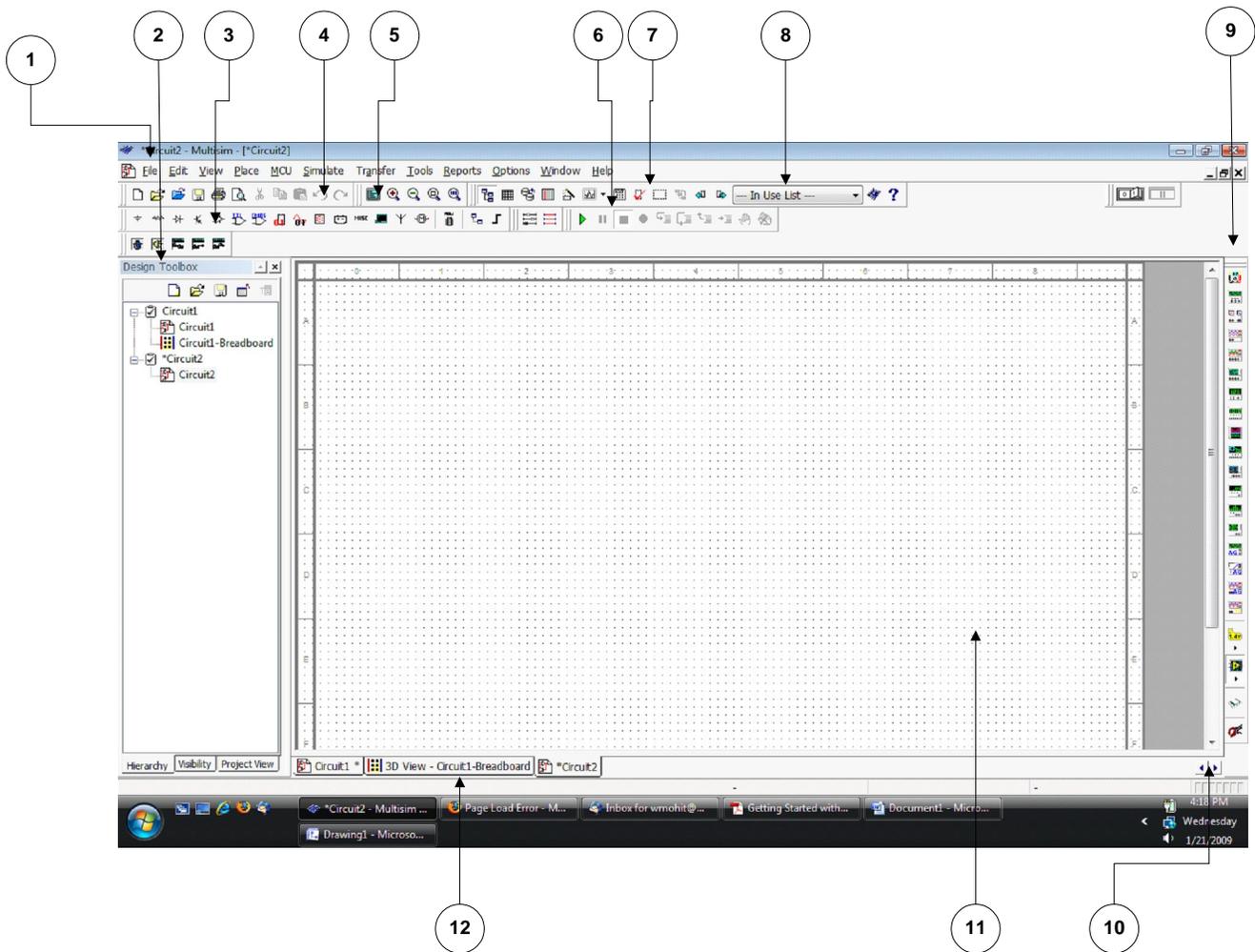


Multisim Manual

Multisim is the schematic capture and simulation application of National Instruments Circuit Design Suite, a suite of EDA (Electronic Design Automation) tools. It is similar to PSpice, but it is more easy to use in practical sense and has lots of features to make circuit drawing/simulating, a really simple task. Here is window of multisim, as it appears first time when you start the software.

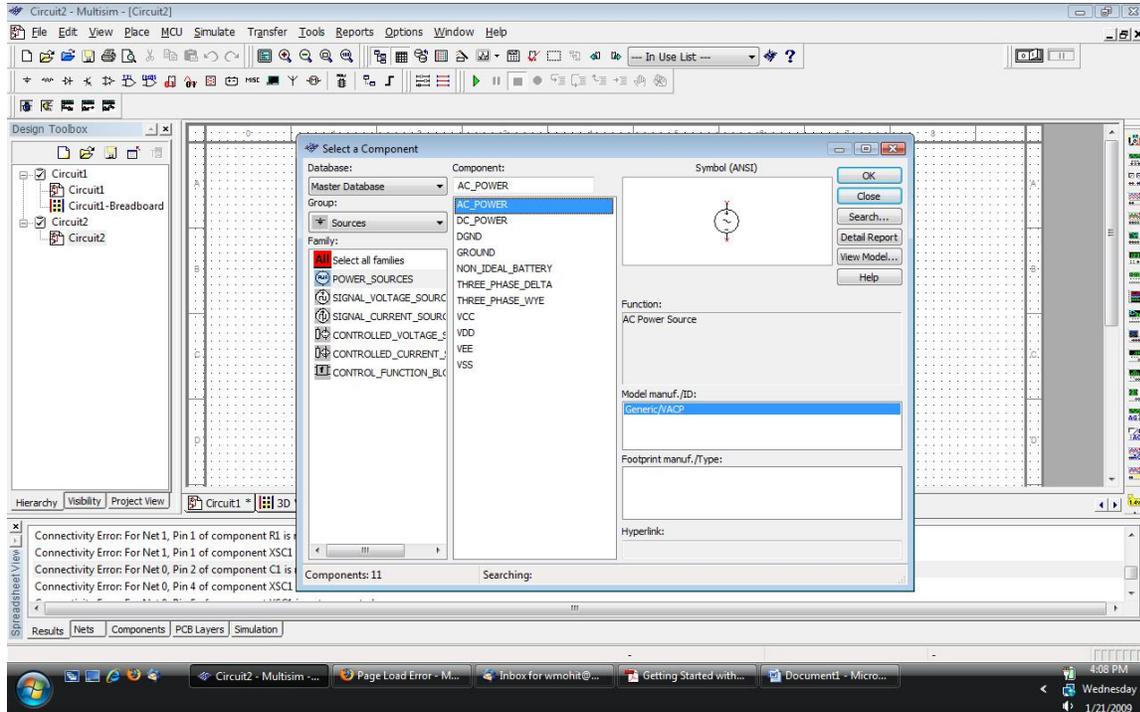


1. The **Menu Bar** is where you find commands for all functions.
2. The **Design Toolbox** lets you navigate through the different types of files in a project (schematics, PCBs, reports), view a schematic's hierarchy and show or hide different layers.
3. The **Component** toolbar contains buttons that let you select components from the Multisim databases for placement in your schematic.
4. The **Standard** toolbar contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.
5. The **View** toolbar contains buttons for modifying the way the screen is displayed.
6. The **Simulation** toolbar contains buttons for starting, stopping, and other simulation functions.
7. The **Main** toolbar contains buttons for common Multisim functions.
8. The **In Use List** contains a list of all components used in the design.
9. The **Instruments** toolbar contains buttons for each instrument.
10. **Scroll Left –right** is to ensure ease in handling larger designs.
11. The **Circuit Window** (or workspace) is where you build your circuit.
12. **Active tab** indicates the current active circuit window.

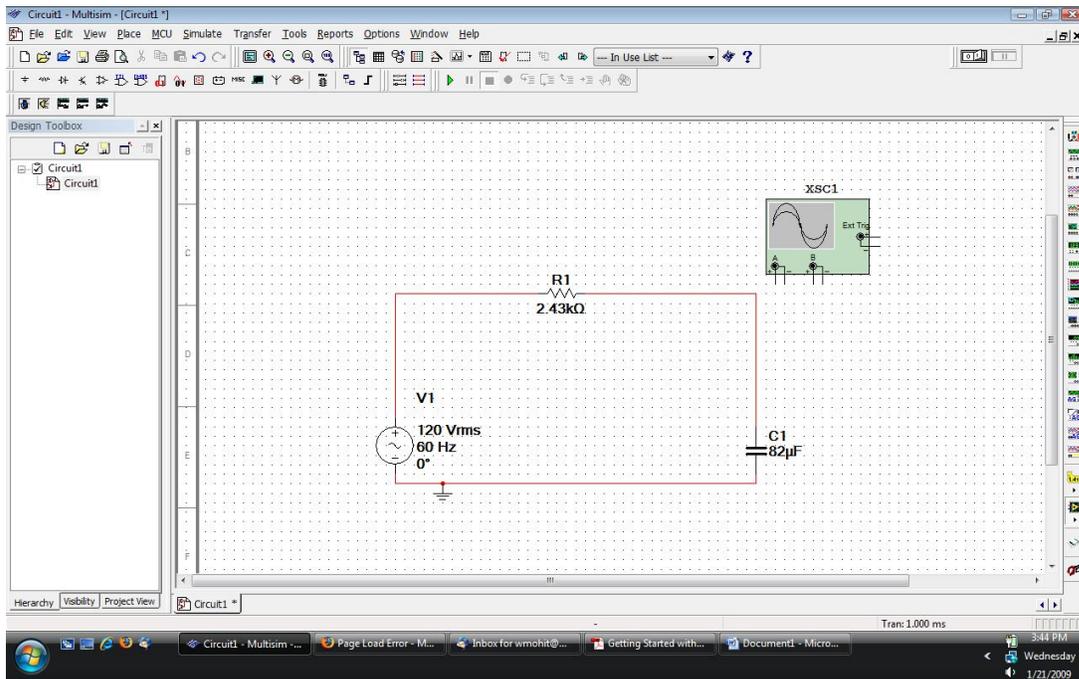
Let's take an example of a RC circuit. We will simulate the circuit to perform the transient analysis. At any point of time you can click **F1** for help on the tool.

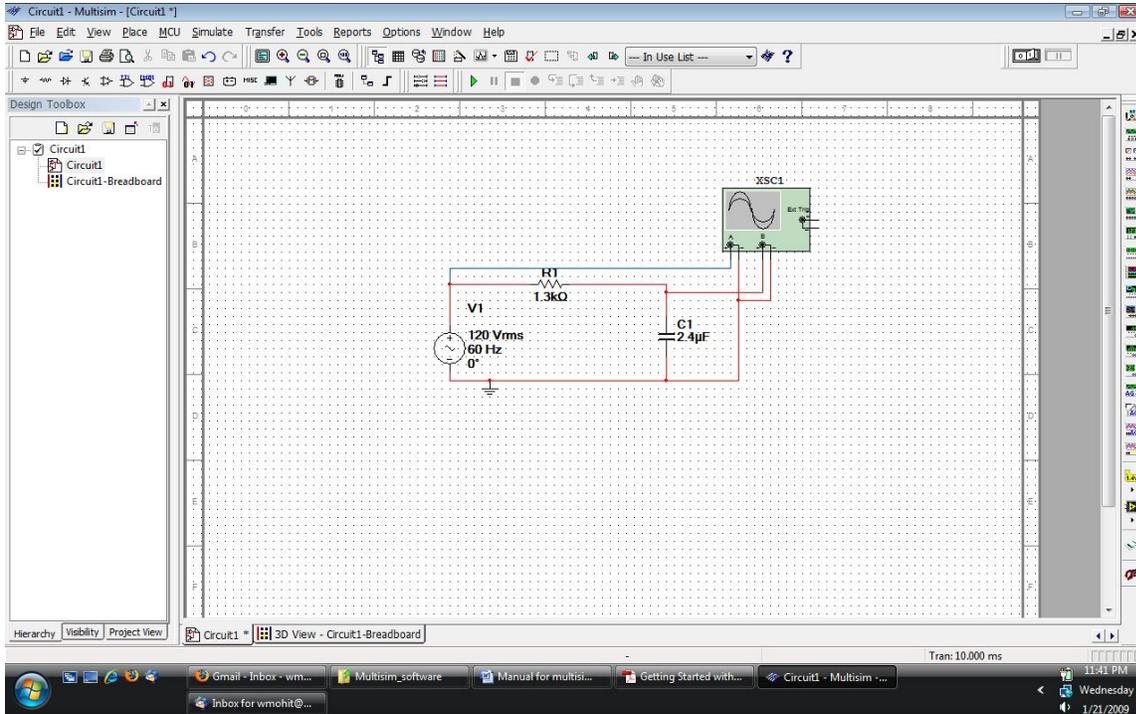
Please follow the steps:

1. Select **Start»All Programs» National Instruments» Circuit Design Suite 10.1» Multisim 10.1**. A blank file opens on the workspace called Circuit1.
2. Select **File» Save As** to display a standard Windows Save dialog.
3. Select **Place» Component** to display the **Select a Component** browser, navigate to the group **Sources** and click on **POWER_SOURCES**. Then choose the **Family: AC_POWER** option. The component appears as a "ghost" on the cursor. (Once you have selected the desired **Group** and **Family**, start typing the component's name in the browser's **Component** field. As you type, the string appears in the **Searching** field at the bottom of the browser.)

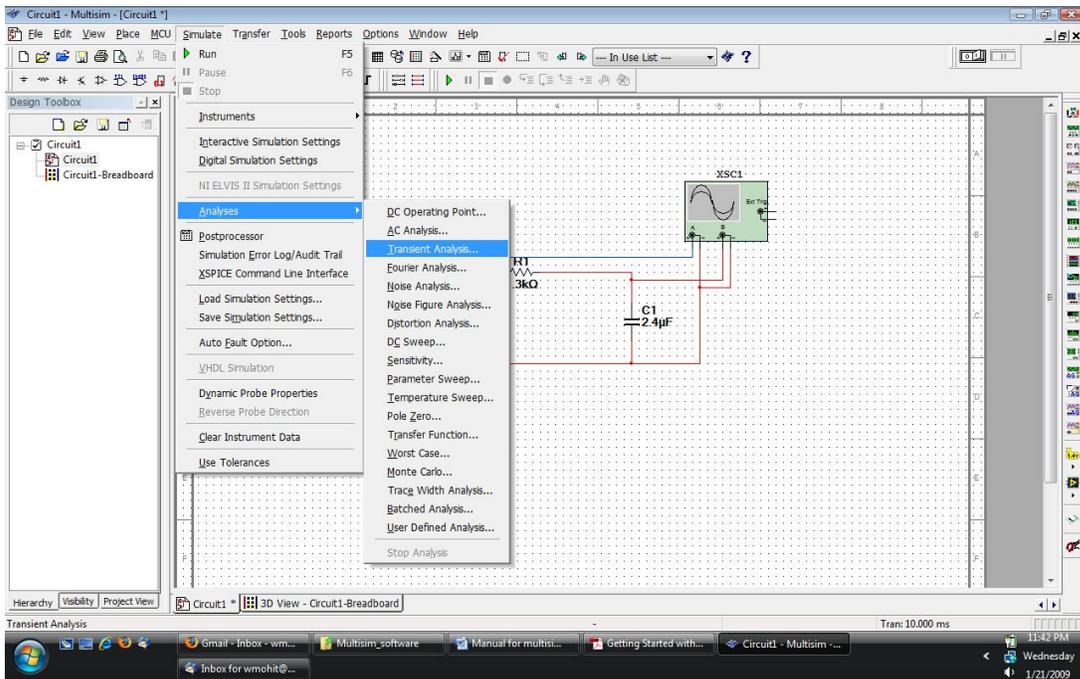


4. Move the cursor to the bottom-right of the workspace and left-click to place the component. Similarly, find the other components and place them. When a part is on the workspace and you want to place the same part again, highlight it and select **Edit»Copy**, then **Edit» Paste**. You can also select it from the **In Use List** and click to place it on the workspace. Press **Ctrl+R** if you want to rotate the component. The oscilloscope is obtained (by dragging it) from the component window on the right hand side of main window.

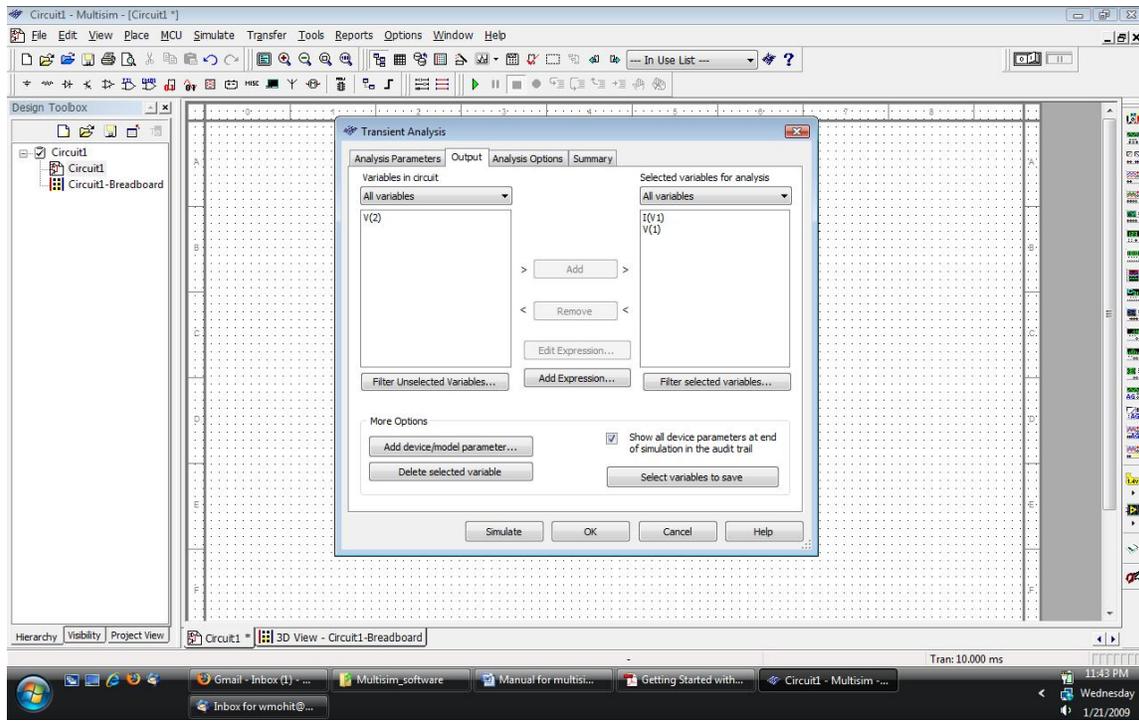




5. All components have pins that you use to wire them to other components or instruments. As soon as your cursor is over a pin, Multisim knows you want to wire and the pointer changes to a crosshair. You can also use **ctrl + q** for wiring the circuit. Don't forget to add **Ground** to the circuit (available in **Sources** option in **Place>>Component**).
6. Choose the type of analysis you want to perform by clicking **Simulate>> Analyses>>**. Here, we have chosen transient analysis.

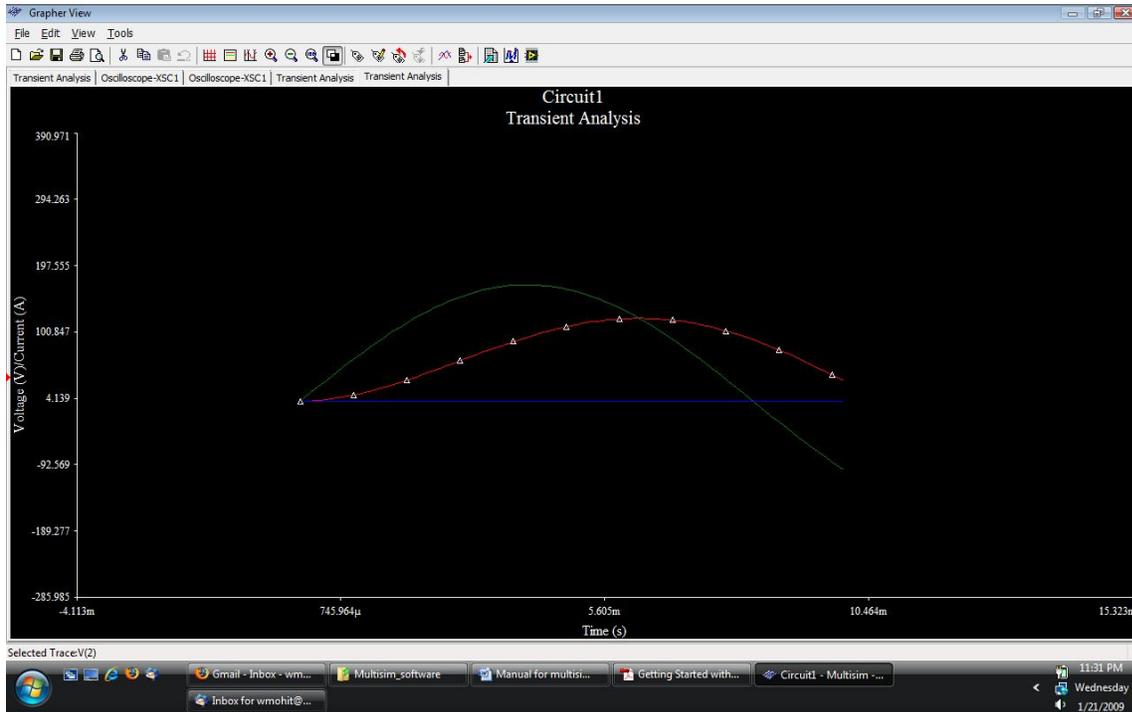
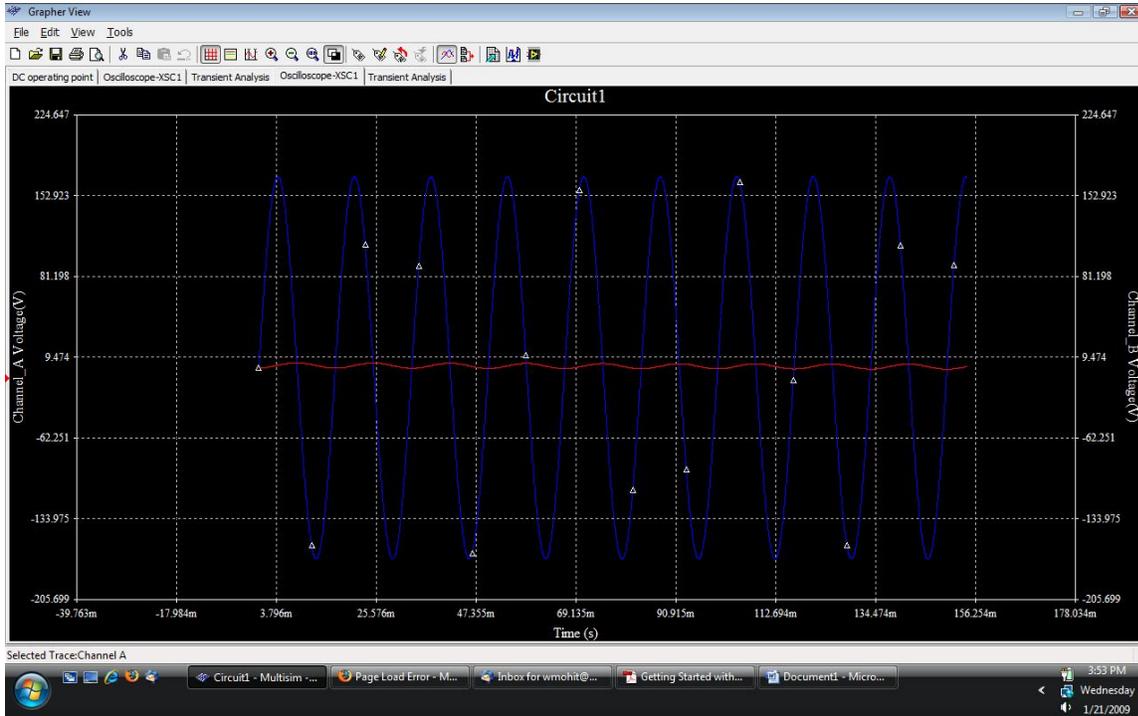


7. Select **Simulate» Analyses»Transient Analysis** and click on the **Output** tab. Add I(v1) to the right column by first clicking on I(v1) in left column and then pressing **Add** tab.



8. Also choose the time (from transient **Frequency parameters**).
9. Click on Simulate tab. The output window appears which consists of tab for oscillator output as well as transient waveforms. Different colored waves can be viewed by choosing the color of respective wire of the electrical quantity (voltage and current). Right click on wire in the circuit and then click on **Color segment...** to choose the color of wire and thus the waveform color (after simulation).

Principles of Electrical Engineering Lab 1



10. With multsim, you can create 3D breadboard with components placed. You can explore yourself this option by clicking on show breadboard tab on main toolbar. Figure below shows example of 3D breadboard created in multsim.

