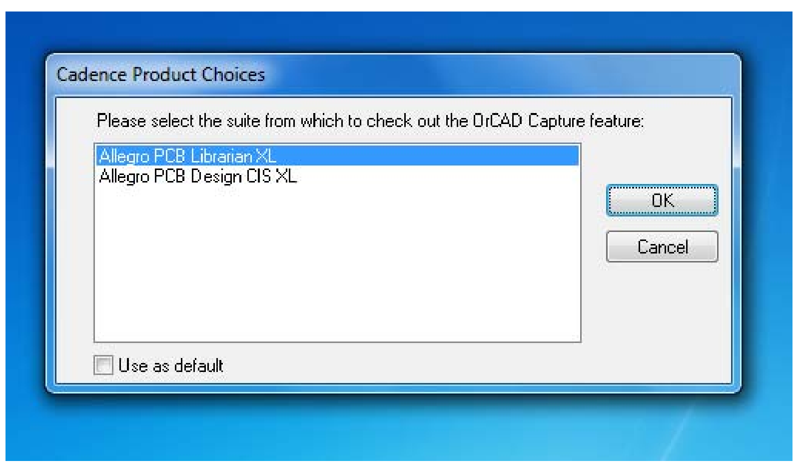
Getting Started with OrCAD Capture CIS v16.2

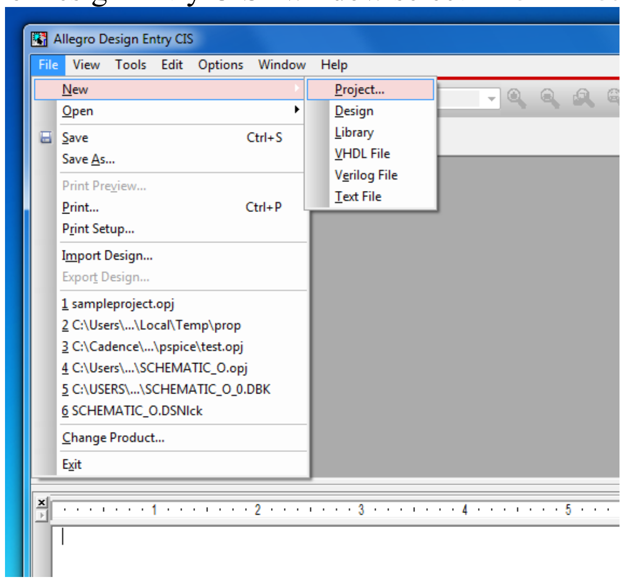
Opening a new circuit in PSPICE:

- **Start –> All Programs-> OrCAD 16.2 –> Orcad Capture CIS**

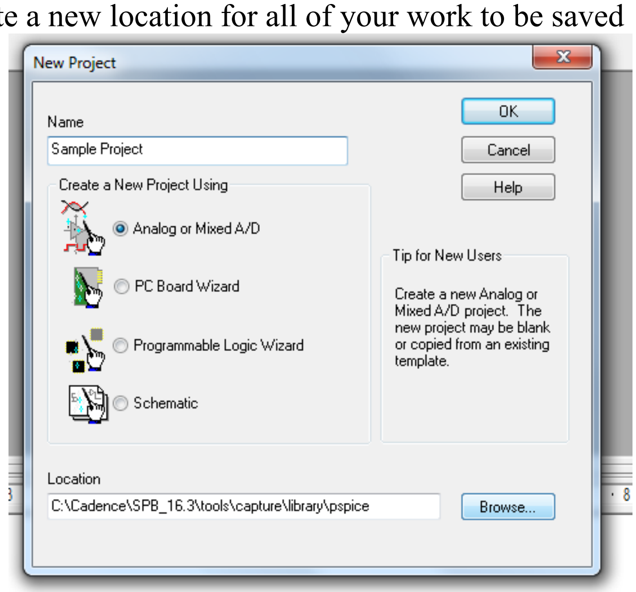
- In the “Cadence Product Choices” window you can the suite from which you want to check out the OrCAD Capture features. For now just leave it at the default choice and **click OK.**



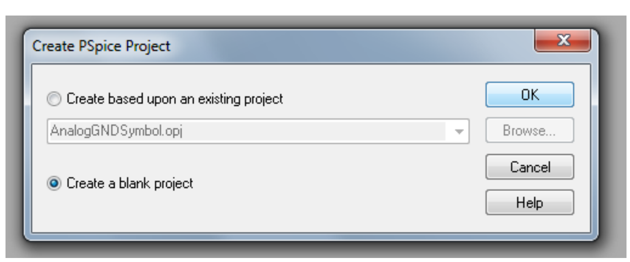
- In the “Allegro Design Entry CIS” window select **File -> New -> Project**



- In the “New Project” window:
  - Type a name for your new project
  - Select “**Analog or Mixed A/D**”
  - Create a new location for all of your work to be saved

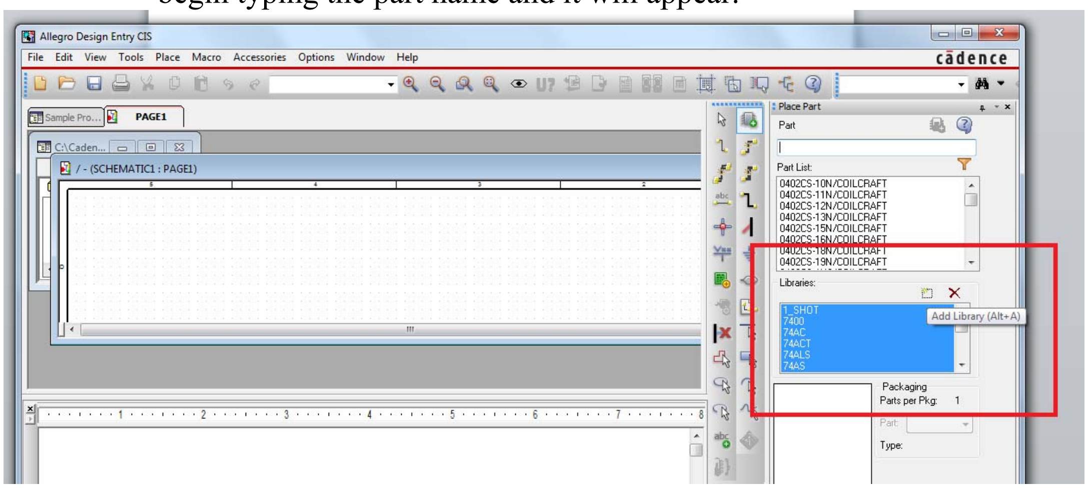


- In the “Create PSpice Project” window:
  - Select “Create a blank project” and **click OK**

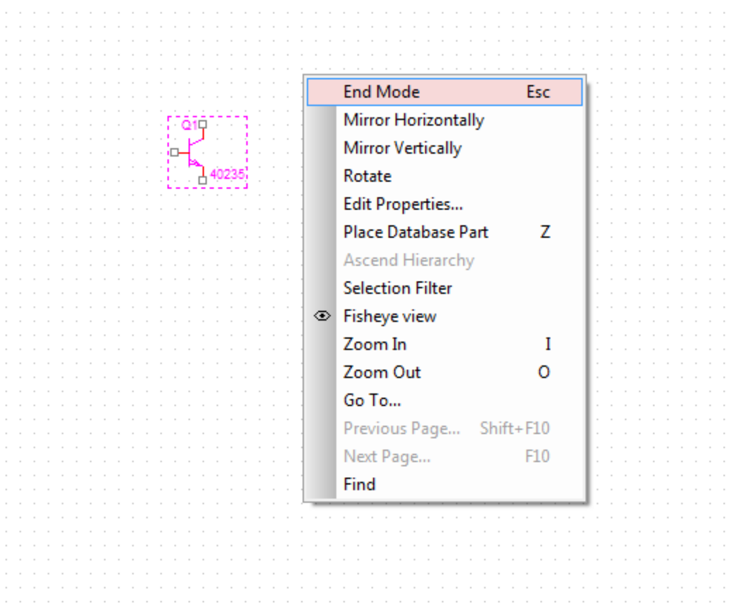


Building a Circuit:

- From the drop down menu select **Place**
- You will usually be placing **Parts, Wires,** or **Ground**
- If you select **Part**, then a window will appear for you to select a part.
  - When you place a part, you must have the library loaded into your library window. To do this, select **Add Library.** For EE 3221, you will most likely need the following libraries:
    - ANALOG – R,L,C
    - JBIPOLAR – Q2N3904 NPN and Q2N3906 PNP transistors
    - SOURCE – VSIN, VAC, VDC,VPULSE
    - PWRMOS – M2N6659 is an FET
    - OPAMP - LM741
  - Once the library is installed and selected, you can search for your part or begin typing the part name and it will appear.



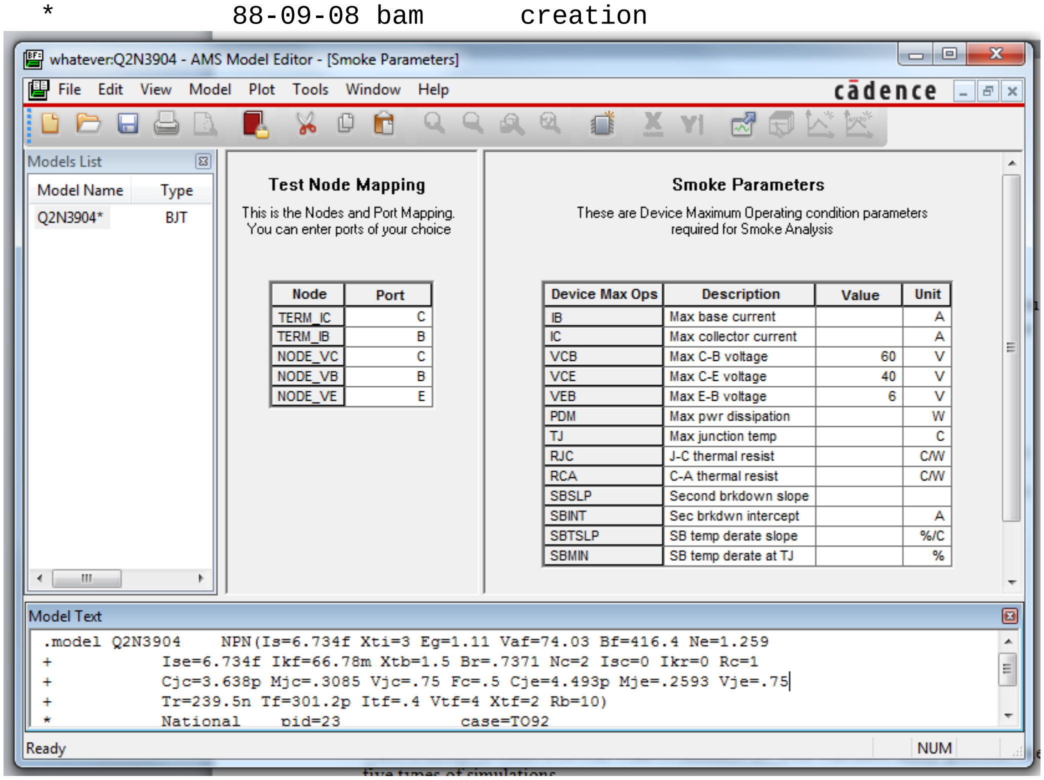
- Right click and select **End Mode** to finish placing parts.



Once the circuit is built, you will need to assign the correct values of parts and sources.

- To change ***component values***, simply double click on the current value and edit the **Value.** You can also change the part name this way.
- To change ***sources***, you also double click on the current value and then change the **Value.**
- When dealing with ***ground connections*** you must change the name of the ground to "0". To do this, double click on the ground symbol and change the name from GND_EARTH to "0". This will prevent Floating Node errors during simulation.
- To Change ***device model parameters***:
  - Select the part you want to edit by clicking it in the schematic window.
  - Select **Edit -> Pspice Model** from the menu bar at the top of the screen (alternatively, you can right click the part in the schematic window and select **Edit Pspice Model** from the drop-down menu.
  - A model editor window will appear (a BJT Q2N3904 in the figure). At the bottom will be the model text. Here you will see a list of many different parameters. Bf is the forward current gain β_o. Cjc is the reverse bias base-collect capacitance (C_μ). Cje is the forward bias base-emitter capacitance (C_π).

```
.model Q2N3904    NPN(Is=6.734f  Xti=3  Eg=1.11  Vaf=74.03  Bf=416.4
Ne=1.259
+           Ise=6.734f  Ikf=66.78m  Xtb=1.5  Br=.7371  Nc=2  Isc=0
Ikr=0 Rc=1
+           Cjc=3.638p   Mjc=.3085   Vjc=.75   Fc=.5   Cje=4.493p
Mje=.2593 Vje=.75
+           Tr=239.5n Tf=301.2p Itf=.4 Vtf=4 Xtf=2 Rb=10)
*           National    pid=23          case=TO92
*           88-09-08 bam     creation
```

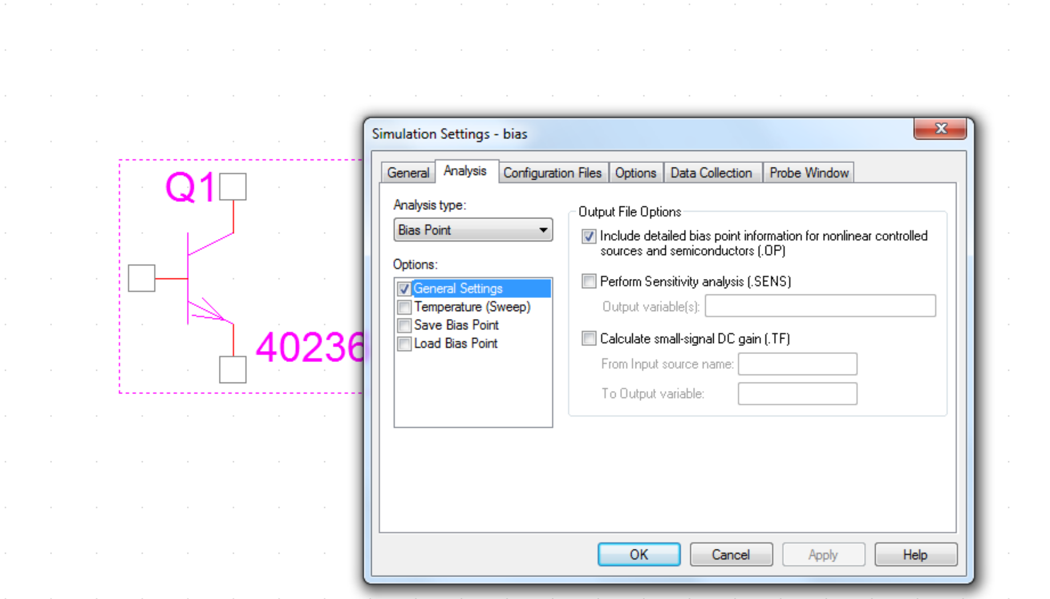


Creating a Circuit Simulation:

Once a circuit is built, you will want to simulate it. You will most likely perform one of five types of simulations.

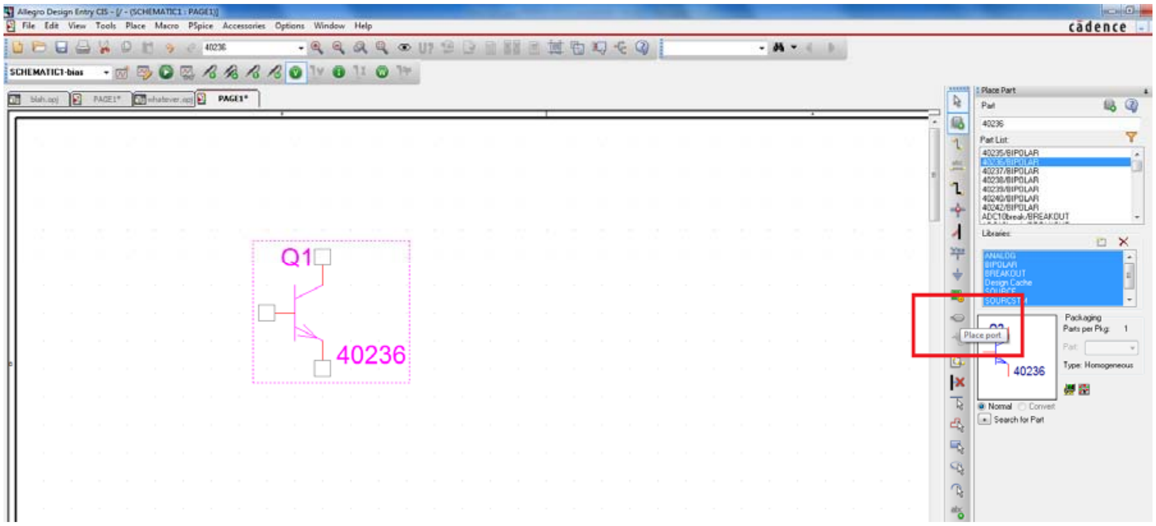
1. DC Analysis
   a. DC sweep
   b. Bias point
2. Transfer Function Analysis
3. AC Analysis
   a. Time Domain (Transient analysis)
   b. Frequency Domain (AC sweep)

To create a simulation: **Select Pspice**>> **Edit Simulation Profile**. Select the **Analysis** tab and then select the simulation you wish to run. Each type will produce a different dialog to the right of the drop-down menu. You will need to set the parameters of the simulation.

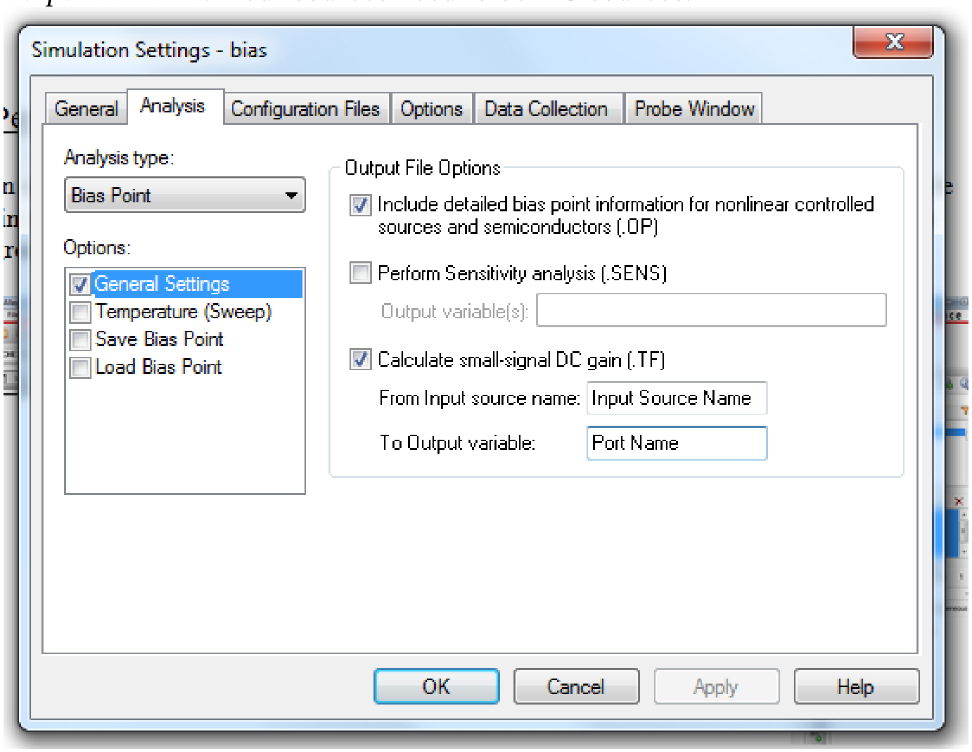


<u>Performing a transfer function analysis (.TF):</u>

In order to perform a transfer function analysis, you must add a port to the output node (in place of the old bubble). It can be found 11 tabs down on the right (two below the ground tab) on the schematics page.



Under Simulation Settings (previously described), select the following options:
Analysis type: Bias Point
Output File Options: Select Calculate small-signal DC gain (.TF). Type the name of your input source in the *From input source name* box and the output port name in *To Output variable*. Your sources need to be DC sources.



Running a Circuit Simulation:

At this time you must save your circuit.

Once the simulation has been created you must run the simulation. Simply hit F11, select Pspice >>Run or hit the blue, right pointing arrow. All three will do the same thing; execute a run of your simulation.

Pspice will generate a ***netlist*** and an ***output file*** for later viewing. It will also flag any errors in your circuit. If you used probes, plots will appear.

You can check your results by either looking at plots or reading the output file.

More Information:

In the "More Pspice Stuff" handout you will find an older handout that you may find useful. It also contains the Pspice project for lab 1 of EE 3221.