TUTORIAL #2:

2-0. EXECUTION OF A CIRCUIT SCHEMATIC

GETTING RESTARTED:

Step #1: Invoke pspice by double-click of the left mouse button on the ‘design manager’ icon as was done for startup under tutorial #1. Then double-click left on the schematics icon for which the blank pspice schematics screen will appear.

Step #2: Click (single click left) on the ‘File’ > ‘Open’ button sequence, for which you should see:
unless you are in some other directory of windows besides your ‘Projects’ directory, in which case you will have to wander through the directory tree and find the circuit schematic ‘pwr_xfer’ that you created once upon a time.

Of course if you have gone directly from tutorial #1 to this tutorial (tutorial #2), you are already there, and steps #1 and #2 are unnecessary.

You should end up with the following screen:
and it is now time to let pspice enjoy a few activities.

First we will label one of the nodes as an output node. In order to do so move the cursor over the wire above R2 and double-click with the left mouse button, for which a ‘label’ window will appear:

![Image of MicroSim Schematics with a node labeled Vout]

Insert the label ‘Vout’ and close this window with an ‘OK’

![Image of Set Attribute Value dialog with Vout entered]

which will leave you with the output node (wire) labeled as Vout. Pspice will recognize this label and include in its output menu:
Now we will put the pspice mathematics analysis to work.

For example: To do an analysis for which we sweep V1 over a range of values, we first toggle the ‘Analysis’ button at the top of the Schematics screen. An intermediate menu will appear that is sufficiently unattached so that it could not be captured for display by this document. But if you select ‘Setup’ (syntax: **Analysis > Setup**) under this document then the following menu should appear:
The boxes can be toggled to invoke whatever type of analysis we might want. In this case toggle the enable box of the Bias Point Detail (to turn it off) and toggle the DC Sweep enable box to turn it on:

Now tap the DC Sweep button (single click left) to obtain the following screen:

A few of its buttons have already been pushed, i.e Swept Var Type has been set as ‘Voltage Source’, and Sweep Type has been set to be ‘Linear’ as represented above.

Although we can push other buttons, these will suffice for now.
In the blanks shown fill in the following:

- **Name:** V1
- **Start Value:** 0
- **End Value:** 10
- **Increment:** 0.01

You have told pspice to sweep the part called V1 from 0 to 10 (volts) in increments of 0.01. Note that this will correspond to 1000 pts that pspice will calculate.

End these analysis specifications by an ‘OK’ on the DC Sweep screen and a ‘Close’ on the analysis screen. Pspice now knows what to do with your circuit.

And you tell pspice to do a ‘go-ahead’ by the hotkey command F-11, which means ‘Run Simulation’ or you can click (single click left) on the icon that looks like a couple of signal traces overlaid by a resistance.

Pspice will zip through its calculations and pop up two windows, one which looks like
and one that looks like

The first window is an ‘error message’ window. If you happen to have errors that cannot be resolved by pspice it will list them and you will have to try and find out what you have neglected. The most common error is that you have forgotten to include a GND. Since the pspice mathematics is based on a modified nodal analysis process, it MUST have one node identified as the reference (GND) node.
The second screen is the ‘Probe’ screen, analogous to an oscilloscope screen. It will show traces of the output vs whatever input may be desired. The default input is the one that is swept, as you might expect. Use your left mouse (single click left) to tag the **Trace > Add** sequence of buttons and you will see the screen

![Add Traces window](image)

Notice that one of these traces is \( V(\text{Vout}) \). That is the one of interest for the moment, even though we could call up any one of the others. Click (single click left) on \( V(\text{Vout}) \) and you will see that it will appear as the trace of choice under the ‘Trace Expression’ line at the bottom of the ‘Add Traces’ window.
If you click on ‘OK’, the Probe display window will then re-appear and you will see:
Big surprise. It’s a straight line. You can toggle the ‘cursor’ icon (looks like a crosshairs) and a cursor will appear. If you drag the cursor center up and down the trace by use of the mouse, you can place it to the approximate coordinates shown, from which you can identify the slope as rise over run.

Big surprise: it is 0.9, just what was specified by the resistance pair as a voltage-divider.

You have now completed tutorial #2, for which you executed an analysis command, told pspice to ‘go ahead’, and displayed the computed results as an output trace on the ‘Probe’ display screen.

Of course there are a few other options, and we can proceed (tutorial #3). Or you can save what you have and return to it later. Save is done automatically as you close out the windows, or if there is a change that has not been applied, pspice will alert you that you need to save, and will await your concurrence.