# 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.** 

| Please select the suite from which to check out       | the OrCAD Capture feature: |
|---|----------------------------|
| Allegro PCB Librarian XL<br>Allegro PCB Design CIS XL | OK<br>Cancel               |
| Use as default  |                            |

• In the "Allegro Design Entry CIS" window select File -> New -> Project



- In the "New Project" window:
  - Type a name for your new project
  - o Select "Analog or Mixed A/D"
  - o 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

| Create PSpice Project                 | ×              |
|---------------------------------------|----------------|
| Create based upon an existing project | ОК             |
| AnalogGNDSymbol.opj                   | Browse         |
| Create a blank project                | Cancel<br>Help |
|                                       |                |

### 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  $\beta_0$ . Cjc is the reverse bias base-collect capacitance (C<sub>µ</sub>). Cje is the forward bias base-emitter capacitance (C<sub>π</sub>).

```
.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
                                    Vic=.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=T092
*
           88-09-08 bam
                           creation
```

| whatever:Q2N3904 - AMS                             | Model Editor - [Smoke Parameters]  |                    |   |                             | - 0   | x     |
|--|--|--------------------|---|-----------------------------|-------|-------|
| 💾 File Edit View Mode                              | l Plot Tools Window Help   |                    |   | cāder                       | ice   | - 8 × |
|  | 📕 🎖 🖻 🖻 Q Q  | a 🔍 🗊 🗶            | YI 🛃 🗊 🖢  | × 🗠                         |       |       |
| Models List 🛛 🕅<br>Model Name Type<br>Q2N3904* BJT | Test Node Mapping<br>This is the Nodes and Port Mapping.<br>You can enter ports of your choice | These are Dev      | Smoke Parameters<br>rice Maximum Operating cc<br>required for Smoke Analy | 5<br>ondition parame<br>sis | eters | Â     |
|  | Node Port  | Device Max Ops     | Description   | Value                       | Unit  |       |
|  | TERM_IC C  | IB                 | Max base current  |                             | Α     |       |
|  | TERM_IB B  | IC                 | Max collector current   |                             | Α     | -     |
|  | NODE_VC C  | VCB                | Max C-B voltage   | 60                          | V     | =     |
|  | NODE_VB B  | VCE                | Max C-E voltage   | 40                          | V     |       |
|  | NODE_VE E  | VEB                | Max E-B voltage   | 6                           | V     |       |
|  |  | PDM                | Max pwr dissipation   |                             | W     |       |
|  |  | TJ                 | Max junction temp   |                             | С     |       |
|  |  | RJC                | J-C thermal resist  |                             | C/W   |       |
|  |  | RCA                | C-A thermal resist  |                             | C/W   |       |
|  |  | SBSLP              | Second brkdown slope  |                             |       |       |
|  |  | SBINT              | Sec brkdwn intercept  |                             | A     |       |
|  |  | SBTSLP             | SB temp derate slope  |                             | %/C   |       |
|  |  | SBMIN              | SB temp derate at TJ  |                             | %     |       |
|  |  |                    |   |                             |       | -     |
| Model Text   |  |                    |   |                             |       |       |
| .model Q2N3904 N                                   | NPN(Is=6.734f Xti=3 Eg=1.11  | l Vaf=74.03 Bf=416 | .4 Ne=1.259   |                             |       |       |
| +  | 734f Ikf=66.78m Xtb=1.5 Br=  | 7371 Nc=2 Isc=0 3  | Ikr=0 Rc=1  |                             |       | -     |
| + Cjc=3.6  | 538p 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)       |   |                             |       |       |
| * Nationa  | al pid=23 cas  | se=T092            |   |                             |       | Ψ.    |
| Ready  |  |                    |   |                             | NUM   |       |

### 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.

|  | 402 | 236 | Simulation Settings - bias         General       Analysis         Configuration Files       Options         Data Collection       Probe Window         Analysis type:       Output File Options         Dispons:       Include detailed bias point information for nonlinear controlled sources and semiconductors (DP)         Perform Sensitivity analysis (SENS)       Output variable(s):         Save Bias Point       Calculate small-signal DC gain (.TF)         From Input source name:       To Output variable: |
|--|-----|-----|--|
|  |     |     |  |
|  |     |     |  |
|  |     |     | OK Cancel Apply Help   |

### Performing a transfer function analysis (.TF):

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.

|  | General Settings         Temperature (Sweep)         Save Bias Point         Load Bias Point         To Output variable:         Point Normality of the state of the stat |  |
|--|---|--|
|--|---|--|

#### <u>Running a Circuit Simulation:</u> *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.