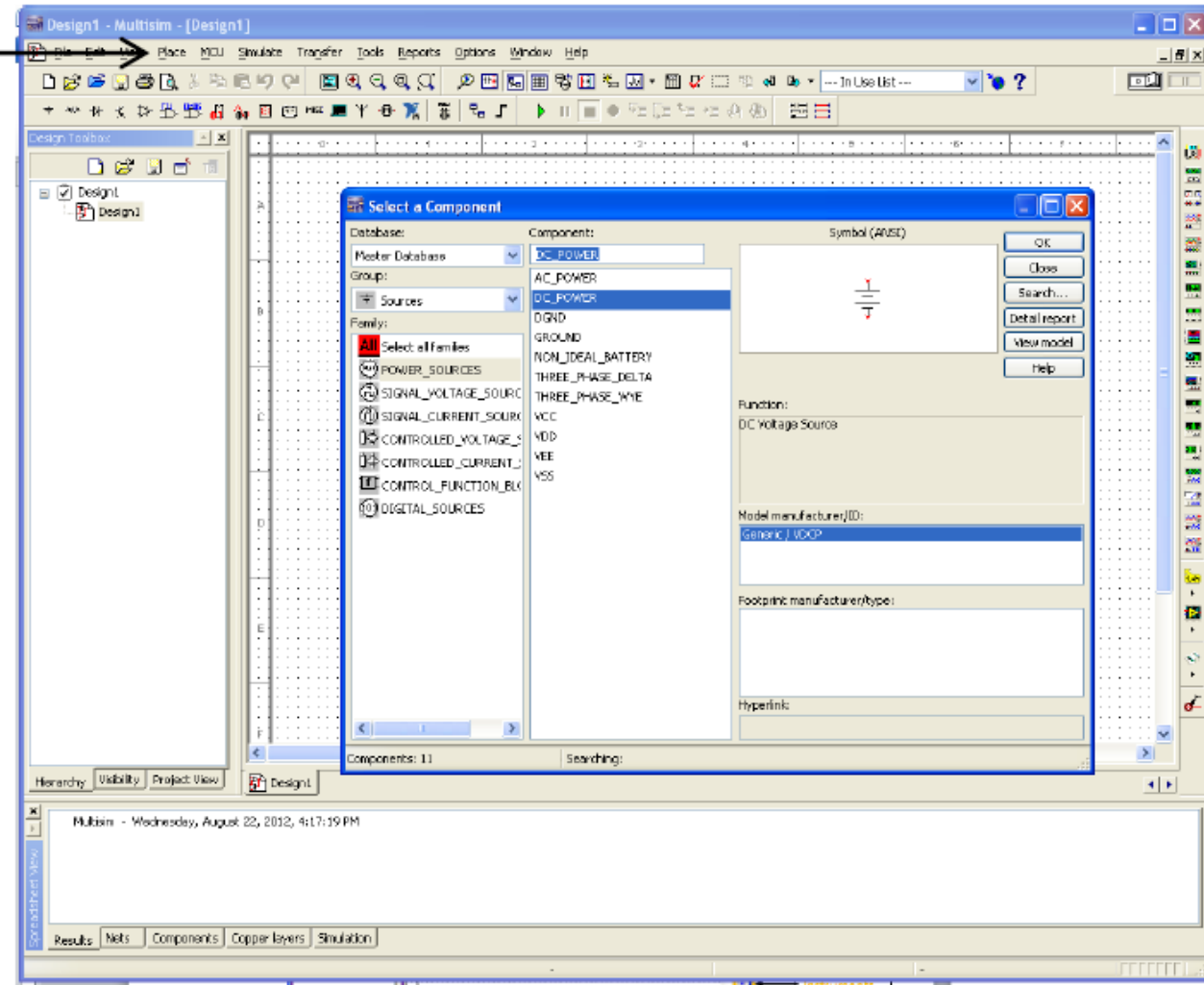
Placing Components:

To place a new component:

→ Place

→ Components

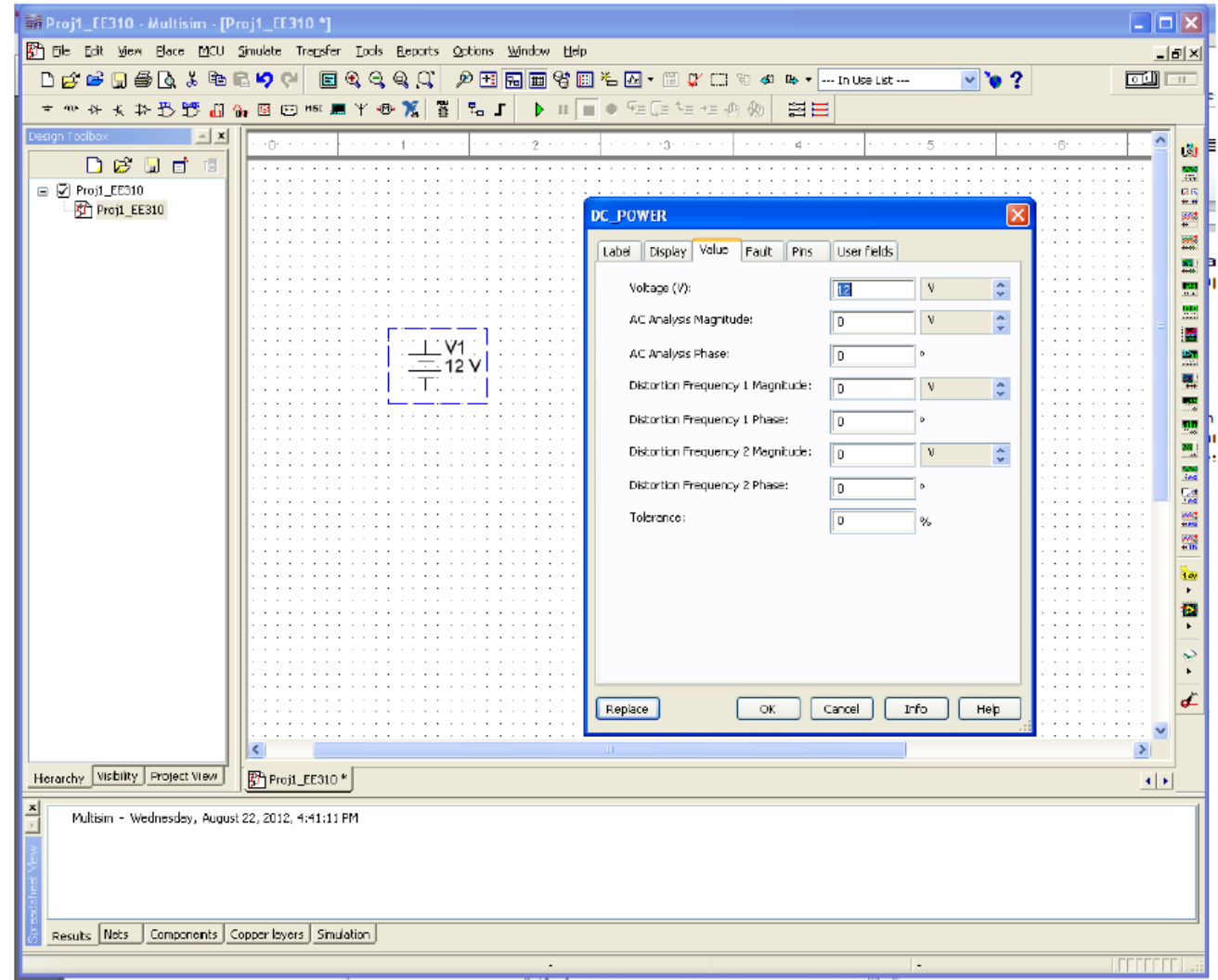
You will see a component selection window that pops up where you can place your components and sources.



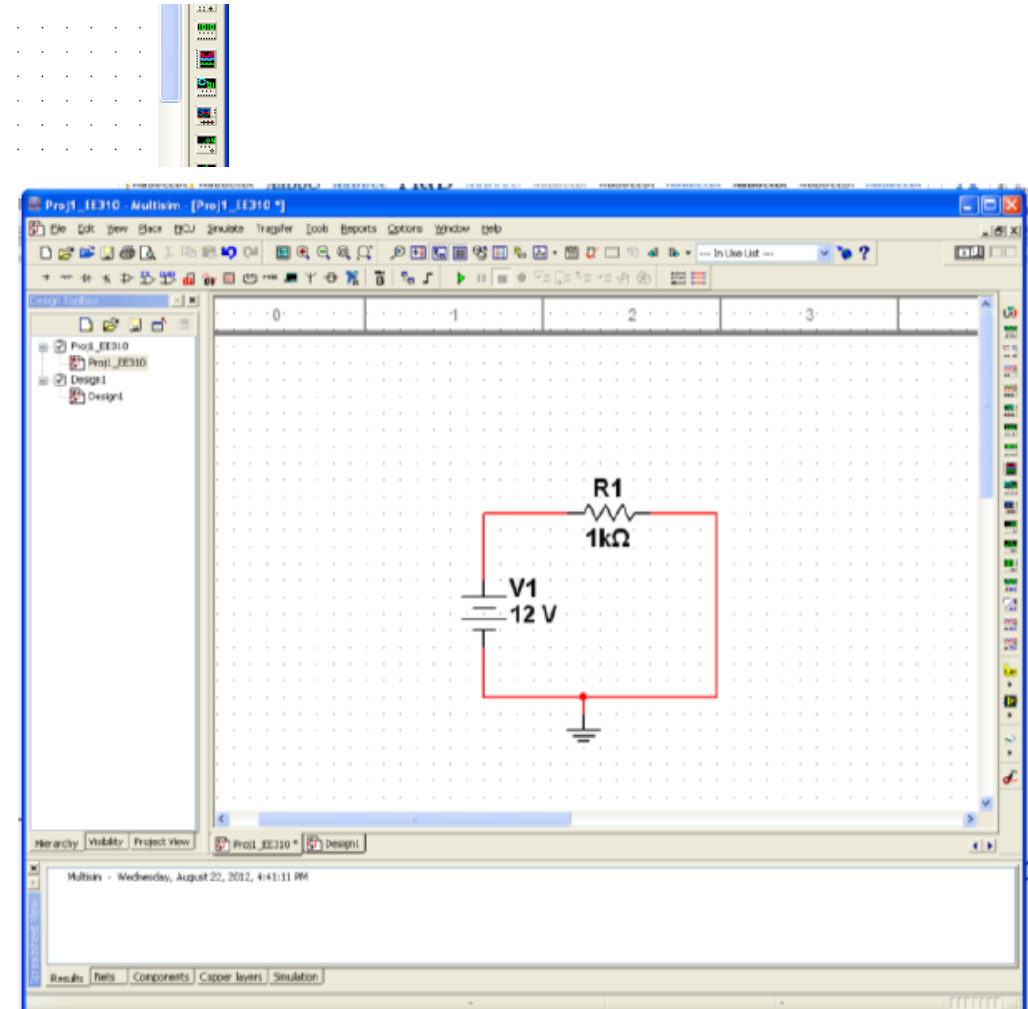
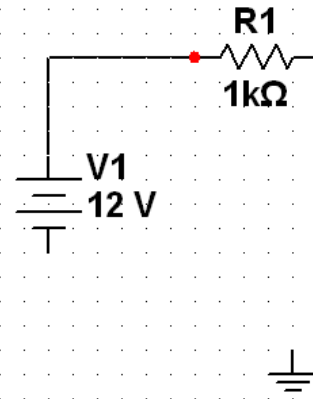
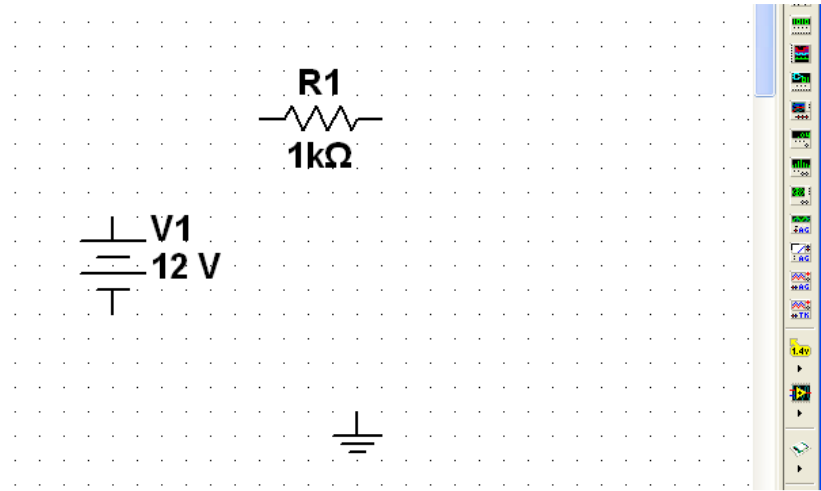


Changing Component Parameters:

→ To change the values or names of the component, you can either right click on the component and select properties, or double click on the component, and a screen will pop up allowing you to change the properties and values of the component.



Wiring:

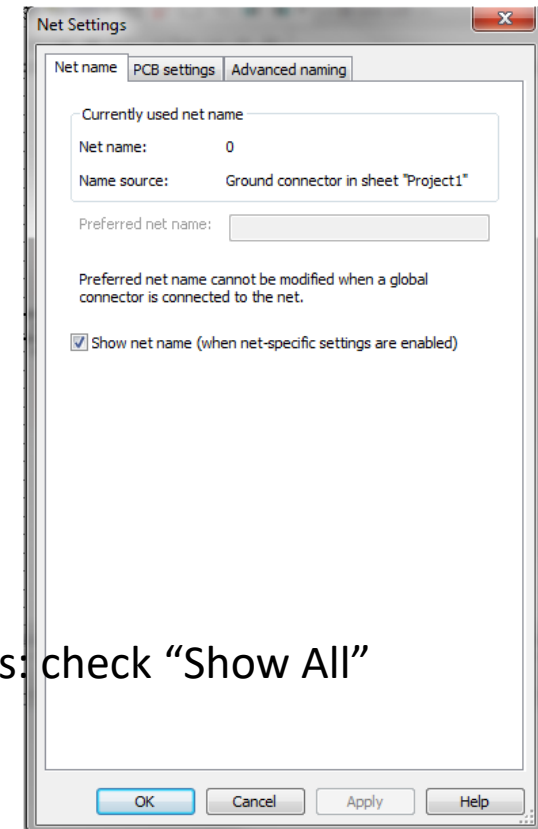
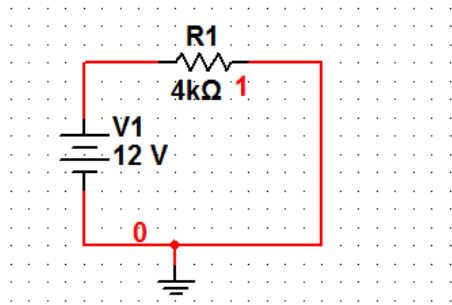


When wiring, place cursor at the corners of component and you should see a black dot appearing, click the mouse once and drag your wire to the edge of another component where a red dot should appear, click again to connect.



Displaying Node numbers/ Wire names:

To show the net names of the wires, select the desired wire and right click Properties, and a Net Settings window should pop up. Checking the show net name box will allow one to see the wire names/net on the circuit.



One can also do Options → Sheet Properties → Circuit → Net Names: check “Show All”

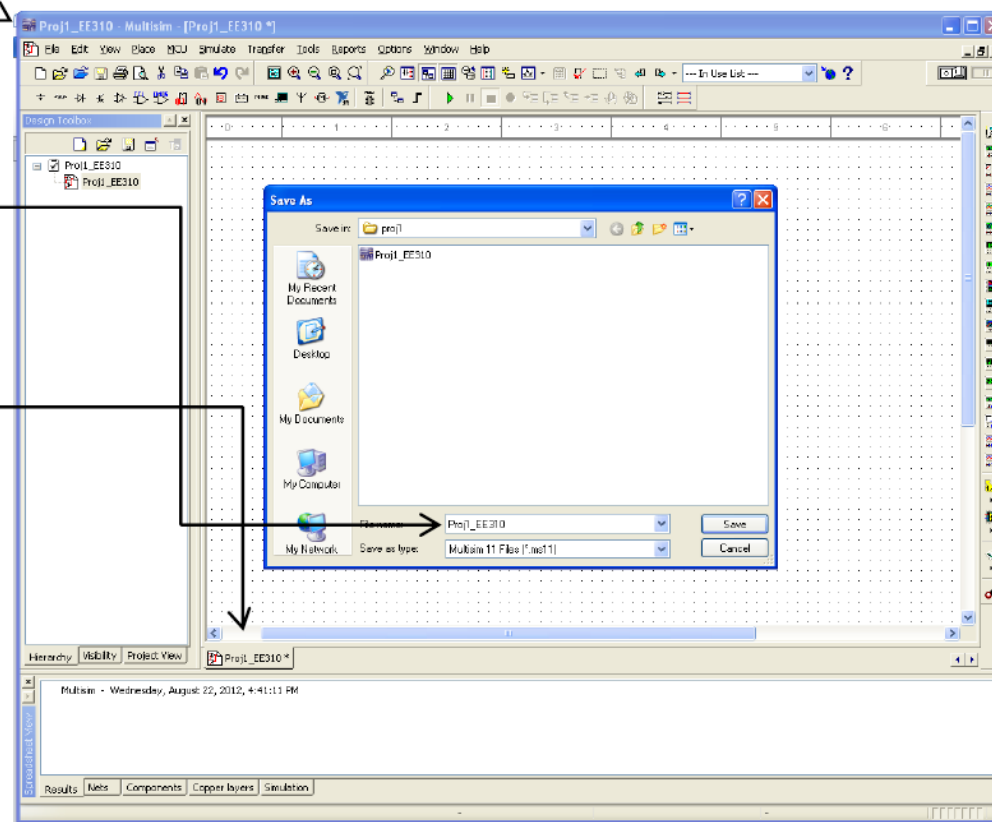
Saving

Saving:

To save your project:

- ➔ File
- ➔ Save as
- ➔ Name your design

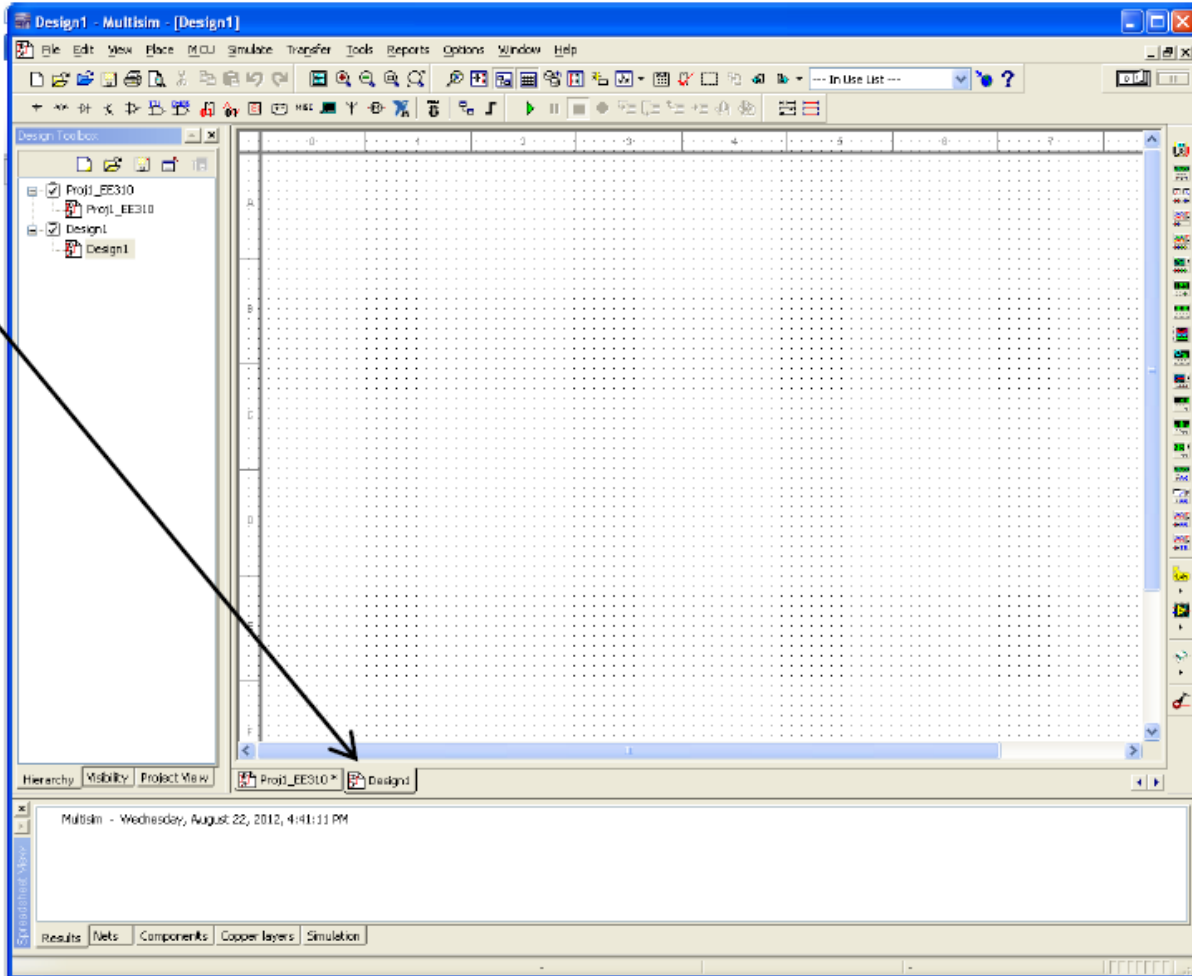
Once saved, you will see the name appearing on top of the window and the original name Design1 will change to your new project name.





You can also create more than one project in one setting by:

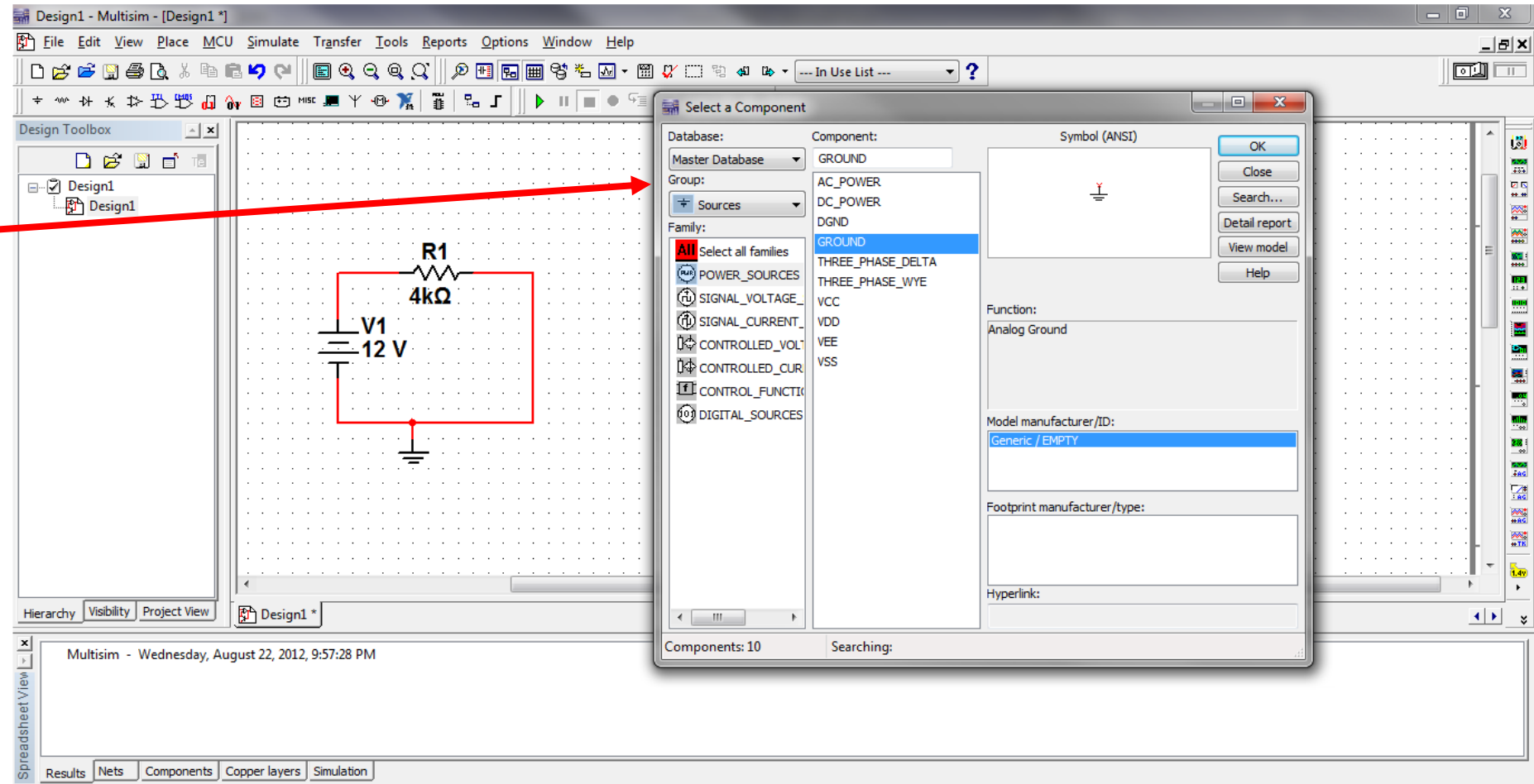
- ➔ File
- ➔ New
- ➔ Design



Simulation Analysis:

First, draw the desire circuit:

- 1. Place -> Components
- 2. In the Select Component windows, under **Group**, select Sources for DC_POWER and GROUND.
- 3. In Group, select Basic and chose RESISTOR.
- 4. Connect the wires to complete the circuit.





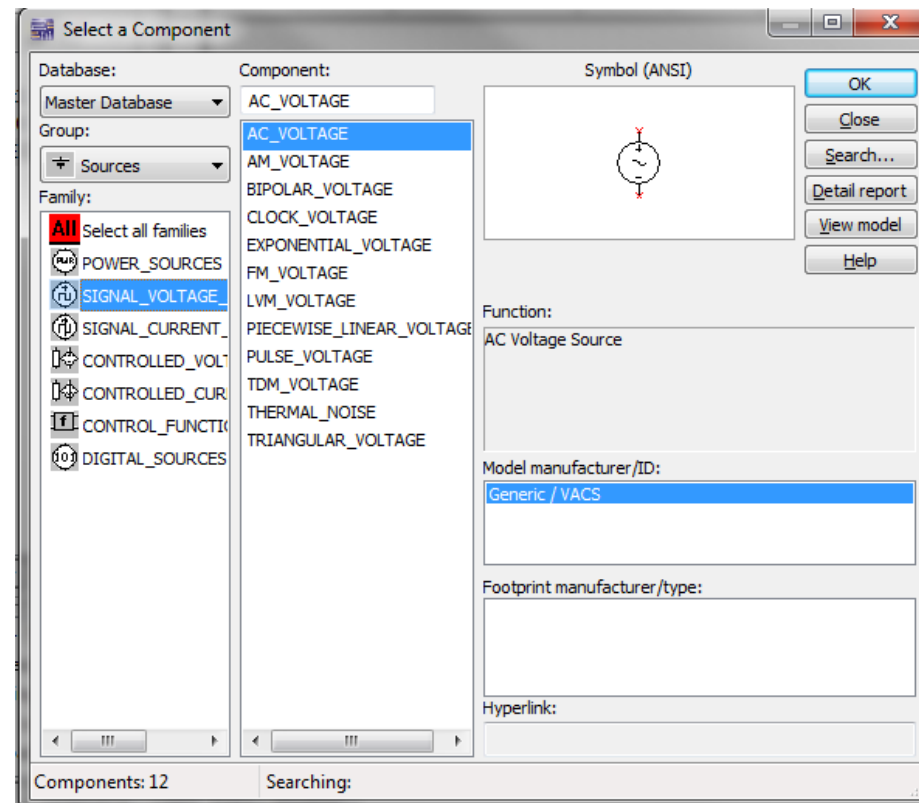
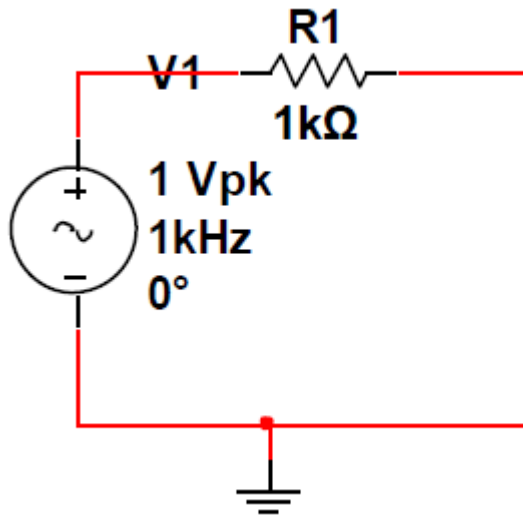
Copy Graphs and Circuits:



- To copy circuit schematic: Select the entire circuit using the mouse and right click → copy
- To copy the graph desired from Grapher View: Edit → copy graph

Transient Analysis:

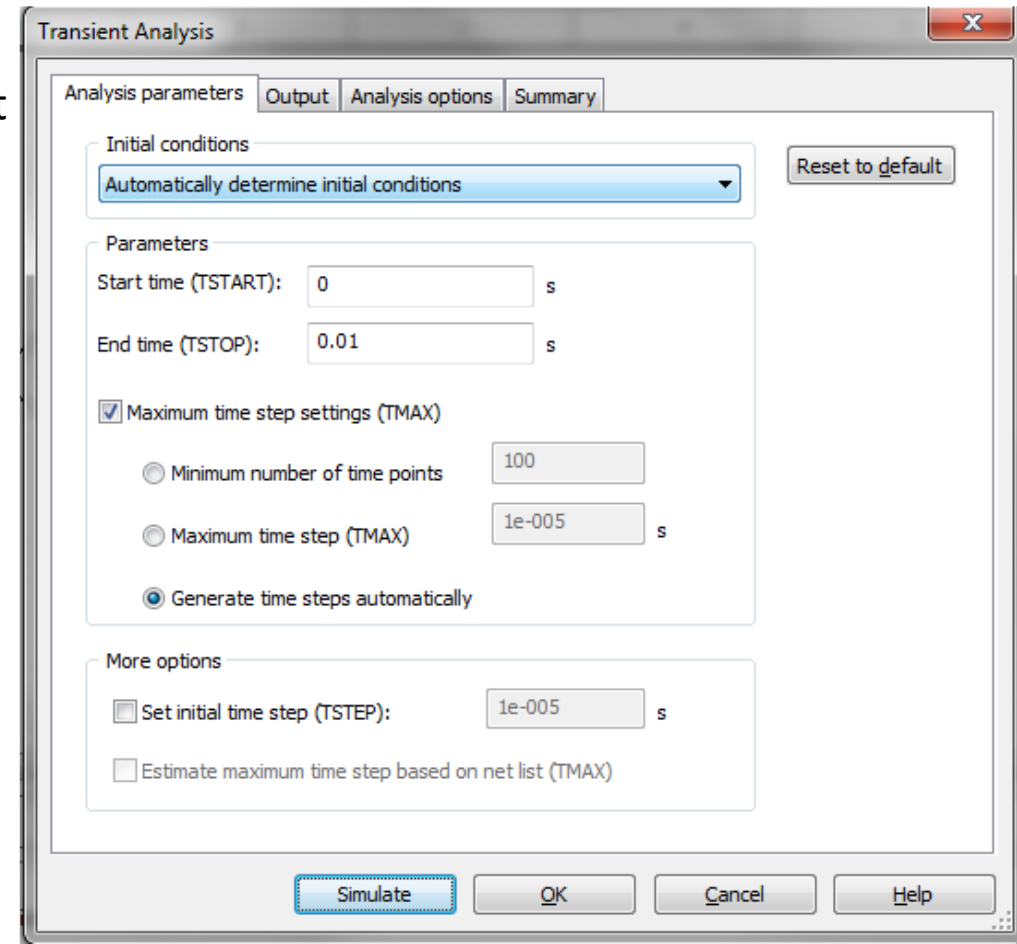
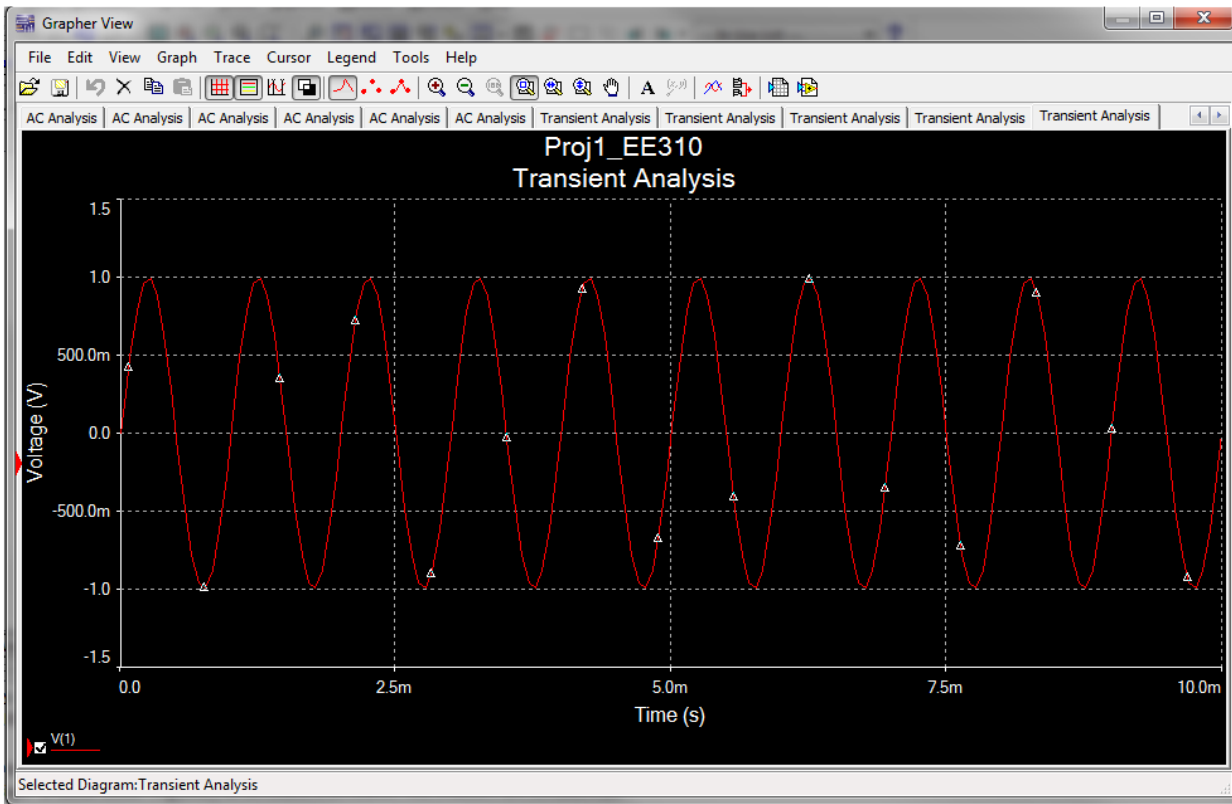
For transient analysis, we can change the voltage source to an **AC_VOLTAGE** under the **SIGNAL_VOLTAGE_SOURCES**.





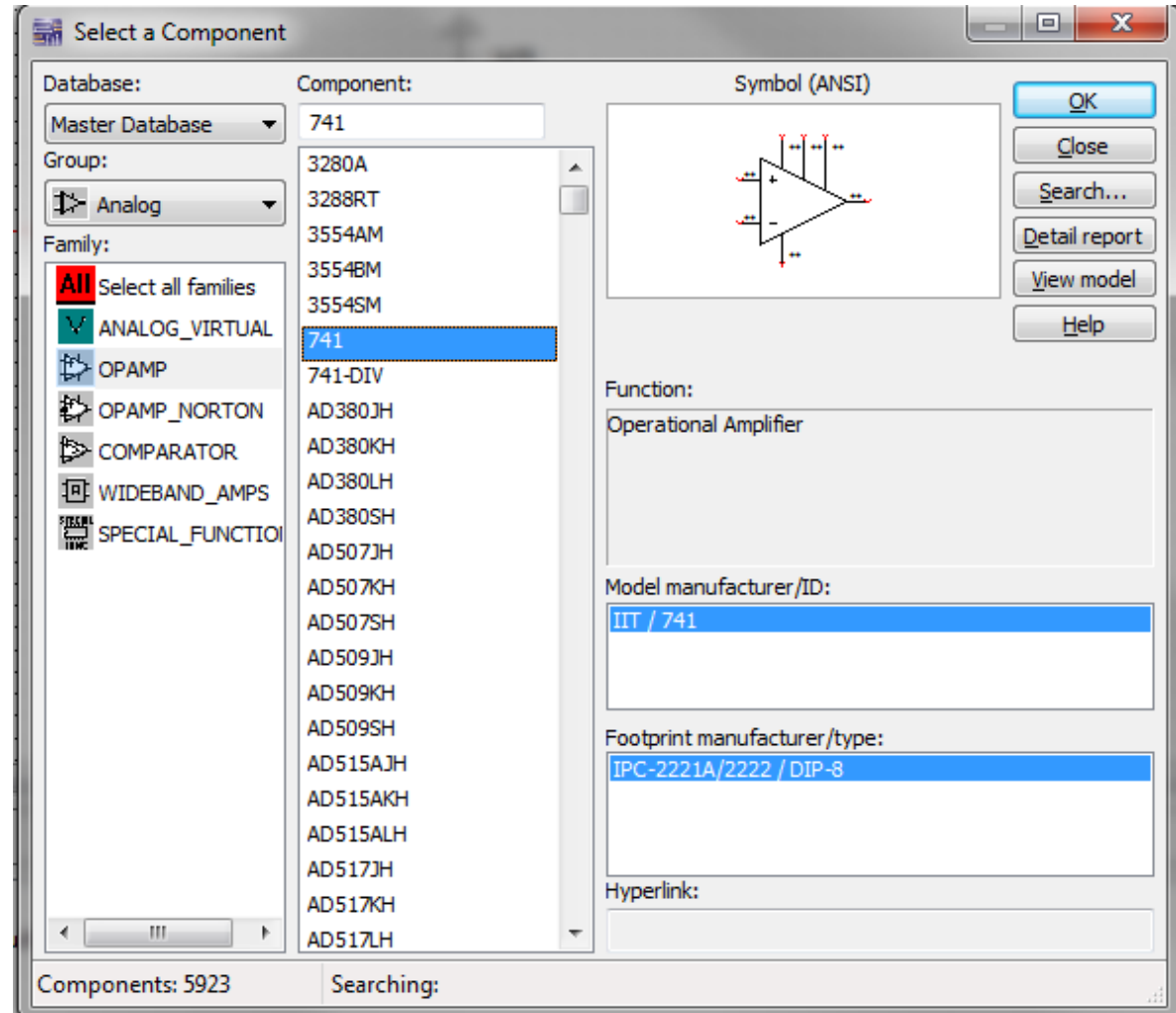
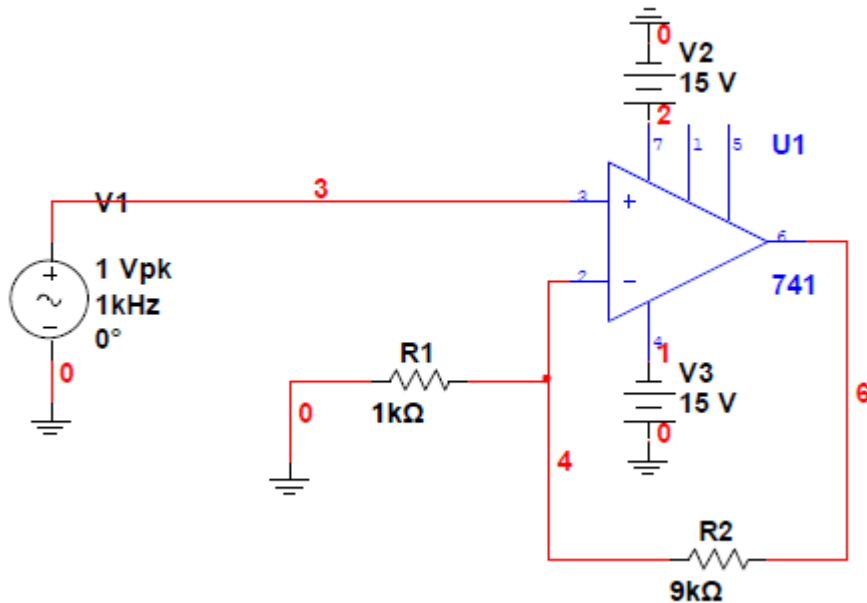
Transient Analysis (Cont.)

- ➔ Simulate
- ➔ Analyses
- ➔ Transient Analysis
- Change the following parameters
- For output, we can verify our voltage source by adding V(1) to our output



AC Analysis:

Draw the following circuit (closed loop inverting amplifier), for the amplifier, under Group, select Analog and select the OPAMP under Family and type in 741 for the amplifier part.



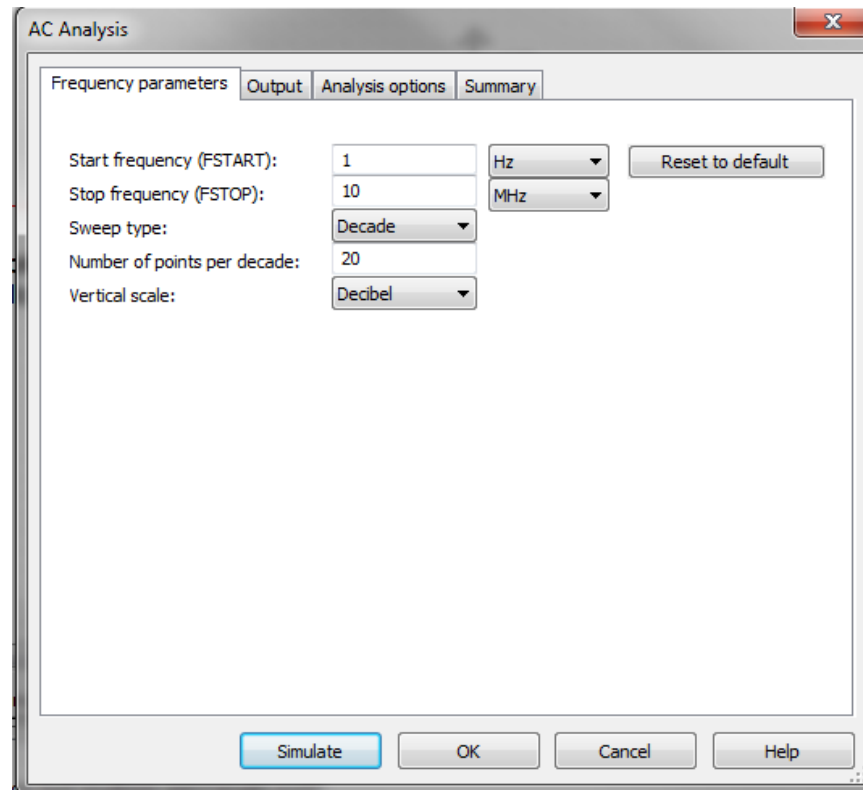


AC Analysis (Cont.)



- Simulate
- Analyses
- AC Analysis

A window for the AC analysis should pop up, change to the following parameters:



For the Output tab, choose V(6), the output of the amplifier.

AC Analysis (Cont.)

Two graphs should be show in the Grapher View, where the first graph shows the magnitude of the gain of the amplifier, and the second graph shows the Phase vs. Frequency (we will usually look at only the first graph). Again, if we would like to see or plot points on the graph, we can choose Cursors, and move or type in our desired location.

