***PSpice* Basics**

If you have not used the ***PSpice*** software package before, it is highly recommended that you read this document before starting to run any laboratory exercises for EE4133: Instrumentation Design. In addition this is a good resource for EE3122: Electronics II.

The ***PSpice*** package comprises three main programs that are linked together through a graphical user interface (GUI). **Capture** is a graphical pre-processor that is used to enter a schematic and perform the necessary conversion into a ***PSpice*** netlist. ***PSpice*** is the main engine that is used to perform the actual circuit analysis that originated in Berkeley, California and was initially written in Fortran. ***Probe*** is a graphical post-processor that is used to display the results of the simulation and to perform simple analysis functions.

PSpice directory set up *(recommended)*[1]

Before starting to create schematics and use ***PSpice***, it is recommended that the following directories should be created on your workspace, whether this is on a home machine or Novell:

…\EE4133	*a root directory for Course*
…\EE4133\library	*location of specific library files*
…\EE4133\Lab1	*location of the first lab*
…\EE4133\Lab2	*location of the second lab*
…….(continue as needed)	

Two custom library files (***EE4133.olb*** *and* ***EE4133.lib***) should be obtained from the course WEB site under the "***Additional Materials***" page. These libraries hold parts specific to EE4133 and should be copied to the ***…\EE4133\library*** directory in your workspace. The ****.olb*** file hold the graphic outlines for the components and is used by **Capture**. The ****.lib*** file hold the ***PSpice*** model statements for the same components. Once these library files have been placed in the appropriate directory, the process of schematic entry can be initiated by running **Capture**. This opens a **Capture** window with a small **Session Log** window in the upper left-hand corner.

[1] Please note in this document the course number is assumed to be that of Instrumentation Design. If you are reading this to assist you with Electronics II, please replace all references to EE4133 with EE3122, the course number for Electronics II.

Creating a project

The first step in any simulation is to create up a project in the particular directory chosen for the work. This forms an integrated design environment that will be used to group various items such as libraries, schematics, simulation profiles, output files, parts lists and PCB layouts associated with a particular design.

A project is created from the **Capture** main menu from by the command sequence, **File**, **New**, **Project:**

1st Dialog: **New Project**

Name: ***BridgeAmplifier*** *or suitable title*
Create a New Project Using: ***Analog or Mixed A/D***
Location: …***EE4133\Lab1***…*or appropriate location*
Click *OK*

2nd Dialog: **Create *PSpice* Project**

Create a blank project…then OK

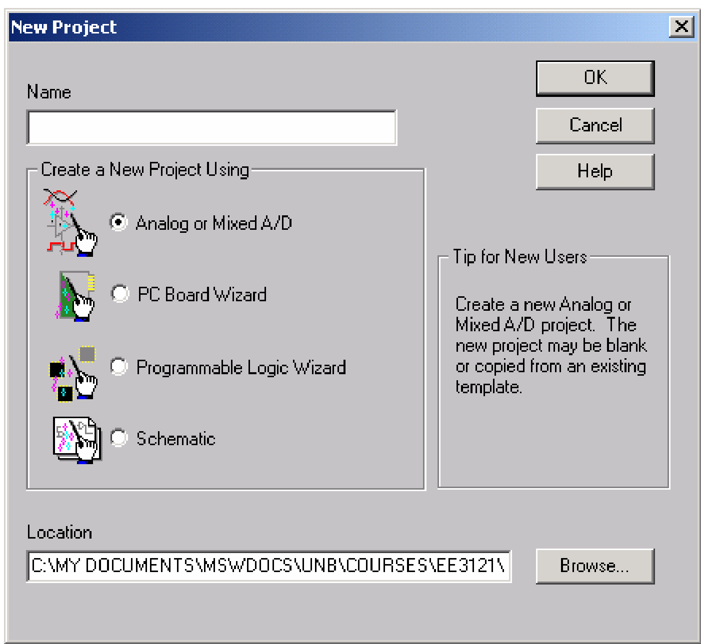

Figure 1: "New Project" dialog

This will result in two new windows being displayed; namely, a project window (***.opj***) and a schematic window ***SCHEMATIC1 Page 1***. The project window, with the same name as entered in the "*New Project*" dialog, shows the tree structure of the project files while the schematic window is where the 1st circuit schematic will be drawn. As more items are added to the project, such as additional schematic sheets, libraries and simulation profiles, the project tree structure will grown to show this progression in the design.

Adding a custom title block

The second step is to add a custom title block in the lower right hand corner of the schematic sheet. This will be used hold such information as the name and owner of the design along with any subsequent revisions. As ***PSpice*** puts a default title block in this location this should first be deleted. Select the existing title block by clicking on outline and then delete it by using the **Del** key.

Next from the **Capture** menu, select the command sequence **Place**, **Title Block:**

1st Dialog: **Place Title Block**

Add library

2nd Dialog: **Browse file**

Select *\EE4133\library\EE4133.olb* …then OPEN

1st Dialog: **Place Title Block**

select *EE4133TitleBlock* then OK
drag block to lower right hand corner of the schematic sheet
left click to anchor the block
right click and select *End Mode*

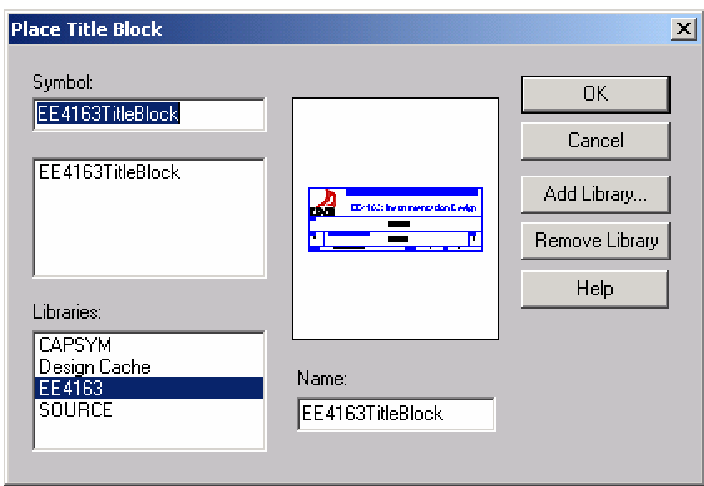

Figure 2: "Place Title Block" dialog

The UNB title block should now be located in the lower right-hand corner of schematic sheet so that the borders of the sheet and the border of the title block coincide. The various fields within the title block can be edited at any time by left clicking on the field. This action invokes a floating menu, from which an "*Edit Properties…*" dialog box can be selected by a right click. This in turn allows editing of the selected field

Adding components to the schematic sheet

This is quite intuitive and can be done quite rapidly. For example, to place a resistor to the schematic sheet, from the **Capture** menu, by the command sequence **Place**, **Part:**

Dialog: **Place Part**

Select library – ANALOG
Select part – R…then OK
left click to anchor the block
right click and select *End Mode*

This is basically the same procedure used for placing the title block on the schematic sheet. Note that before left clicking to anchor the block, ***Ctrl-R*** will allow you to rotate the part before adding it to the schematic.

For parts that are not built-in to the ***PSpice*** project, eg. TL081 op amp, it will be necessary to add the EE4133 library to the project. This is done in the exact same way as the custom title block was added. Once the library has been installed in this way, subsequent parts can be selected just as in any other library.

Changing the component labels & values

When components are first placed on the schematic sheet they have default values associated with them. For example placing a resistor results in a 1kΩ resistor being added to the schematic sheet. **Capture** automatically names the resistor, R1, if this is the first resistor added. As more resistors are added, the name, or ***part reference*** as it is more correctly termed, is automatically incremented.

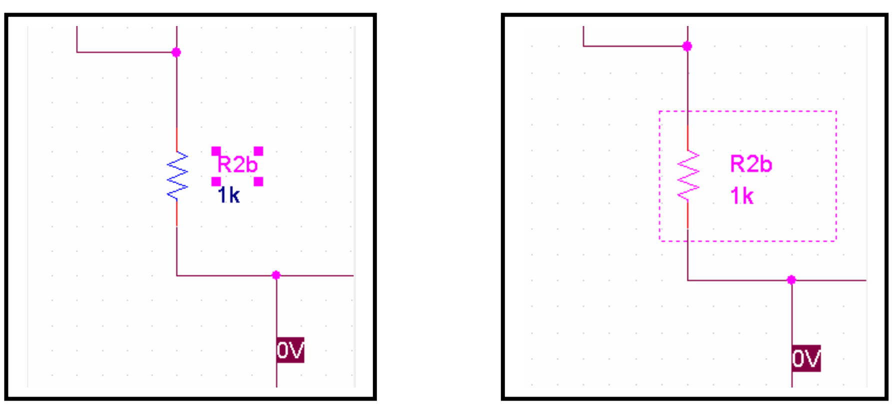

Figure 3: Part or the complete device can be selected

The name and part references of the components can be changed in two ways. Firstly if only the value requires changing select only the text value by placing the cursor on the name and then click left. Once the name is selected, the right button can be used to invoke a floating menu. Select "*Properties…*" from this menu and edit the value in the associated dialog box. If only the part reference requires changing, the same general procedure can be followed, left clicking only on the reference.

If both the name and part reference are to be changed, an alternate approach is to select the component, by left clicking on the body of the component. Once the component is selected, the right button can be used to invoke a floating menu, from which "*Edit properties…*" should be selected. This brings up a database table from which the "*Value*" and "*Part Reference*" then can be edited in the same operation.

Note that the part reference does not have to have a numerical part. Value references include Rload, Rsource, Rc, RB etc. In this way when inspecting the results of the simulation the selection of the particular variables is made more intuitive.

Connecting the components with wires

Once all the components have been placed on the schematic sheet, they must be connected together using the appropriate wiring. This is accomplished from the **Capture** menu, by the command sequence **Place**, **Wire**. The normal arrow cursor is replaced by a cross-hair; this indicates the wiring mode.

To join two components, the cross-hair should be placed over the terminal where the wire is to start. Anchoring the wire to the first component is performed by a single left click. Subsequent movement of the mouse will now drag the free end of the wire. As the free end of the wire moves over component terminals a small red connection dot appears. If the wire is to be connected at this spot, right clicking will invoke a floating menu with the option of "*End Wire*". Selecting this item will terminate the wire, but not the wiring mode. Note that wires can cross over each other without making a connection.

After completing the connection between two components, the cursor remains a cross-hair and can be used to add more wiring. Once all wiring is complete, to return the cursor to its normal function, right click and select "*End Wire*" one final time.

Creating a simulation profile

A simulation profile specifies what type of analysis is to be conducted on the design. In ***PSpice*** this can range from a simple operating point analysis, to a full Monte-Carlo statistical analysis of a filter frequency response dependency on component tolerances. As it is usual that more than one stimulation type will be used with a particular design, ***PSpice*** allows the creation of multiple simulation profiles within a single project. For example when evaluating a filter design, both the steady state (Bode) performance and the transient response would have different profiles.

A simulation profile is created from the **Capture** menu, by the command sequence ***PSpice***, **New Simulation Profile:**

Dialog: **New Simulation**

Name – supply appropriate name
Create

This invokes a tabbed dialog box in which all the simulation parameters can be specified.

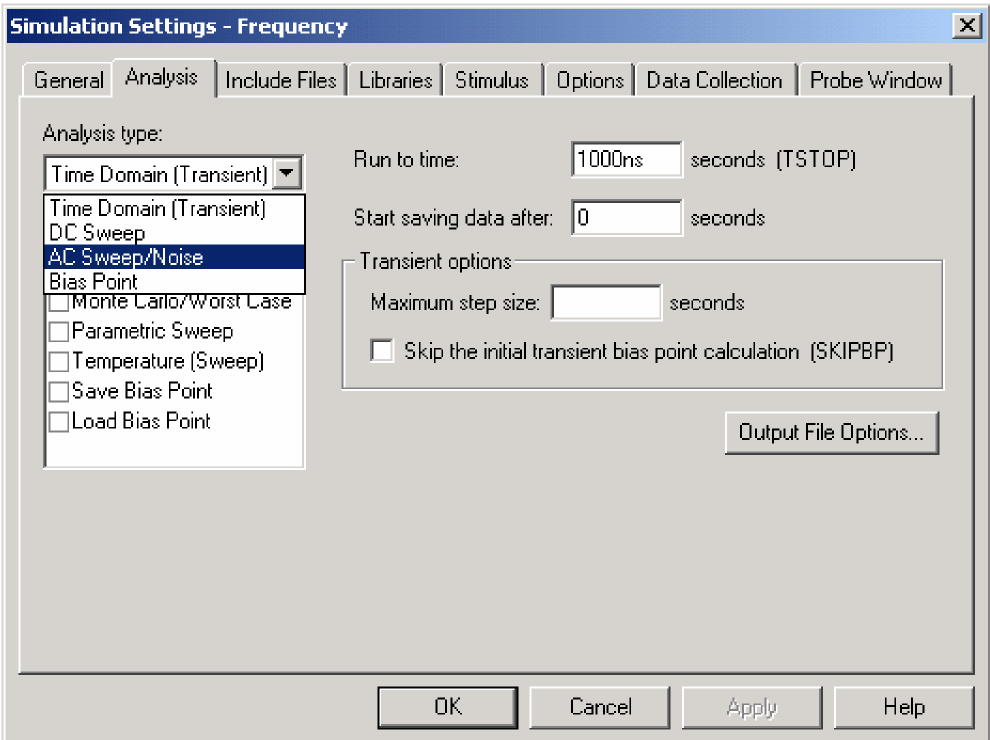

Figure 4: Simulation Settings" dialog for profile named Frequency

As the power of the ***PSpice*** engine is quite formidable, it is impossible to cover all the instances of stimulation profiles in this introductory text. However, some of the more often used analyses are listed here, with a typical application.

The main analysis type can be one of four types, namely:

Time Domain (Transient):
…produces a time simulation. For example, this type of analysis would be used to investigate the step response of a low pass filter.

DC Sweep:
…a DC voltage or global parameter can be varied and the effect recorded. For example, this type of analysis would be used to investigate the voltage transfer characteristic (VTC) of an inverter.

AC Sweep/noise:
…an AC voltage source is frequency swept and the response determined. For example, this type of analysis would be used to investigate the frequency response of an amplifier or filter.

Bias point:
…a DC analysis of the circuit is performed. For example, this type of analysis would be used to investigate the operating condition of a MOSFET circuit.

The actual appearance of the "*Stimulation Settings*" dialog changes depending on the analysis type selected. For example in the Time Domain analysis additional information is required as to the duration of the simulation, however in the AC Sweep/noise analysis this information is meaningless. In this case the frequency range over which to perform the analysis is a more useful parameter.

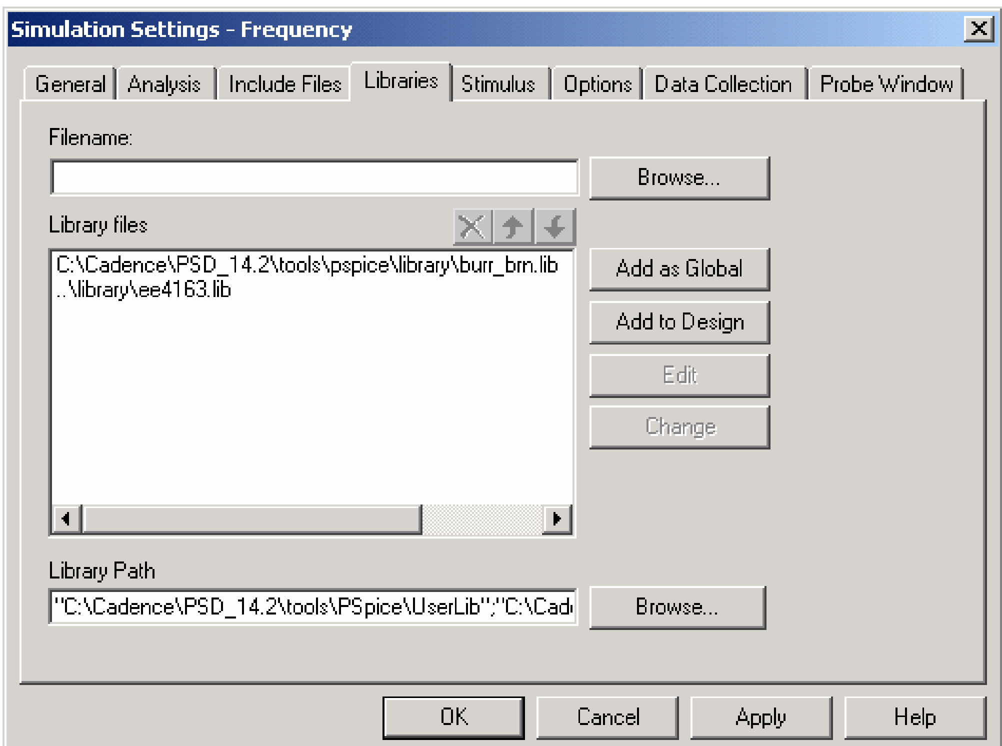

Figure 5: "Simulation Settings" dialog for showing libraries

One of the tabs on this dialog box is labeled "*Libraries*". This is used to specify the library files that include the model statements that are used to define the operation of most of the circuit components. Today most manufacturers supply model libraries for use with ***PSpice***, and the full version of the software comes complete with an array of libraries.

The student version of ***PSpice*** however only has a few basic components. Consequently, for this course the instructor has created a small library that contains all the required components for the laboratories and assignments in this course. ***EE4133.lib*** is an ASCII file that contains the model statements. Consequently, prior to running any simulation, this library should be installed by using the Filename "*Browse…*" button, followed by the "*Add to Design*" button.

This completes this brief introduction to *PSpice*.

For further information see the links to the extensive ***PSpice*** documentation that have been included in the WEB sites:

http://www.ece.unb.ca/Courses/EE4133/DFL

and

http://www.ece.unb.ca/Courses/EE3122/DFL