Getting started with PSpice (Version 9.1)

1) Start the PSpice Schematics program-
   - Campus computer: After logging on, click the **Start** button (lower left hand corner), then successively select **All Programs**, **DEPT**, **ECE**, **PSpice Student** to get to the PSpice menu. Click on **Schematics** (bottom of menu).
   - Personal computer: Click **Schematics** shortcut. [**Note:** When installing PSpice 9.1 on computer, when prompted, install **both** the ‘Capture’ and ‘Schematics’ options.]

2) You should now see all or part of the schematic entry screen/worksheet (white with little blue dots). This worksheet represents/symbolizes the large sheet of paper (blueprint) on which draftsmen used for draw electrical circuits.

   a) To adjust how much of the worksheet you are viewing, you can use the options under the **View** menu as desired **or** you can use one of the icons (near top just below the drop-down menus and left of center, look like little magnifying glasses). For the electrical circuits done in EE 220, you should be able to fit them in a quarter of the page (lower/upper lefthand corners are nice).

   b) To save a new project, successively select **File → Save As…** to bring up a little window called “Save As” which is a standard windows program box. Browse around to select/create the proper drive and directory to save your project, enter a filename and click **<Save>**. This will create a file named “filename.sch”.

3) You are now ready to build/draw the circuit. While it is possible to get and place circuit components several ways, the easiest way is to use the **“Get New Part”** icon (looks like an IC chip with a pair of binoculars on top).

   a) To get and place parts, click on the **“Get New Part”** icon. This brings up a little window called “Part Browser Basic”. [Note: By default all the necessary parts libraries should be available. If not, click on <Libraries> and select the needed library from the list.]

      - In the lower lefthand corner, there will be a list of parts. Scroll down the list and select the part you wish to place in your circuit and click **<Place>** or **<Place & Close>**. Hint: you can get to the part you want quicker by typing the first letter of its name in the “Part Name” box near the top of the window, e.g., “R” for resistor.
      - You will then have a cursor arrow with the symbol for that part attached displayed on schematic worksheet. Move cursor to desired location and click the left mouse button to place the part. This can be done as many times as needed/desired. PSpice will autoname parts as they are placed (e.g., R1, R2, …). When you are done with a part, press the **<Esc>** key on the keyboard or click the right mouse button. [Note: The **Edit** menu window also allows you to flip and/or rotate parts. Alternately, press **<Ctrl>-f** and/or **<Ctrl>-r**.]
      - Repeat as necessary to place all parts. [Note: Circuits **MUST** have a ground (e.g., “EGND” part) for PSpice to work. You are allowed to use multiple ground symbols on one circuit. PSpice treats them all as one node.]
      - If necessary, double click on the value of the part to bring up a “Set Attribute Value” window. Enter desired value for the part as well as the appropriate units (no space between value and unit). Standard MKS prefixes are allowed (e.g., m = milli-, k = kilo-, u = micro-).
      - If desired, double click on the name (reference designator) of the part to bring up an “Edit Reference Designator” window. Enter desired name for part (can’t give two parts same name).

b) To wire up the circuit, i.e., interconnect the parts you just placed on the worksheet, click on the **"Draw Wire"** icon (looks like a pencil drawing a thin line). To start a wire, place the cursor over the end node of one component and left click, move the cursor to the next wire node location, left click, and repeat as desired. The wire terminates when you press the **<Esc>** key on the keyboard, right click the mouse, or click on another node connected to a component. To finish with the wiring mode, press the **<Esc>** key again.

c) Sometimes it is necessary to change the electronic model of a circuit component. For example, you might want to tailor a transistor model to have particular β or V_{BE} values. To do this, click on the part to select it. Then, successively select **Edit → Model…** to bring up the "Edit Model" window. In this window, select the <**Edit Instance Model (text)…**> button to bring up the "Edit Model Text" window where you can change/edit the various parameters associated with the part (e.g., Bf = 255.9 can be change to 125 and/or Vje = 0.75 can be changed to 0.7 to change the β or V_{BE} values for the model). Click the **<OK>** button when done to save changes. Note: Not all circuit components have parameters that can be changed (e.g., simple ideal components like resistors).

4) If desired, text, arcs, ellipses, lines, and rectangles can be drawn on the worksheet using the appropriate icons (see icon bar on the left side of the screen), or choose the desired option under the **Draw** menu.

5) Re-save your project (lets PSpice know about changes in the circuit) by successively selecting **File → Save** on the top drop down menu, or press the "**Save"** icon (looks like a little red floppy disk).

6) You are now ready to simulate or run PSpice for your circuit. You can do so by clicking the **"Simulate"** icon (yellow square with data traces), pressing the <**F11**> key, or successively selecting **Analysis → Simulate**. A new program screen "filename- OrCAD PSpice A/D Demo" will now come up. If voltage and/or current markers were placed in the circuit, data traces related to them will be displayed near the top. In the lower lefthand corner will be a log of messages, check to see if there are any errors (they show up in **red**).

7) Without closing this window, go back to the **Schematics** program by clicking on it (e.g., MS command bar at bottom of screen). Voltage and current information can now be displayed by clicking on the $\boxed{V}$ and/or $\boxed{I}$ icons (near center top).

8) Changes to the circuit can be made as needed. Just remember to re-save the project before re-running the circuit simulation.

9) For printing, you can cut-n-paste your circuit simulation(s) into MS-Word. Or, select the parts of the worksheet you want (left click and drag with mouse) to print and successively select **File → Print** to get the print menu. Ensure that the option "Only Print Selected Areas" is selected before printing. Printing the entire worksheet on 8.5"x11" paper can make the circuit, labels, and components too small to read.

10) Close and/or save all program windows when done.