



# PSPICE TUTORIAL WORKSHOP

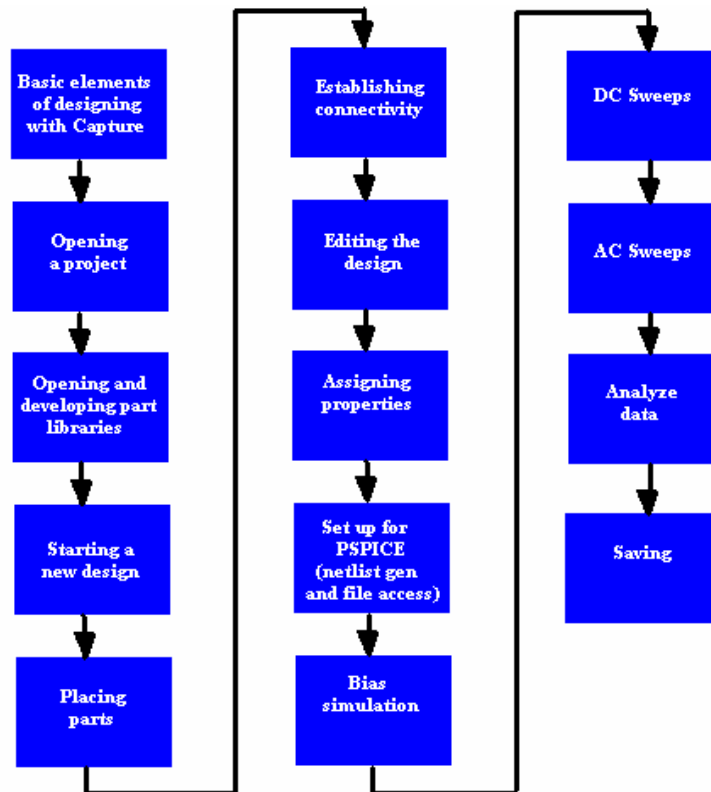
Written by: Dr. Hugh Grinolds and Michael Neuberg  
Colorado State University IEEE Student Branch  
March 5, 2007

This tutorial can be found at <http://www.engr.colostate.edu/ieee/>

## OBJECTIVES:

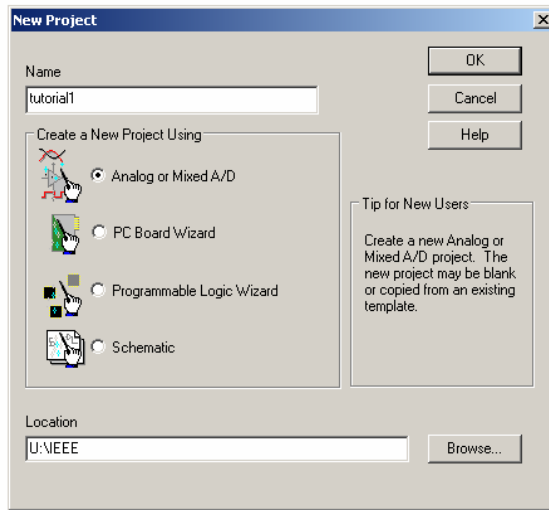
- Become familiar with using CSU's current version of PSPICE
- Basic set up to design and analyze circuit
- Understand how to plot data using a simulation
- Editing and modifying parts

Here is a workflow diagram of available steps used to setup and run a PSPICE circuit. We will only cover the basics in this tutorial.



## Let's Begin:

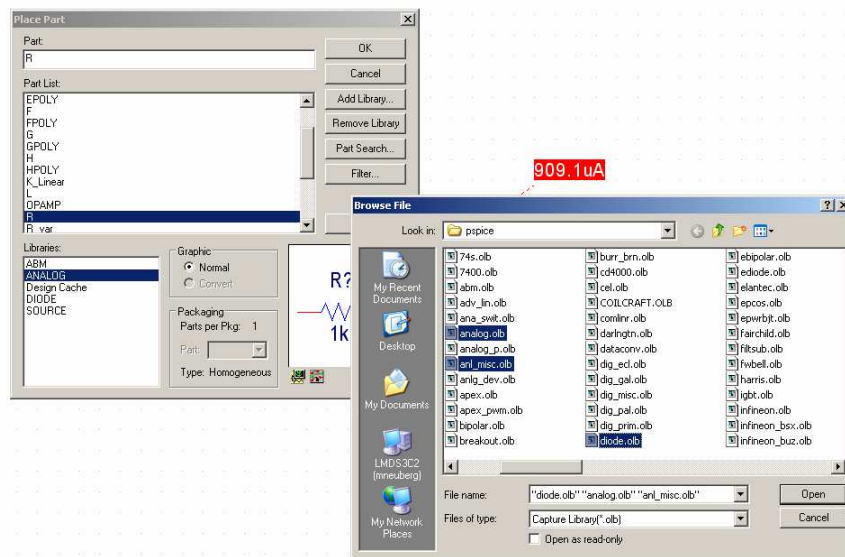
- Run OrCAD 15.7 Capture.
- Use the OrCAD Unison Ultra version.
- Once you launch Capture, you will want to begin a NEW project. Make sure it is a project you start and make sure you select Analog or Mixed A/D.



- Create a blank project.

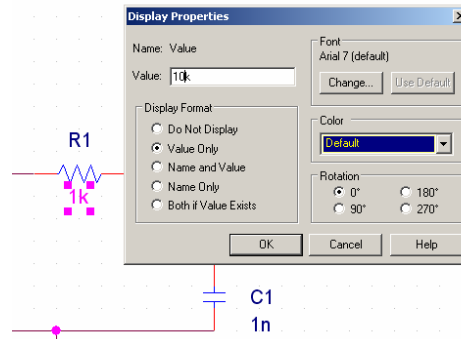
## Start a schematic:

- Under the Place tab, select Part.

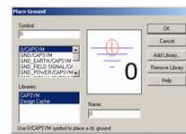


- You will need to add specific libraries depending on your design to use the correct parts.

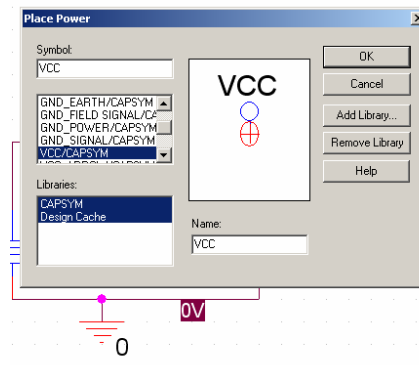
- To find the libraries go to Add Library C:\OrCAD\OrCAD\_15.7\tools\PSpice\Library.
  - Some of the common libraries are: analog.olb, opamp.olb, source.olb, special.olb-others will vary with specific needs.
  - **\*\*Important note:** If using the virtual lab you may not have access to all of the libraries.
- Begin to build your circuit using the parts from the libraries.
  - To change values or names of parts double click on the value or name of the part.



- Under the Place tab, select Ground.
  - Make sure you have a zero ground (i.e. 0/CAPSYM).

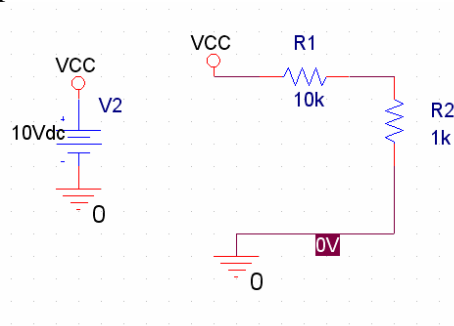


- To power up the circuit make sure you have a power source from the SOURCE library.
- To place the same voltage at multiple locations using the VCC bubble
  - Under the Place tab, select Power.
  - Go to VCC/CAPSYM.



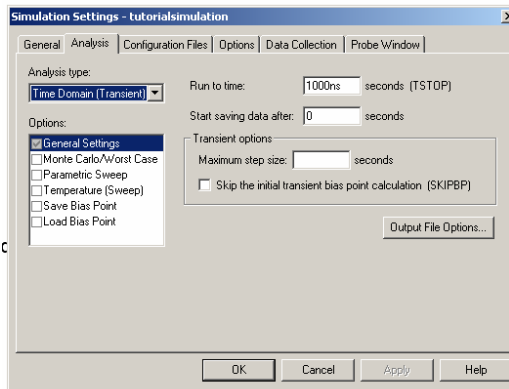
- Separate from the circuit connect to a power supply, with your desired voltage, which is connected to a ground.

- Then every place that you place another VCC “bubble” will have that voltage at that point.



**Run a simulation:**

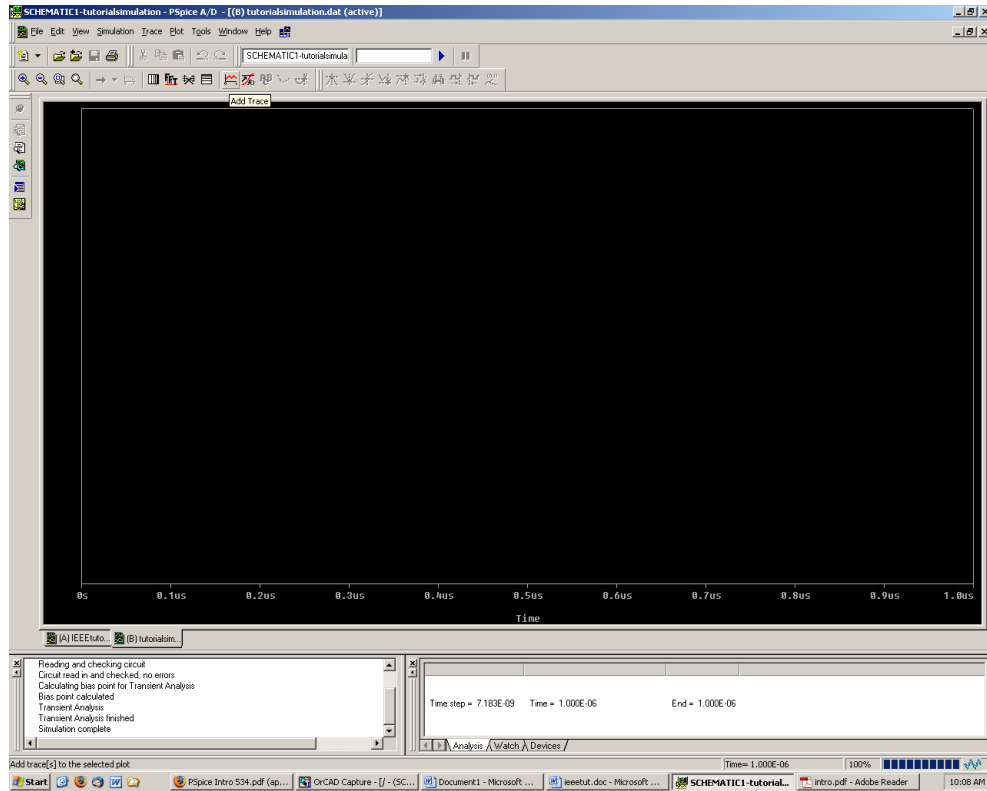
- Under the Pspice tab, select New Simulation Profile.
  - Name your simulation (don’t inherit one unless you know what you are doing).
  - Now the edit simulation window will pop up.
  - First, go to the Configuration Files tab.
    - Check the libraries to see if nom.lib is included.
    - If not you can browse and find it at C:\OrCAD\OrCAD\_15.7\tools\PSpice\Library.
    - If you include specific devices you will need to add these libraries also.
    - Add any libraries need to Design.
  - Second, go to the Analysis tab.



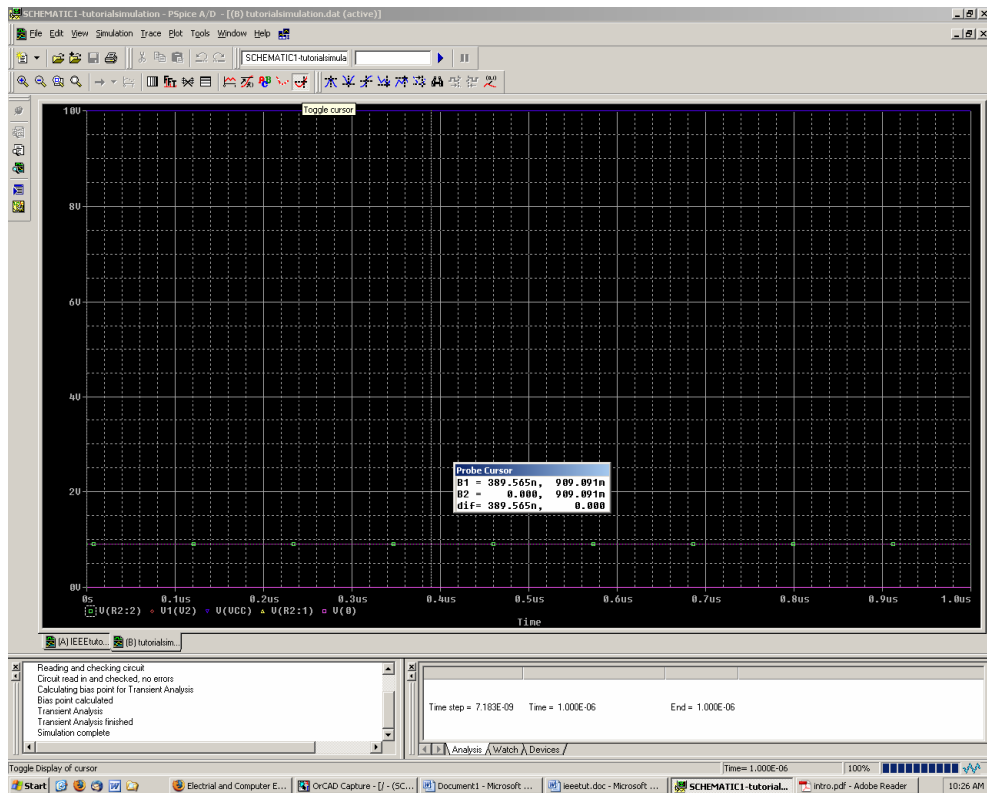
- For a general DC and AC analysis make sure you are under the Time Domain Transit type.
    - For a DC sweep select this type and enter desired values.
    - For a AC sweep (i.e. frequency) select this type and enter desired values.
    - Select any other options desired then press OK.
  - Now Run the simulation.
    - If errors occur you will need to try and find the problem.

**Plotting results:**

- Depending on your objective you can plot different values by selecting the Add Trace button.
- If you go to the Probe Window tab under the Simulation Settings and check the box 'Last Plot' will have all traces and functions remain for multiple simulations.



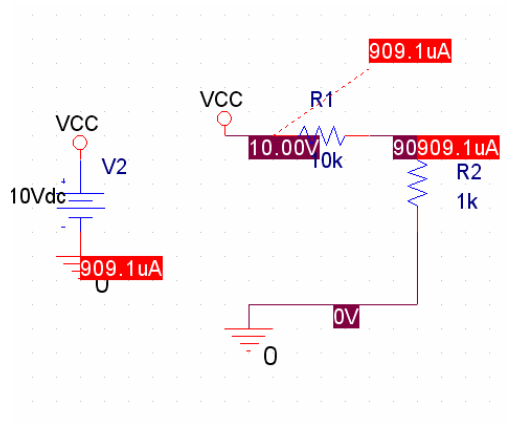
- The values given are in relation to the names and labels or your circuit.
- Select your desired values, scales, relationships, etc.
- To find values on the graph select the Toggle Cursor button.



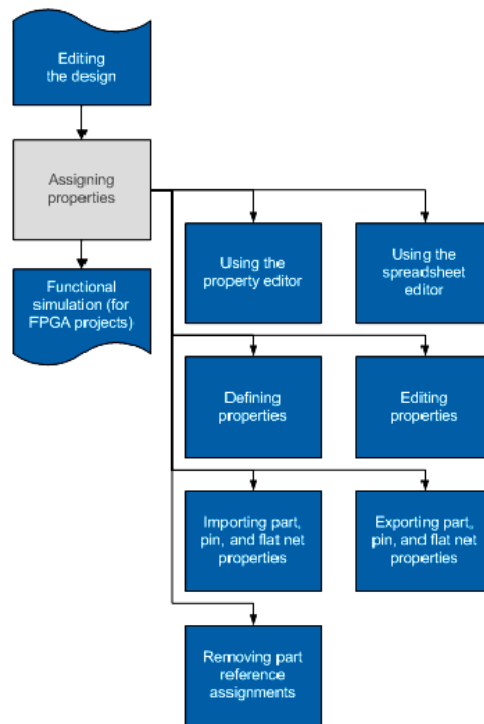
- This gives the ability to select different curves and move along the curve to locate points.

### Analyze Circuit:

- On your circuit you can see the values at given points by selecting the icons at the top of the screen.
- If you wish to plot a certain point you can use the probes to measure values and then select these values to plot under the Add Trace.
- You may need to rerun the simulation every time you change the circuit to see the results.



# Advanced PSPICE Skills



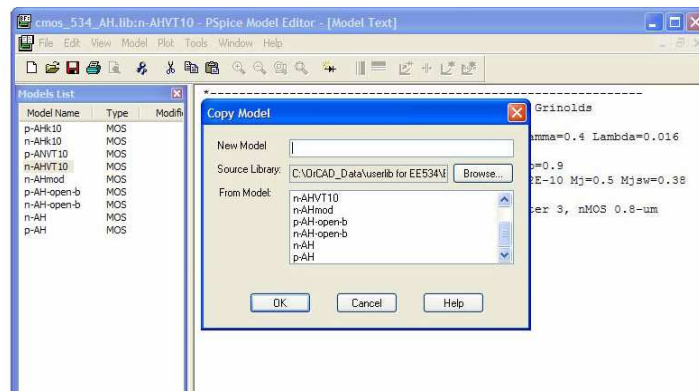
## Editing part properties:

- Right clicking on any portion of the part will bring up that portion or the entire part to edit properties. Doing so on an MOSFET (e.g. N-GM) will display a variety of parameters.
- Scroll to find the parameter 'M'. This parameter multiplies the W of the device. Highlight the column, select display and choose 'Name and Value' then OK. Click 'Apply' and close the window.
- You will see 'M=' displayed near the part. You can select this to move it if desired. Right-clicking on 'M' now will bring a window to enter a value. The value becomes the integer multiplier on W/L of that device.
- With devices, you can bring out both W and L to edit their values. Doing so will be necessary in the future as we explore cascodetopologies, current sources and transfer switches.
- Another useful part editing is on sources for renaming them to more pertinent names (e.g. from V1 to VGS).

## **Modifying and Adding Models:**

- In the OrCADsuite, a pSPICEModel editor is available and is the appropriate vehicle for editing the \*.lib files.
- Find the program under pSPICEAccessories under the OrCAD15.7 program menu
- Use the ‘file open’ to access the .lib file you wish to edit.
- The library file is referenced by the schematic file (\*.olbfile in Capture). Thus, after editing, there is an Export of the \*.lib to an \*.olb.
- Make sure that when done, both files can be accessed by the Capture and pSPICEprograms (Capture calls up pSPICEfor simulations). The best way is to save both away in a Library Directory as mentioned when importing the initial libraries

## **Using the Model Editor:**



- This shows a library open and ‘Copying’ an existing model to create another.
- Access the copy function from the Model menu.
- Choose a name and select the model for the starting point.
- The chosen name will be reflected in the model text. Make the appropriate edits. Save the model.
- Note the model is added to the list in the library.

## **Exporting:**

- When done, go to the file menu to ‘Export’ to Capture library.
- If this runs with any errors, you will need to go back and edit. Most likely, you have a syntax error.
- Another function exists to relate this model to a stored set of schematics and pinouts. The model wizard provides this function. In most cases, you

will not need to do this. Go ahead and run it though to see the action but don't change any of the mappings unless you wish to do so.

## **FAQ's:**

How can I get PSPICE at home?

- You can download PSPICE from the following site: [www.electronicslab.com/downloads/schematic/013/](http://www.electronicslab.com/downloads/schematic/013/) Go to the bottom of the web page for the download link.

I get an error that says nom.lib is missing when I try to run a simulation?

- You will need to add this under the Configuration Files tab, and then browse to find the library here -  
C:\OrCAD\OrCAD\_15.7\tools\PSpice\Library -then you will need to add this to the design.

I don't see a PSPICE tab to run a simulation?

- If there is no PSPICE tab then you have probably set up a schematic. You will need to copy your circuit then create a new project using Analog or Mixed A/D.

Where are the basic DC and AC power sources, resistor, capacitor, and opamps?

- The power sources should be under the source library. The resistors and capacitors should be under the analog library. The opamps should be under the opamp library. When placing a part you can also select Part Search to help you out.

I want to set up a table of various component properties so I can edit and vary them or simulate response for different values. How can I do it?

- There are multiple steps involved to construct the capability. Overall, you need to label each value and then enter these into a parameter table. An example of doing so with a resistor is:
  - Right click on the resistor you wish to vary. Label with a variable name enclosed in {variablename}.
  - Place the PARAM part. This is a part in the SPECIAL.olb library so you will need to load that library (and make sure you reference it in the configuration or library files for pSPICE simulation). You can find the library where the SOURCE and ANALOG libraries are located under the OrCAD/OrCAD\_15.7\tools\capture\library\pspice directory.
  - Edit the properties of the PARAM part. Here, you create a column with your variable name. You should display the name and value. You can enter your desired value.
  - Now you can use the name of this variable in a parameter sweep (without the {}).