

Spice Directions

Layout

Open the program up. On the right hand side is a list of buttons, on top is the menu options and below that are the tabs. In the middle is the work space. (If you have opened the program for the first time you will need to do the following to get a blank workspace File=>New Schematic). On the workspace is a yellow outline, this is where you will build your circuit.

Buttons

The buttons from top to the bottom are as so:



- Analyze the circuit



- Move an Item in the Circuit (the default mouse pointer)



- Lines, Wires, Nodes, and “No Connection” points



- Resistors, Pots, Capacitors, Inductors



- Diodes, Fets, BPNs



- Non-linear Circuit Components (ie Op Amps, Transformers)



- Voltage Source, Battery Source, Current Source, Current/Voltage ctrld sources, Gnd



- Test Points









- Text



- Digital Logic Components (NAND, OR, AND, XOR)

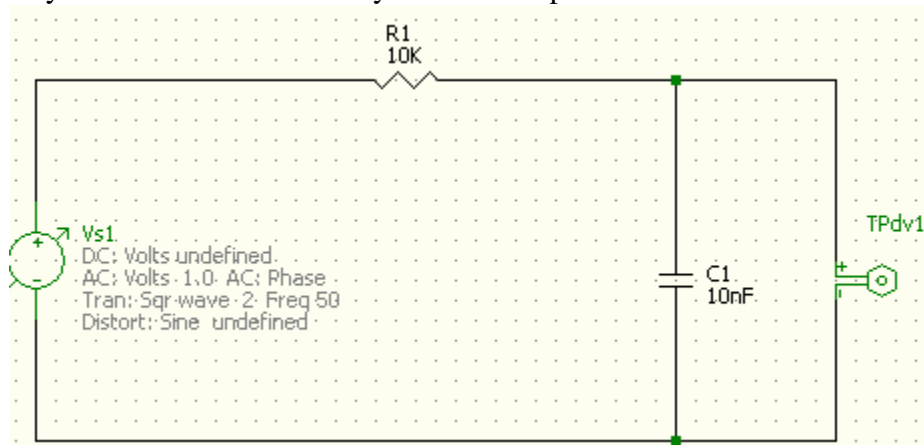
The most important buttons are the Wires, Resistors/Capacitors, Diodes, Voltage/Battery Sources and Test Points.

Rules

- A circuit must have a test point at the place where you want to analyze either the Current or Voltage.
- A circuit must have a Ground point (found in the Voltage Sources button as )
- A circuit must be a complete loop. Any open points that must exist should be labeled with the “No Connection” point found in the Wires button as .
- There can only be one Voltage Source (Maybe Two) the rest must be batteries. A Voltage Source and batteries can be found in the Voltage Source Button as , a Battery is: , a current source is: , the equivalent of a battery for current is: .


An Example Circuit

The best way to see how to build a circuit is to build one for you. Here is an example circuit for the problem on your homework that said you can use Spice:



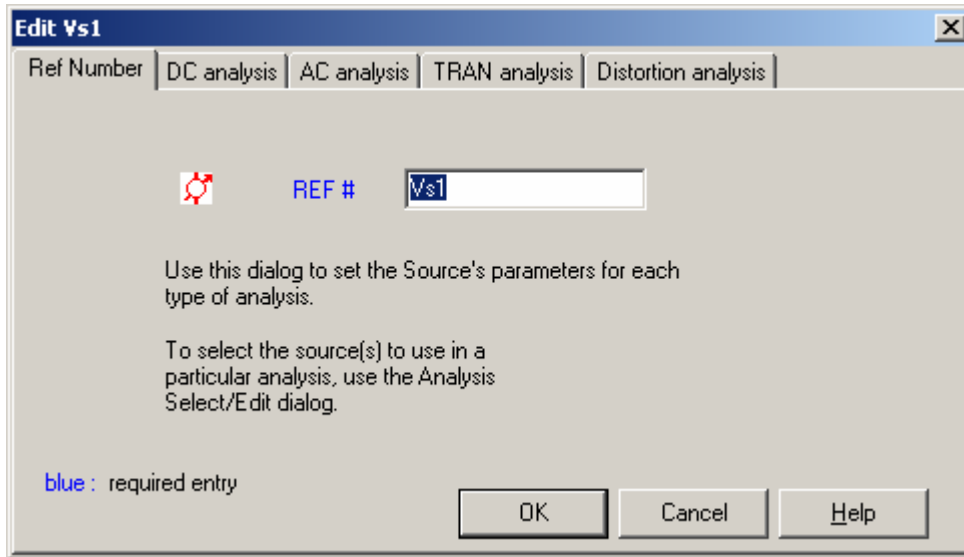
Build a Circuit

To build this Circuit do the following steps:

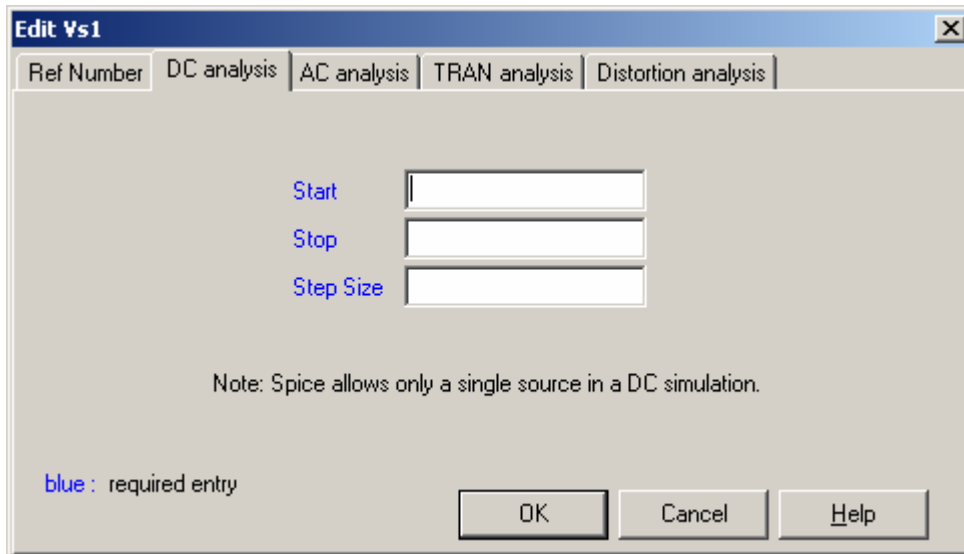
1. Click on the Voltage Source Button, Click on the Voltage Source, and then your mouse Icon should change to an outline. Click on the Workspace inside the yellow line to place the voltage source.
2. Click on the Resistor Button and choose the Resistor, again your mouse will turn to an outline, click on the workspace to place the resistor.
3. Click on the Resistor Button and choose the Capacitor, again your mouse will turn to an outline, click on the workspace to place the resistor
4. Click on the Test Point button and choose the voltage test point that has a positive and negative end . Again your mouse will turn to an outline, click on the workspace to place the resistor.
5. Your components will be mis-rotated. To rotate a component, click on it and it will be highlighted red, the hit the “R” button on your keyboard. You will need to rotate the resistor and the test point.
6. Arrange your circuit properly, as shown above. Then click on the Wire Button, and select the Wire (red line). Your point will change to a pencil. Click on one end of a component to start the wire, and then click on the beginning of another component to end the wire. The components will be connected together. Do this for the whole Circuit

Setting Voltage Source Parameters

Double click on the Voltage Source in the Workspace. It will pop up a properties menu. Depending on they type of source you have (DC, AC, Wave) you will choose to set a different tab. The First tab: Ref Number is the name of the Voltage Source.



The Second tab is for a DC Source

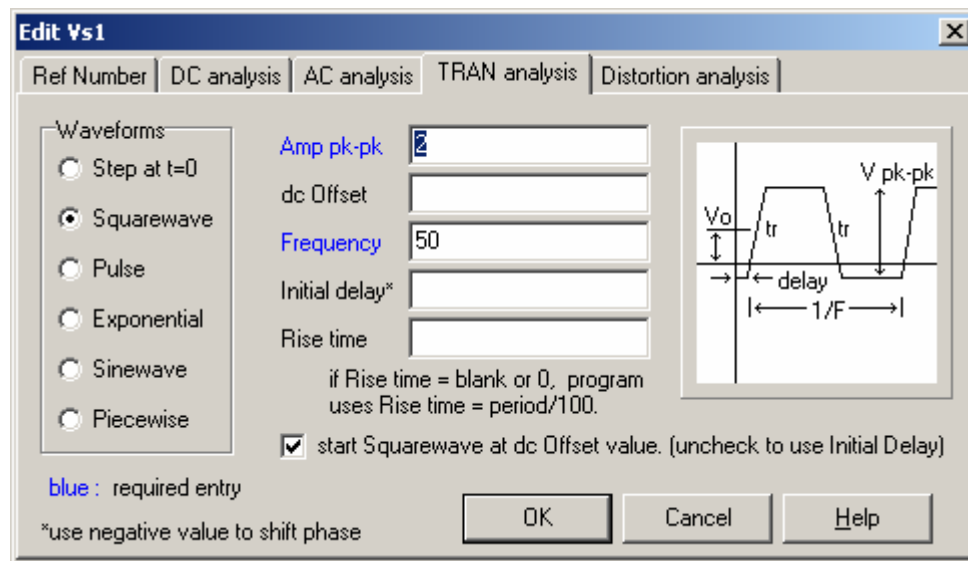


Start is the starting Voltage of the source

Stop is the ending voltage

Step Size is the size of each increment that the analysis will use.

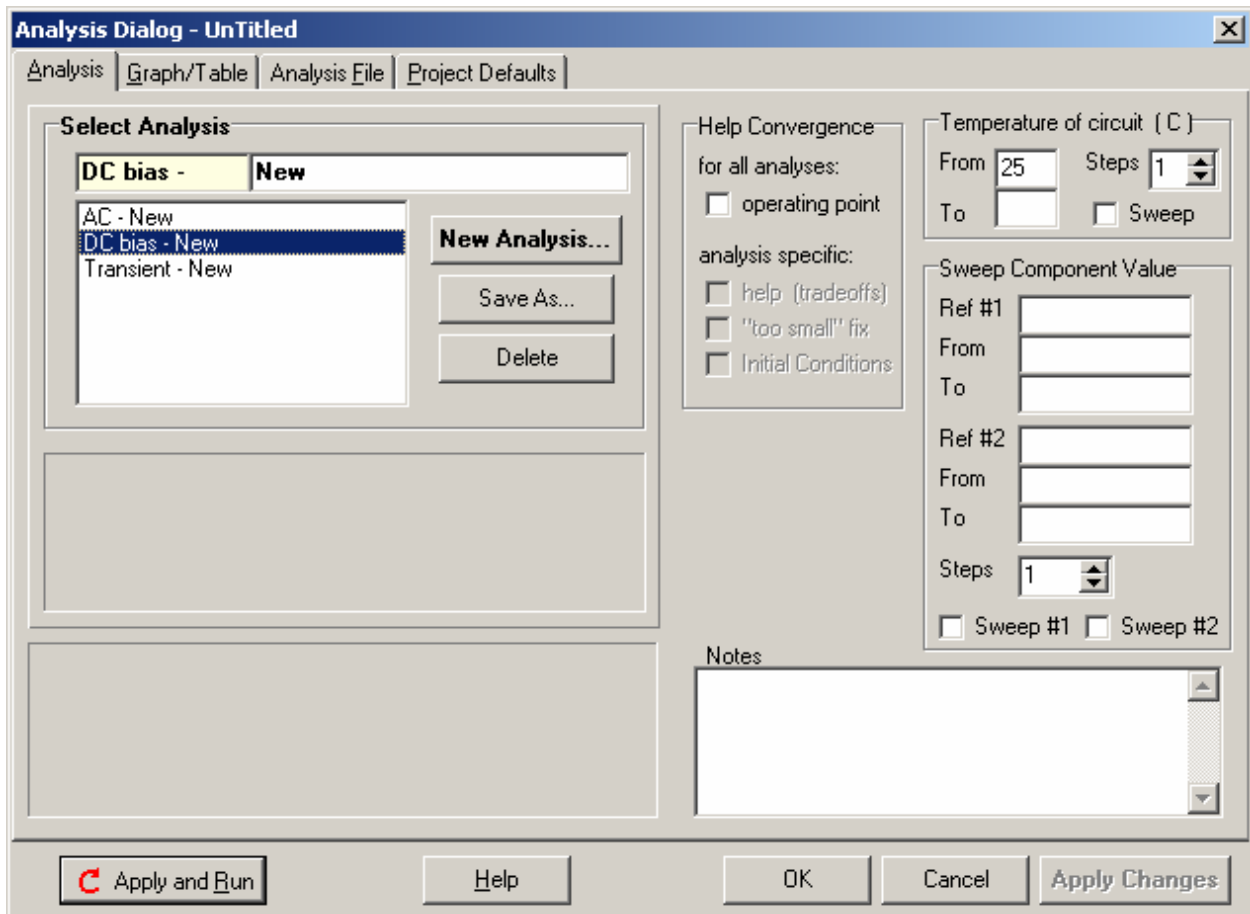
The Ac analysis tab is for an AC wave. Most of the time, however, you will use TRAN analysis for AC. This will be for a continuously wave.



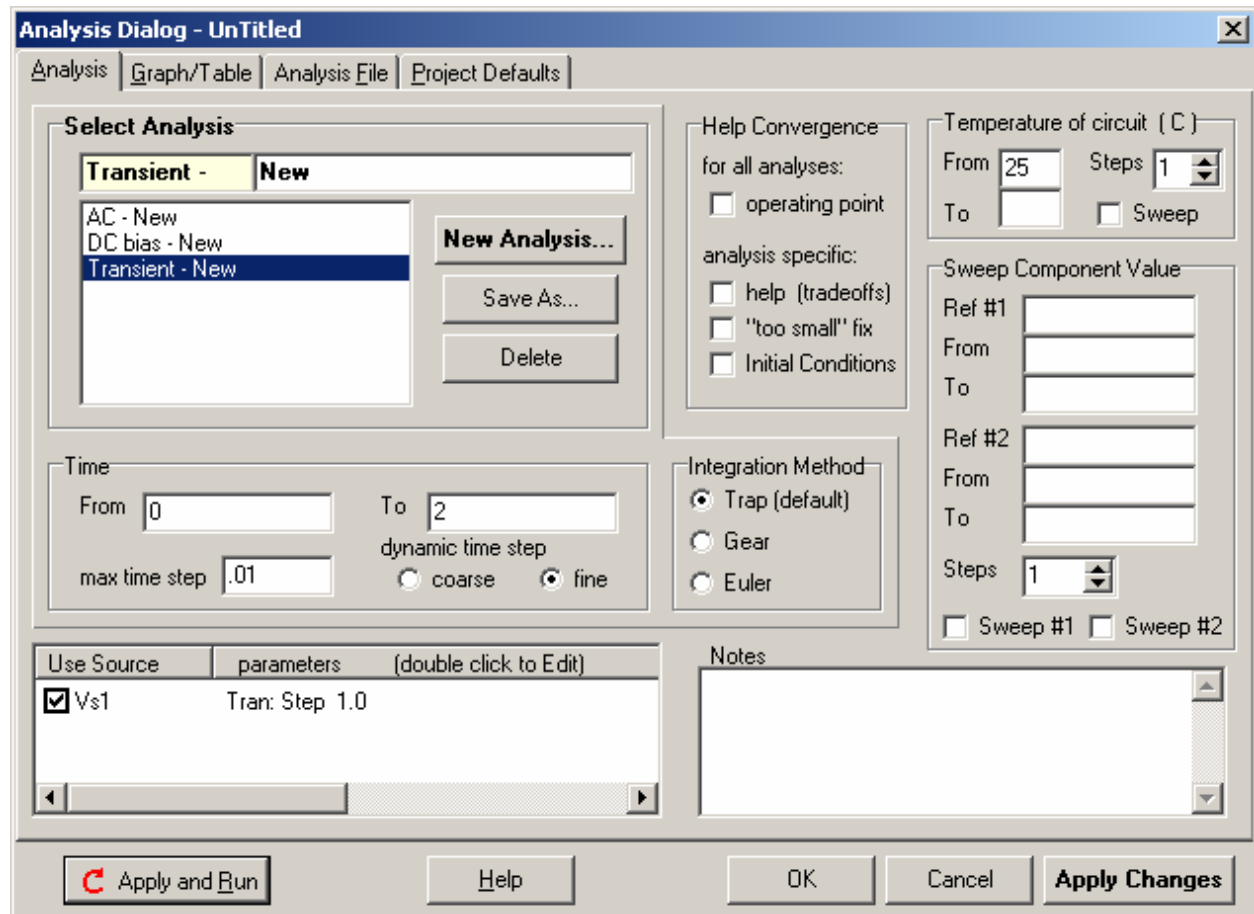
This is to create a wave. There are all types of waves you can choose from and different properties for each type of wave. The properties are pretty self-explanatory.

Analyzing

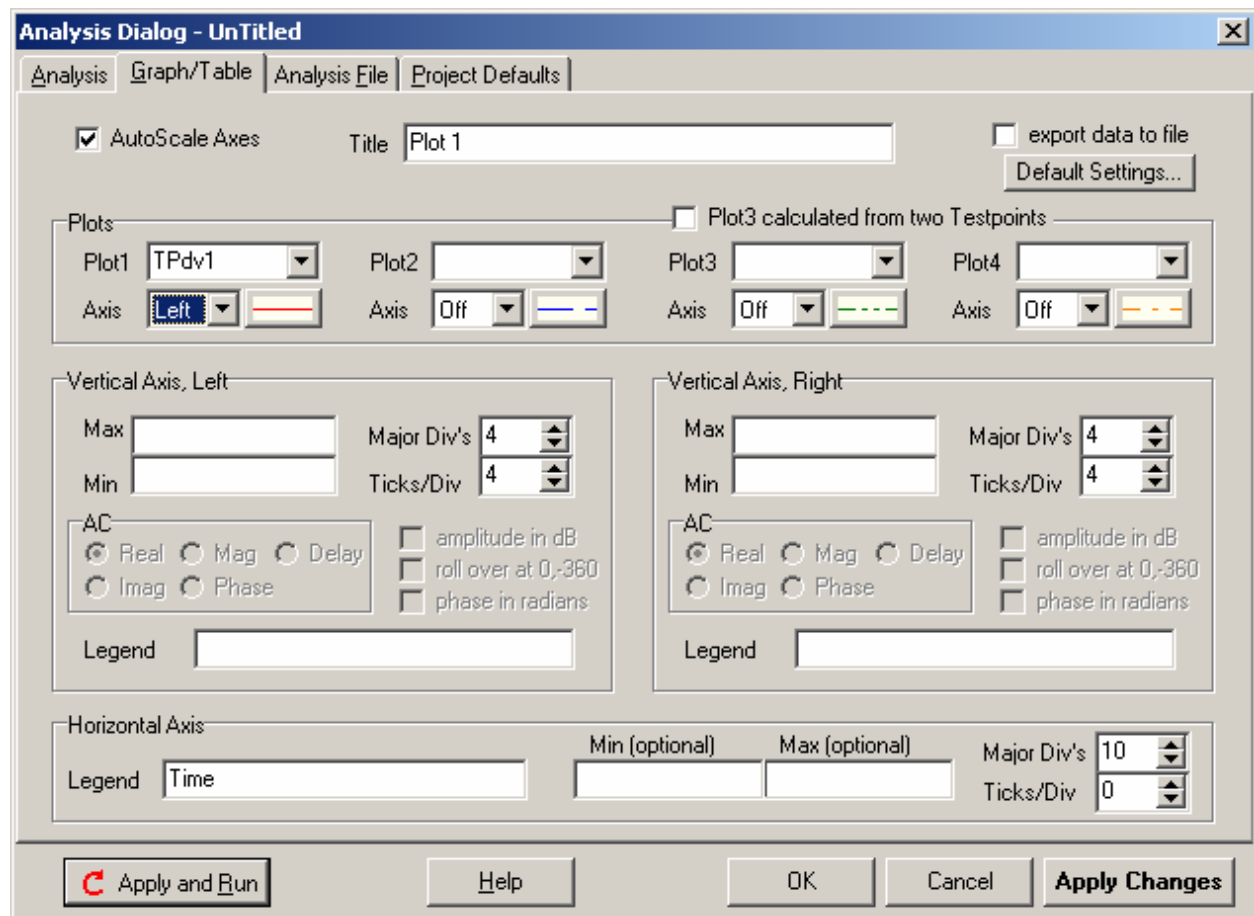
To Analyze your circuit, click on Analyze=>Select/Edit [Analyze Dialogue]...



On the Right side is Select Analysis. You can choose An AC, DC or Transient (time). If you choose DC you will not be able to make a graph. But if you choose AC, you can make a graph. Here is a transient set up.



Then you can click the Graph/Table analysis and setup your graph as so:



Then click on Apply and Run and your set up should provide you with a graph of the setup.

That should be the basics. You can play to find out the rest.