2

This chapter introduces the mixer circuit and shows all the basics of DC simulations, including a family of curves and device biasing calculations.

Lab 2: DC Simulations

OBJECTIVES

- Build a symbolized sub-circuit for use in the hierarchy

- Create a family of curves for the device used in the mixer

- Sweep variables, pass parameters, and the plot or list the data

- Use equations to calculate bias resistor values from simulation data

NOTE about this lab: This lab and the remaining labs will use the BJT mixer to demonstrate all types of simulations. Regardless of the type of circuit you design, the techniques and simulations presented in these labs will be applicable to many other circuit configurations.

PROCEDURE

The following steps are for creating the mixer BJT sub-circuit with package parasitics and performing the dc simulations as part of the design process.

1. **Create a New Project and name it: mixer**

2. **Open a New Schematic Window and save it as: bjt_pkg**

3. **Setup the BJT device and model:**

   a. Insert the BJT generic device and model: In the schematic window, select the palette: **Devices–BJT**. Select the **BJT-NPN** device and insert it onto the schematic. Next insert the **BJT Model** (model card with default Gummel Poon parameters).
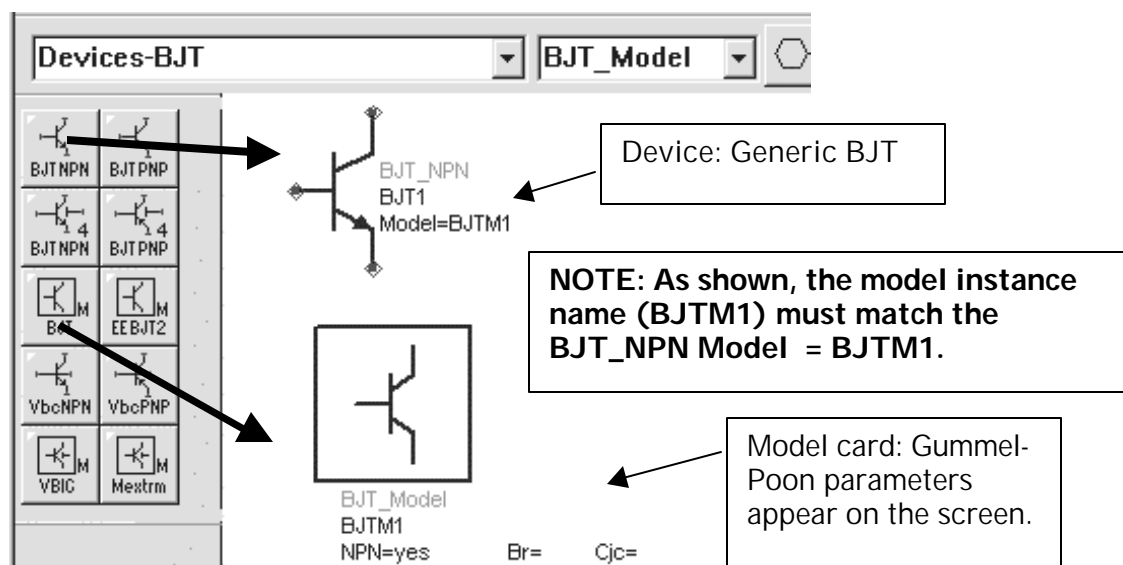
b. Double click on the model. When the dialog appears, click **Component Options** and in the next dialog, click **Clear All** and **OK**. This will remove the parameter list from the schematic.

c. Assign Forward Beta = beta. Double click on the model card you just inserted. Select the **Bf** parameter and type in the word **beta** as shown here. Also, click the small box**: Display parameter on schematic** for **Bf** only and then click **Apply**. The numerical value of beta will be assigned in the next steps.
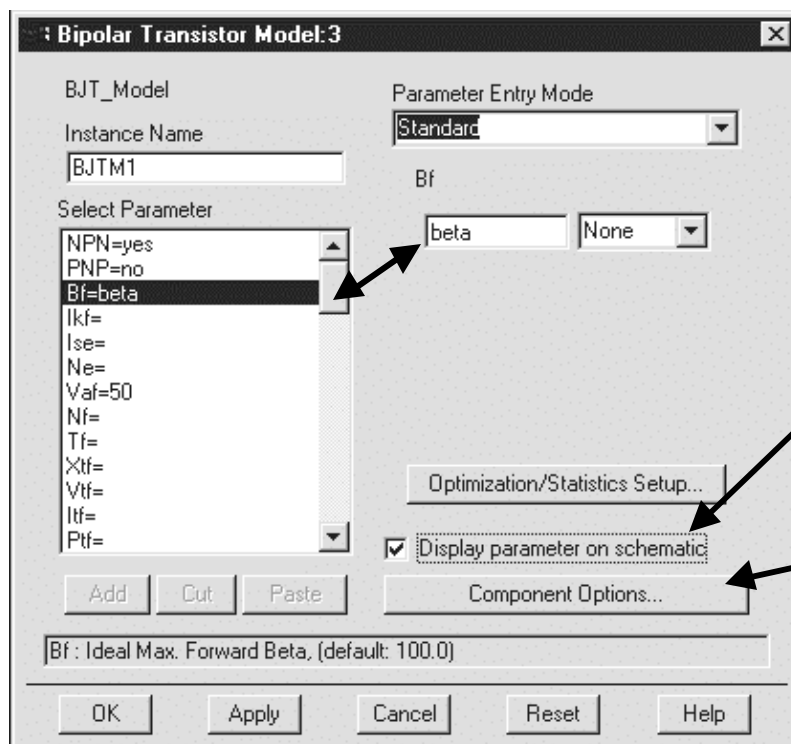
Afterward, click here to display an individual value.

First, click on Component Options to clear the entire parameter display.

d. Type in the value of **Vaf** (Forward Early Voltage) as **50** and display it by clicking **Apply** and **OK**. This will make the dc curves more realistic.

e. Click **OK** to dismiss the dialog box with these changes.

f. For the **BJT device** or any component, you can also remove the unwanted display parameters (Area, Region, Temp and Mode) by editing it in the same way.
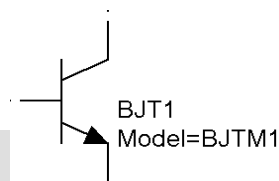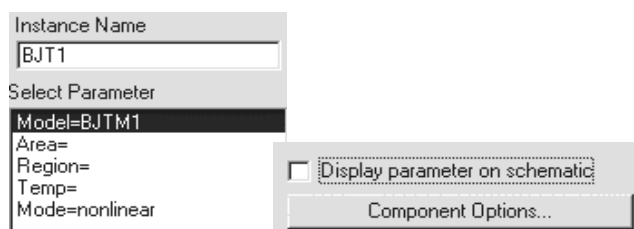
4. Build the rest of the subcircuit

The picture here shows the completed subcircuit. Follow the steps to build it or simply build it as shown:

NOTE: Connect the components together or wire them as needed.

| Add Wire icon. |

a. Insert the package parasitics L and C: Insert three lead inductors (**320 pH**) and two junction capacitors (**120 fF**). Be sure to use the correct units (pico and femto) or your circuit will not have the correct response. Also, add some resistance **R= 0.01 ohms** to the base lead inductor and display the desired component values as shown.

b. Insert port connectors: Click the port connector icon (shown here) and **insert the connectors exactly in this order**: 1) collector, 2) base, 3) emitter. You must do this so that the connectors have the exact same pin configuration as the ADS BJT symbol. Edit the port names – change P1 to C, change P2 to B, and change P3 to E.

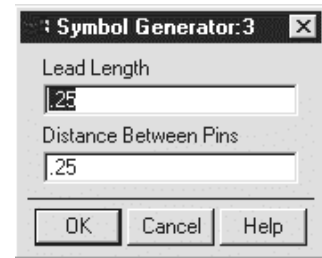NOTE: You must number the ports (num=) exactly as shown or the device will not have the correct
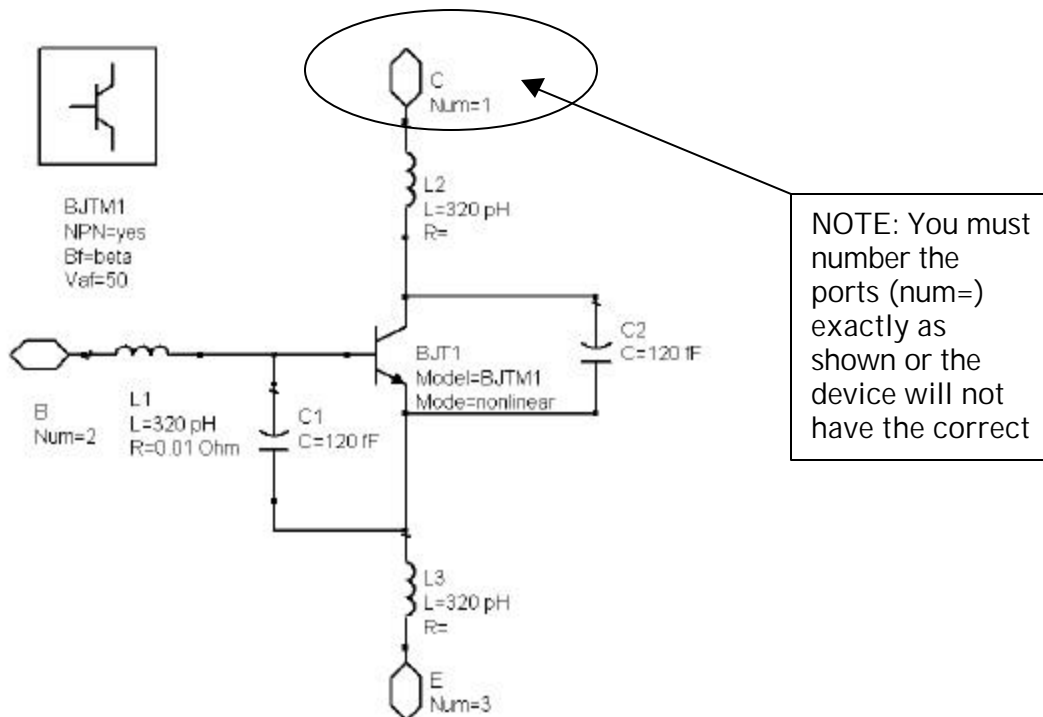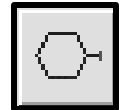
c. Clean up the schematic: Position the components so the schematic looks organized – this is good practice. To move component text, press the **F5**-Key and then **select the component**. Use the cursor to position the text.

5. Create a symbol for the sub-circuit

There are three ways to create a symbol for a circuit: 1) Use a default symbol, 2) Use a built-in symbol (a standard symbol), or 3) Create a new symbol by drawing one or modifying an existing one. For this lab you will use a built-in bjt symbol which looks better than the default three-port symbol. The following steps shows how to do this:

a. To see the default symbol, click**: View > Create/Edit Schematic Symbol**. The symbol page will replace the schematic page and a dialog will appear. Click OK to use the defaults.

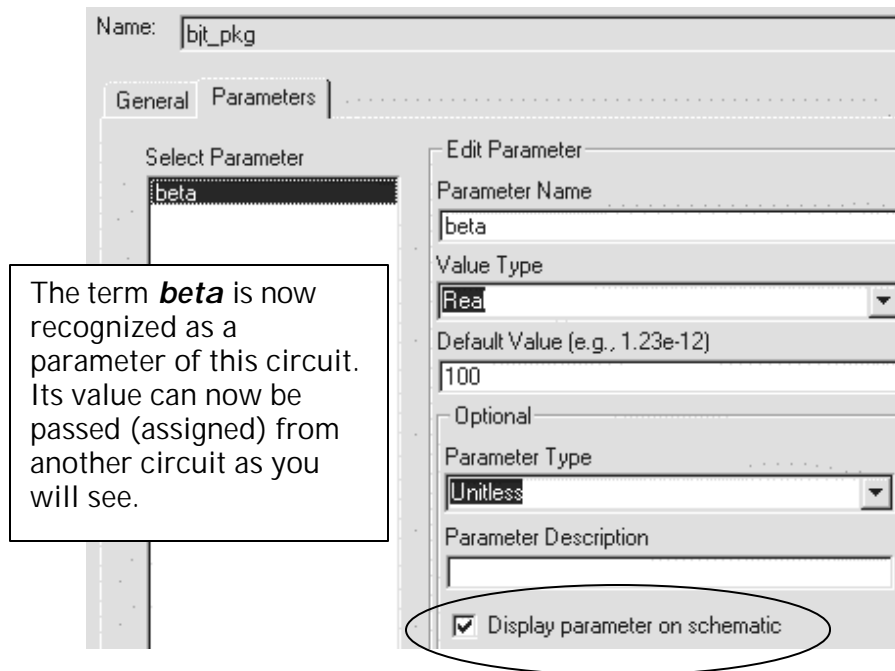b. Next, a rectangle or square with three ports is generated:

NOTE: You will be replacing the default symbol with a built-in BJT symbol in the next steps. As you do, you must assign the pin (port) numbers exactly as shown to match the built-in symbol for the emitter, base, and collector.
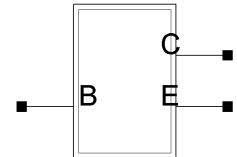
c. To change the symbol to a built-in symbol that looks like a transistor, delete the entire symbol you just created: **Select > Select All**. Then click the **trash can** icon to delete the symbol.

d. Return to the schematic: **View > Create/Edit Schematic**. Now click **File> Design Parameters**. In the **General** tab, there is a **Symbol Name** parameter list. Click the arrow and select: **SYM_BJT_NPN**. Also, Change the component instance name to **Q**.

File >Design Parameters
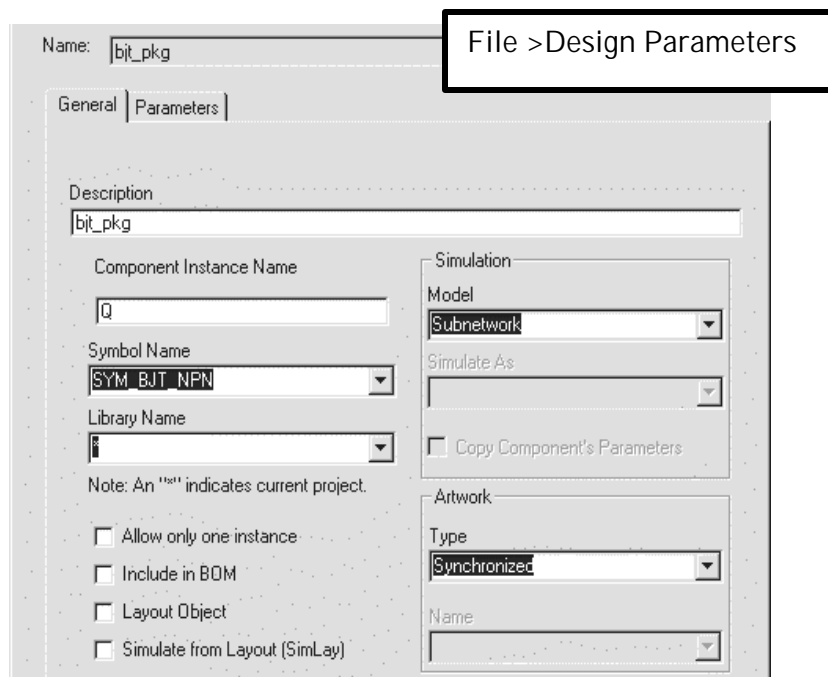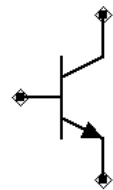
e. Set beta as a pass parameter: To do this, click the **Parameters** tab. In the Parameter Name area, type in **beta** and assign a default value of **100** by clicking the **Add** button. Be sure to click the **Display** button as shown in the picture. Click the **OK** button at the bottom (not shown here) to save the new definitions and dismiss the dialog.

Name: bjt_pkg

General | Parameters |

Select Parameter

beta

Edit Parameter

Parameter Name

beta

Value Type

Real ▼

Default Value (e.g., 1.23e-12)

100

Optional

Parameter Type

Unitless ▼

Parameter Description

The term *beta* is now recognized as a parameter of this circuit. Its value can now be passed (assigned) from another circuit as you will see.

☑ Display parameter on schematic

f. In the schematic window, **Save** the design to make sure all your work is save and **close the window**. You now have a sub-circuit that will be available for use in other designs and other projects.
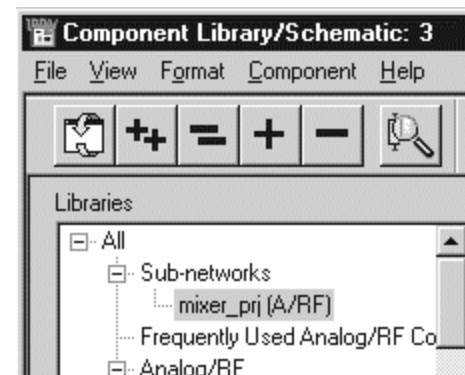
## 6. Create another circuit for DC simulations

a. Open a new schematic from the Main window and save it as: **dc_curves**. This will be the upper level circuit.

b. Click on the **Library list icon** and the library browser will appear. Select the **mixe**r project and you will see the **bjt_pkg** circuit listed as an available component.

Component Library/Schematic: 3

File   View   Format   Component   Help

Libraries

☐ All
  ☐ Sub-networks
    mixer_prj (A/RF)
  Frequently Used Analog/RF Co
  ☐ Analog/RF

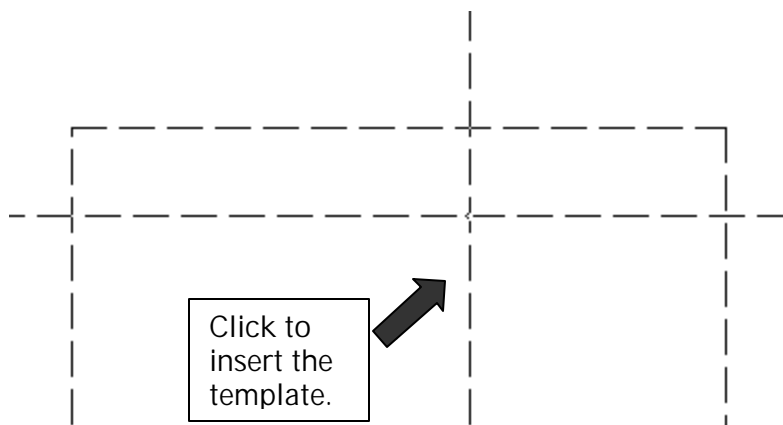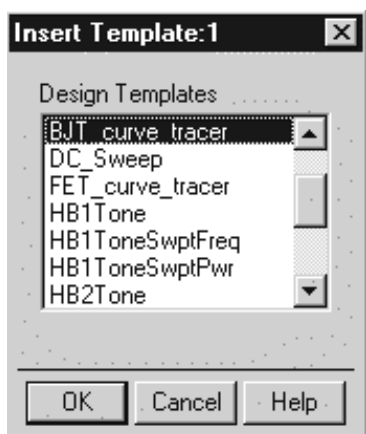| Components	
Component	Description
bjt_pkg	bjt_pkg

c. Select the **bjt_pkg component** and the npn transistor symbol will be
    appear on your cursor. Click in the dc_curves schematic to insert the
    bjt_pkg. You can now **close the library window** and save the
    dc_curves design (good practice to save often).


bjt_pkg
Q1
beta=100
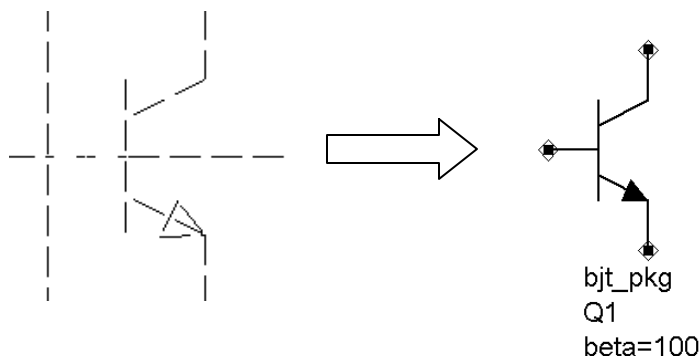
7. Set up a dc curve tracer

For this step you will use a template. ADS built-in templates make it easier to set up
the simulation after the schematic is built. In this case, the dc curve tracer template is
set up to sweep VCE within incremental values of base current IBB.

a. On the schematic, click **File > Insert Template** and select the
    BJT_curve_tracer to insert it. Click **OK** and it will appear on your
    cursor - to insert it, click near your bjt_pkg symbol.
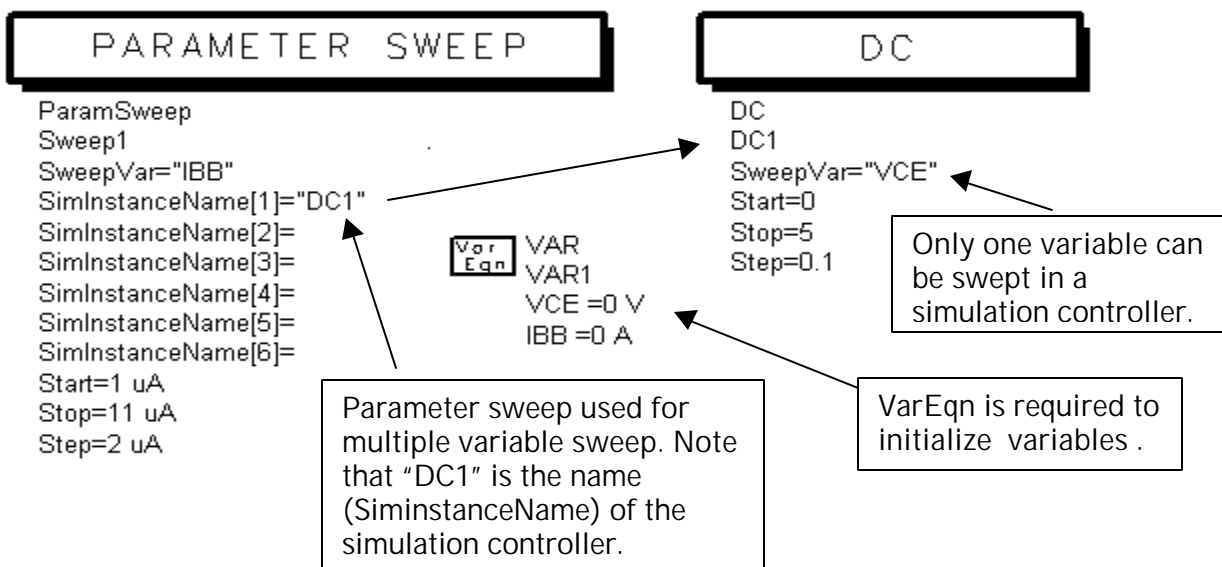

Click to
insert the
template.

b. With the curve tracer template inserted, **wire the circuit together** so it looks like the shown here. Note that you can move the component text using the F5 key so that the schematic looks good.
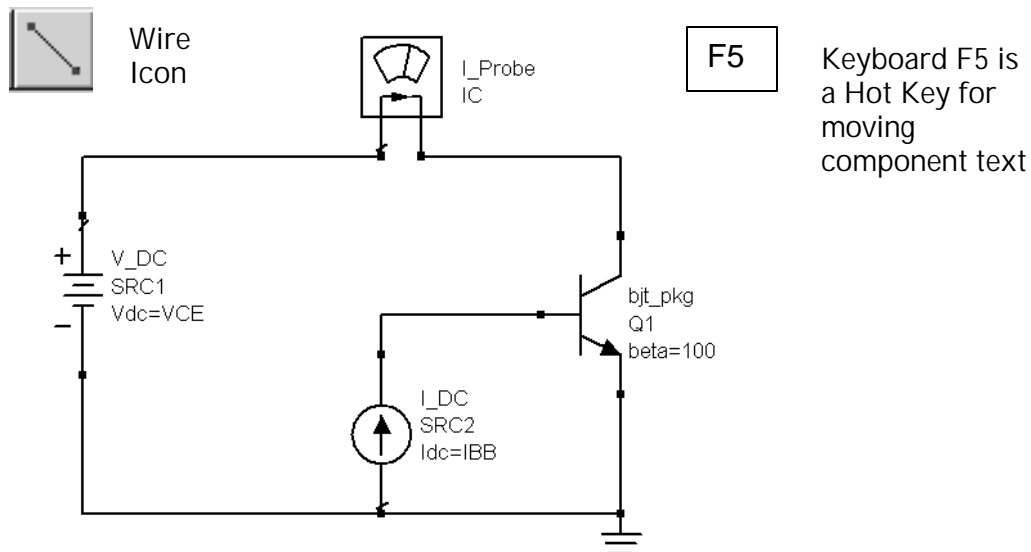
NOTE: If you did not use this Template, you would have to insert every component (the V_DC source, the I_Probe, the I_DC source, etc.) one at a time. Also, you would have to assign and set up the variables (IBB, VCE) for the swept simulation.

c. Set the **Parameter Sweep** IBB values: **1 uA to 11 uA in 2 uA steps**. Parameter Sweep components are available in all simulation palettes. Set the DC simulation controller **SweepVar VCE**: **0 to 5 in 0.1 steps**. Notice that the VAR1 variables VCE and IBB can be used as is because they only initialize the variables but it is best to use reasonable values.

8. **Name the dataset and run the simulation**

   a. Click **Simulate > Simulation Setup**. When the dialog appears, type in a name for the dataset **dc_curves** as shown.

   b. Click **Apply** and **Simulate**.

   c. After the simulation is finished, click the **Cancel** button and the setup dialog will disappear. If you get an error message, check the simulation set up and repeat if necessary.
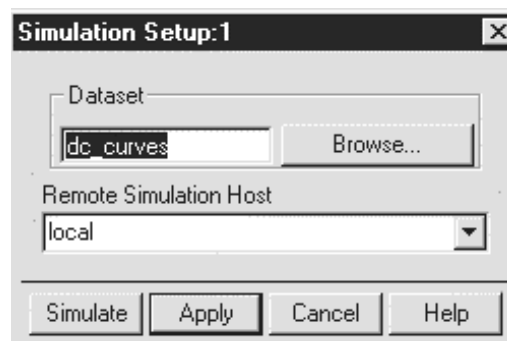
9. **Display the results, change beta, and resimulate**

   a. Click the **New Data Display icon** (shown here). Insert a **rectangular plot** and add the **IC.i data**. Note that voltage VCE is the default X-axis value. The results should look similar to the "beta=100" plot shown here.

   b. On the schematic, change the value of **beta = 144**. The value will automatically be passed down to the sub circuit that you set up in the previous steps. Simulate again and notice the change as shown here. NOTE: **You will use beta =144 for the remainder of the labs**.
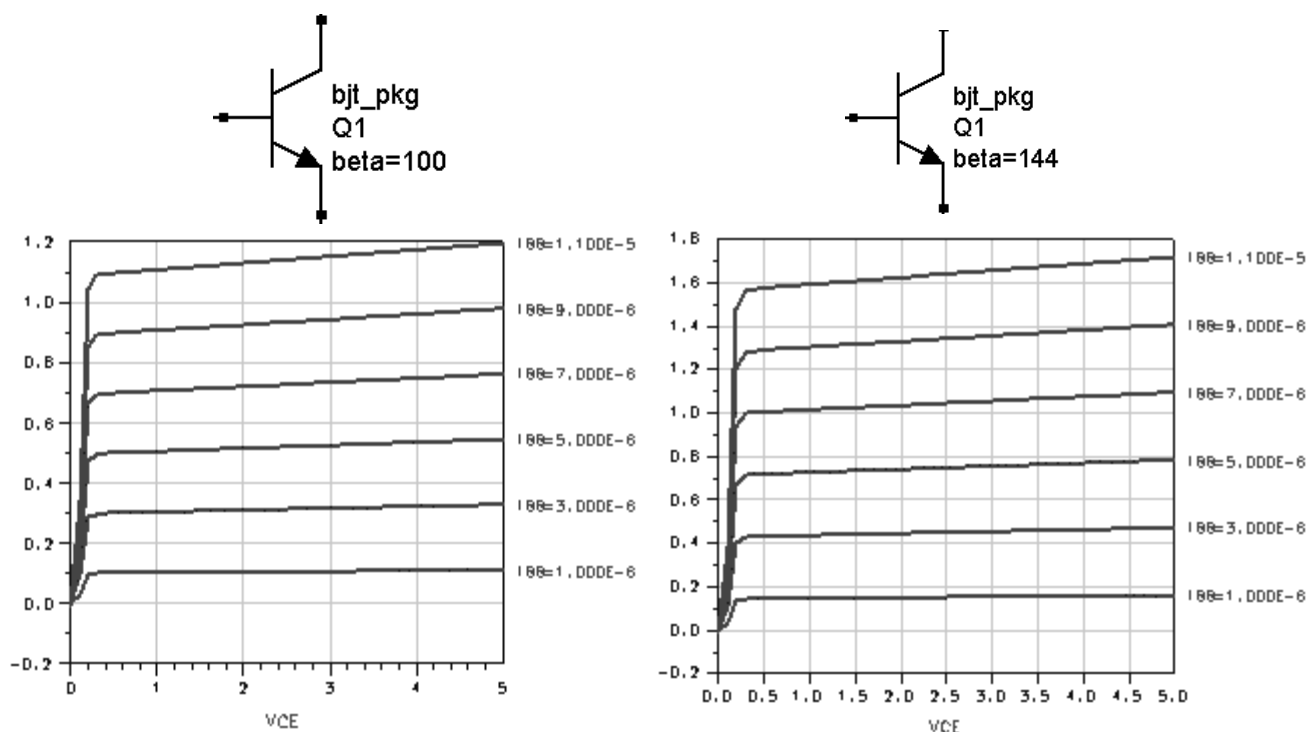
c. **Insert a marker** on the *dc_curves* trace (as shown here), where the initial specification of 1 mA at VCE corresponds to about 7 uA of base current.
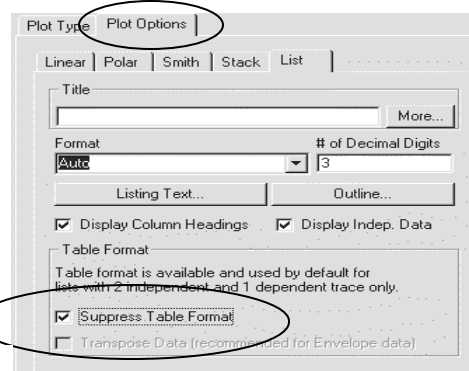
d. **Insert a list** (click the icon).

 List Icon

e. Select collector current IC and add it . If the list is in table format as shown (box with X across it), edit or double click the list and check the box, **Suppress Table Format** and OK. Then scroll through the data.
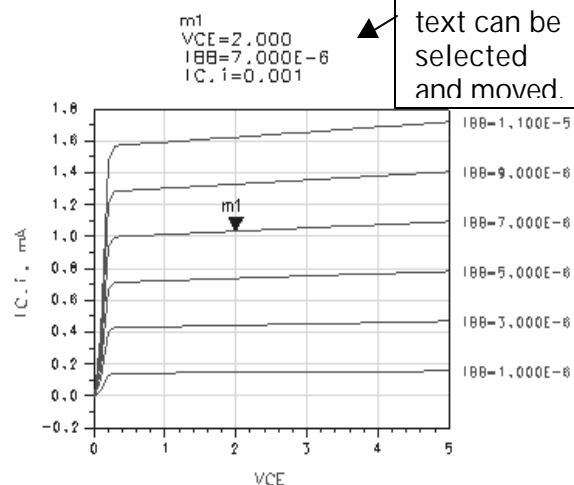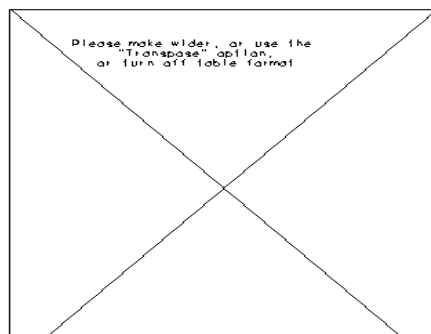
VCE	IC.i
5.000	781.0uA
IBB=7.000E-6	
0.000	-6.952uA
0.100	242.4uA
0.200	939.4uA
0.300	997.3uA
0.400	1.001mA
0.500	1.003mA
0.600	1.005mA
0.700	1.007mA
0.800	1.009mA
0.900	1.011r
1.000	1.013r
1.100	1.015r
1.200	1.017r
1.300	1.019mA
1.400	1.021mA
1.500	1.023mA
1.600	1.025mA
1.700	1.027mA
1.800	1.029mA
1.900	1.031mA
2.000	1.033mA
2.100	1.035mA

Use the scroll buttons to rapidly move through the data at these increments:
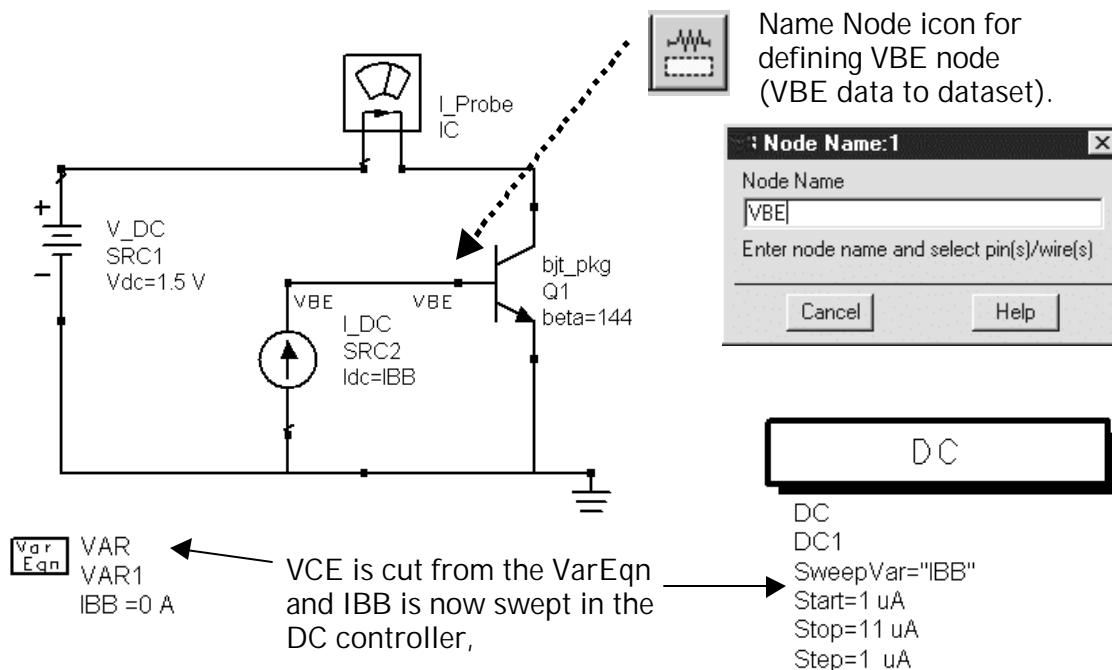
to the end, a page, or one data line.

2V VCE at IBB of 7 uA shows about 1 mA of collector current. Use the scroll button on the top of the display

DC Bias DESIGN CONSIDERATION: When the final circuit is constructed, the LO drive will shift the current slightly higher and this means that the operating point can be a little lower if desired.   In addition, a current limiting collector resistor RC will be required and that will lower the voltage across VCE. Knowing this, it is reasonable to assume that VCC of 2 volts will be divided with a voltage drop of about 0.5V for RC with the remaining 1.5V across the device VCE.

10. Create a new design to calculate bias values

The next steps will sweep only base current for a fixed value of VCE at 1.5 volts. This will allow you to determine values of base-emitter voltage VBE that can be used to calculate the bias resistor values.

   a. **Save the dc_curves schematic**. Next, save it with a new name as follows: click **File > Save As** and when the dialog box appears, type in a new name: **dc_bias**. Now, you have three designs in the networks directory: bjt_pkg, dc_curves and dc_bias.

   b. If only one variable is swept, it is more effective to sweep it in the Simulation controller and not in a Parameter sweep. Therefore, **delete the Parameter Sweep**. Refer to the schematic here to: **1)** edit the DC controller to **sweep IBB: 1uA to 11 uA in 1 uA steps**, **2)** set **Vdc = 1.5V**, and **3)** remove VCE from the VarEqn by editing it (double click) and using the Cut button to remove VCE as a variable.

   c. Insert a **_node name_** to allow you to get simulation data from a node on the schematic. Click the icon or use the command: **Component > Name Node**. When the dialog appears, type in the name **VBE** and click on the node at the base of the transistor.


Name Node icon for defining VBE node (VBE data to dataset).

VCE is cut from the VarEqn and IBB is now swept in the DC controller,

d. **Save and Simulate**: Save the new design by clicking the save icon – this is always good practice. Next, check the dataset name: **Simulation > Setup**) as in the previous simulation. Be sure it appears as: **dc_bias** and then **Simulate**.

11. Display the data (dc_bias) in a list

In this step, you will use the same data display window that contains the dc_curves data. In fact, you can plot numerous datasets in the display but you must explicitly define (dataset name..) the data to be displayed.

a. In the current Data Display window, notice that the default dataset is dc_curves. This is OK. However, if you **change the default to dc_bias**, you will see that the plot becomes *invalid* because the data is not the same array size as the two dimensional one. This is normal. Try this now as shown and then **set it back to dc_curves**.
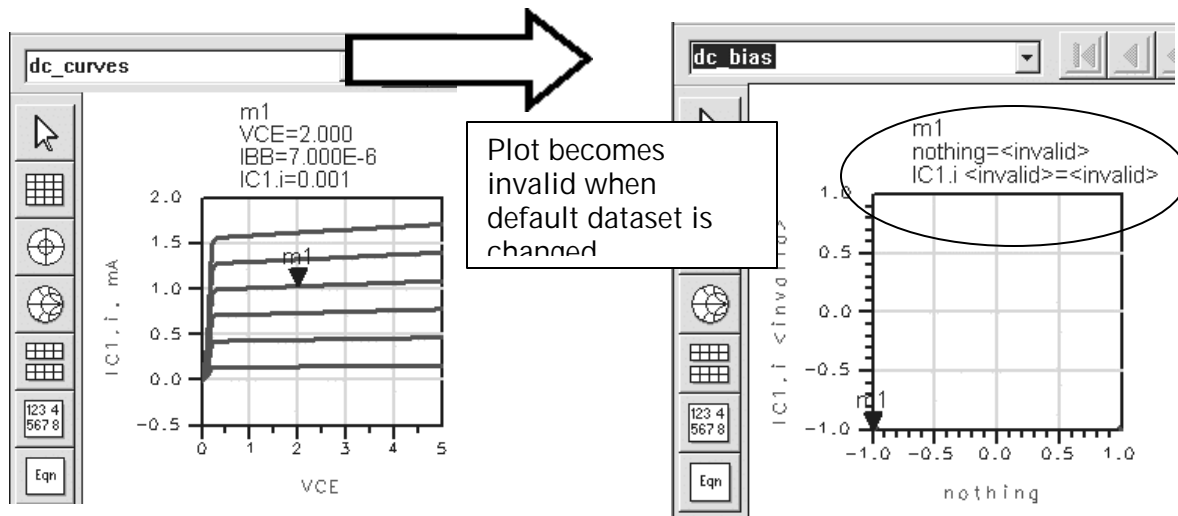
b. Now, in the current Data Display window, make room for the new data by using the **zoom and view icons**. Then **insert a new list**.

c. When the list dialog box appears, select the **dc_bias dataset** and, **add VBE and IC**. You should get results similar to those here where IC is very close to 1 mA.
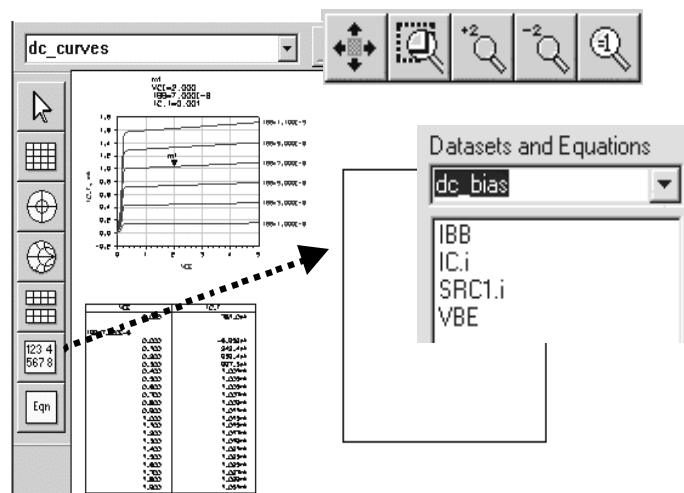
d. **Draw a box around the values** of interest as shown here. To do this, click the **rectangle icon** from the tool bar and draw it on the list. This is one way to highlight the data. Also, the data display window by using Save As and giving it a name like: dc_data.

| IBB	dc_bias..IC.i	dc_bias..VBE
1.000E-6	146.2uA	719.3mV
2.000E-6	292.4uA	737.1mV
3.000E-6	438.5uA	747.5mV
4.000E-6	584.6uA	754.9mV
5.000E-6	730.6uA	760.6mV
6.000E-6	876.7uA	765.3mV
7.000E-6	1.023mA	769.3mV
8.000E-6	1.169mA	772.7mV
9.000E-6	1.315mA	775.7mV
1.000E-5	1.461mA	778.4mV
1.100E-5	1.607mA	780.9mV

12. Write an equation to calculate Rb

a. On the data display, insert an equation by clicking on the equation icon and then clicking in the data display window:

NOTE: minus sign

Eqn Rb=(1.5-dc_bias..VBE)/dc_bias..IBB

b. When the dialog appears, type in the equation as shown by typing and using the Insert button. First, select the **dc_bias** dataset in the upper right (circled). To write the equation type the first part only: **Rb = (1.5 -** and select **VBE** and click <<**Insert**<<. Then type in the parenthesis and division sign: **)/**. Then **insert IBB** in the same way and click **OK**. If the equation is RED (invalid), repeat the step or ask the instructor for help.
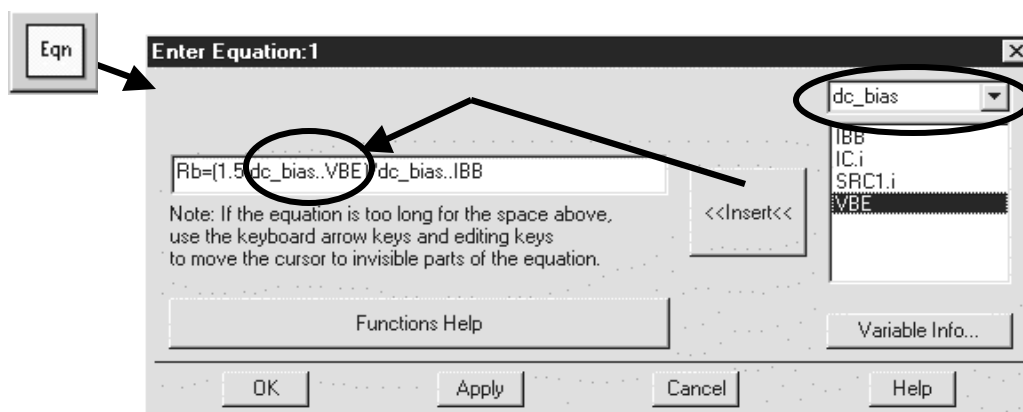
IMPORTANT NOTES on writing equations

Equations that operate on data can either be explicit or generic:

Explicit Equation: A
specific dataset is
referenced: name..

Eqn Rb=(1.5-dc_bias..VBE)/dc_bias..IBB

Generic Equation: When no
dataset is specified, the
equation applies to the
default dataset.

Eqn Rb1=(1.5-VBE)/IBB

The difference in these two equations is in the data being referenced, especially the default dataset in the case of the generic equation. Also, note that equations and data are CASE SENSITIVE.

c. Verify how the generic equation described above will work. Be sure the data display shows **dc_curves as the default dataset**. Now, insert another equation and type it in as shown (generic version):

Rb1 = (1.5 - VBE) / IBB.   After you click OK and it will be *red* (invalid)

d. Now, **change the default dataset to dc_bias** (at the top of the display) and verify that it is valid.

dc_bias

Eqn Rb1=(1.5-VBE)/IBB

Equation is black (valid)
because the default dataset
corresponds to the equation..
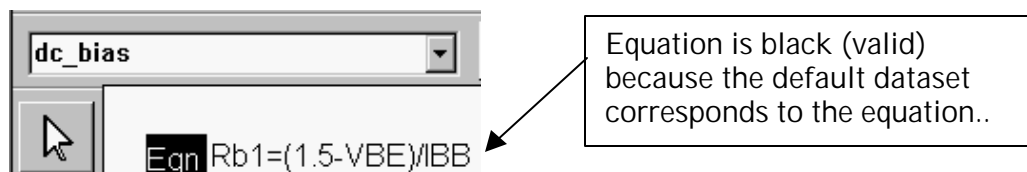
Now, continue with the design by calculating the collector resistor.

e. **Write an equation for resistor Rc**.   You should be able to do this based on what you learned in the previous steps.

Eqn Rc=0.5 / (dc_bias..IC.i + dc_bias..IBB)

f. **List the values Rb and Rc**.
Insert a List and when the
first dialog appears, select
Equations by clicking the
arrow. Then **Add Rb and Rc**
and click **OK**.

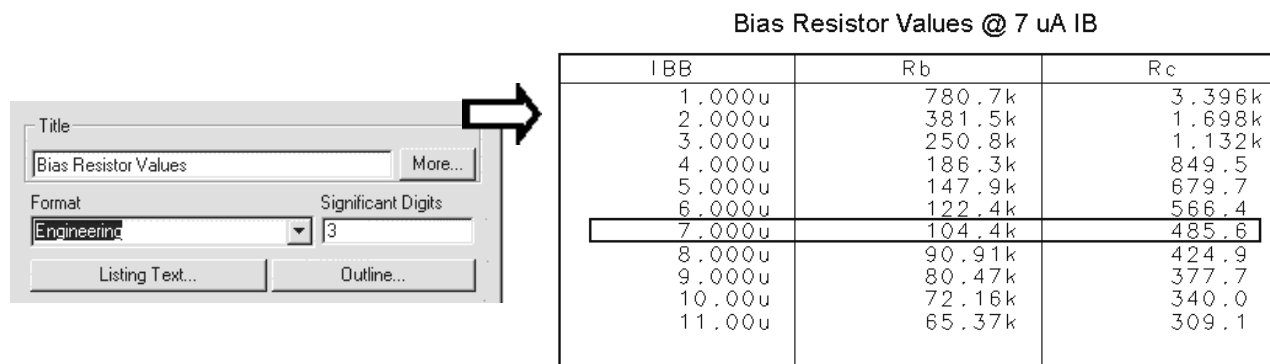| Datasets and Equations		Traces
Equations		Trace Options...
Rb		Rb
Rc	>>Add>>	Rc
m1		
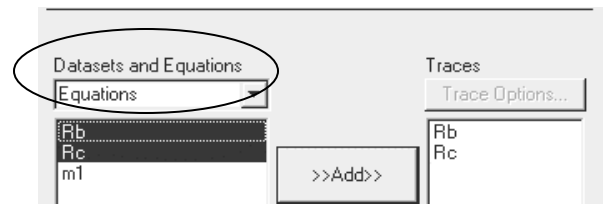
g. When the list appears, you will then see a table of values for Rb and Rc
that correspond to the value of IBB. As a rule, you always get the
independent variable (here IBB) when you list or plot data.

h. Increase the size of a display (if you see dots …after the entries), by
dragging the corner of the list. If dots appear after a number or name, it
indicates there is more data and you should
increase the size of the list or plot.

I BB	Rb	Rc
1	7 8 . . .	3 3 . . .
2	3 8 . . .	1 6 . . .
3	2 5 . . .	1 1 . . .
4	1 8 . . .	8 4 . . .
5	1 4 . . .	6 7 . . .
6	1 2 . . .	5 6 . . .

I BB	Rb	Rc
1 . 0 0 0 E − 6	7 8 0 7 1 4 . 7 9 4	3 3 9 5 . 6 2 1
2 . 0 0 0 E − 6	3 8 1 4 5 2 . 9 6 9	1 6 9 8 . 4 0 2
3 . 0 0 0 E − 6	2 5 0 8 2 9 . 4 7 4	1 1 3 2 . 4 9 9
4 . 0 0 0 E − 6	1 8 6 2 7 4 . 2 6 8	8 4 9 . 4 9 7
5 . 0 0 0 E − 6	1 4 7 8 7 2 . 7 7 9	6 7 9 . 6 7 4
6 . 0 0 0 E − 6	1 2 2 4 4 6 . 5 9 1	5 6 6 . 4 4 7
7 . 0 0 0 E − 6	1 0 4 3 8 8 . 4 2 6	4 8 5 . 5 6 4
8 . 0 0 0 E − 6	9 0 9 1 1 . 0 2 3	4 2 4 . 8 9 7
9 . 0 0 0 E − 6	8 0 4 7 3 . 5 5 6	3 7 7 . 7 0 8
1 . 0 0 0 E − 5	7 2 1 5 5 . 5 0 0	3 3 9 . 9 5 5
1 . 1 0 0 E − 5	6 5 3 7 3 . 2 9 2	3 0 9 . 0 6 5

i. Draw a box (rectangle around the desired values to read it easier. Then
edit the list (double click) and select **Plot Options**. Now, type in a title
and change the format as shown by using the **More** button if desired.

Bias Resistor Values @ 7 uA IB

I BB	Rb	Rc
1 . 0 0 0 u	7 8 0 . 7 k	3 . 3 9 6 k
2 . 0 0 0 u	3 8 1 . 5 k	1 . 6 9 8 k
3 . 0 0 0 u	2 5 0 . 8 k	1 . 1 3 2 k
4 . 0 0 0 u	1 8 6 . 3 k	8 4 9 . 5
5 . 0 0 0 u	1 4 7 . 9 k	6 7 9 . 7
6 . 0 0 0 u	1 2 2 . 4 k	5 6 6 . 4
7 . 0 0 0 u	1 0 4 . 4 k	4 8 5 . 6
8 . 0 0 0 u	9 0 . 9 1 k	4 2 4 . 9
9 . 0 0 0 u	8 0 . 4 7 k	3 7 7 . 7
1 0 . 0 0 u	7 2 . 1 6 k	3 4 0 . 0
1 1 . 0 0 u	6 5 . 3 7 k	3 0 9 . 1

Title

| Bias Resistor Values | More... |

Format — Significant Digits

| Engineering | ▼ | 3 |

| Listing Text... | Outline... |

j. Be sure to **save the display** (.dds file). With these values of Rb and Rc,
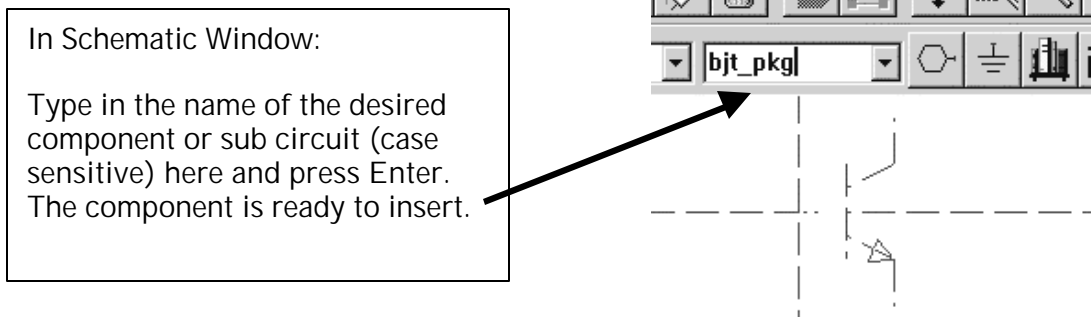the next step is to bias the device and test the bias network.

13. Set up a new design to test the bias network

For this step, you will create the schematic design without using a template. During this process you will learn some efficient ways to do this.

    a. Open a new schematic from the existing one, using the File > New command or the icon and name it: **dc_net**. Notice that this dialog allows you to name the new design and gives you other options.
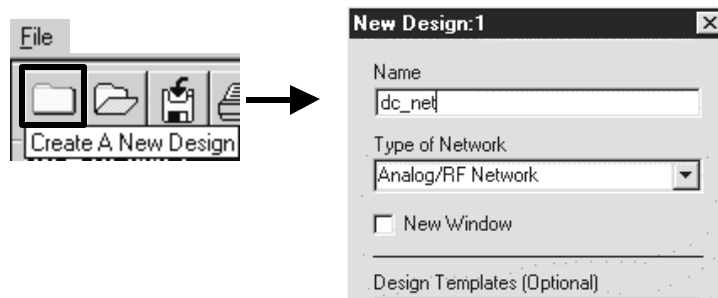

    b. In the new schematic (dc_net), insert your sub circuit **bjt_pkg** by typing in the name in the component history list:

In Schematic Window:

Type in the name of the desired component or sub circuit (case sensitive) here and press Enter. The component is ready to insert.
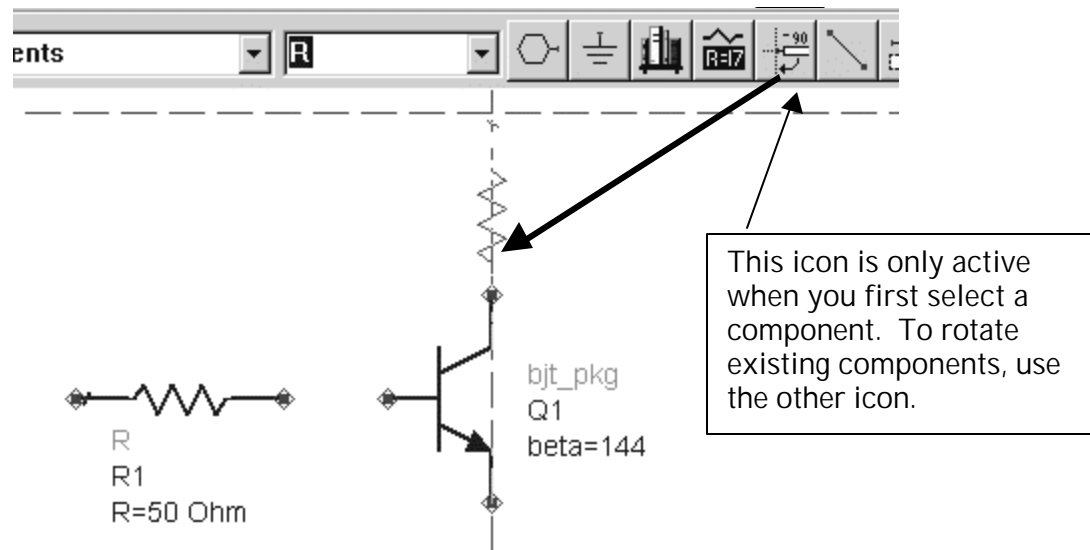
    c. Set the value of **beta** to: **144**

    d. Goto **Lumped Components** palette and insert a **resistor** as the base resistor. Notice that "R" appears in the history list when you do this.


2-16
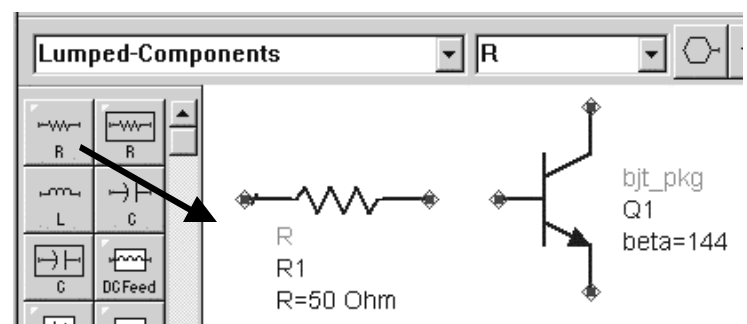
e. Insert the collector resistor and rotate it: put the cursor in the history list "R" and press Enter. Immediately, when the resistor is attached to your cursor, click the –90 rotate icon shown here and the component will increment 90 degrees – then insert it.


> This icon is only active when you first select a component. To rotate existing components, use the other icon.

f. Insert a **current probe (I_Probe)** from the palette or type it in. Connect it to the top of the collector resistor.

> After you connect the component, you can drag it and it is automatically wired.
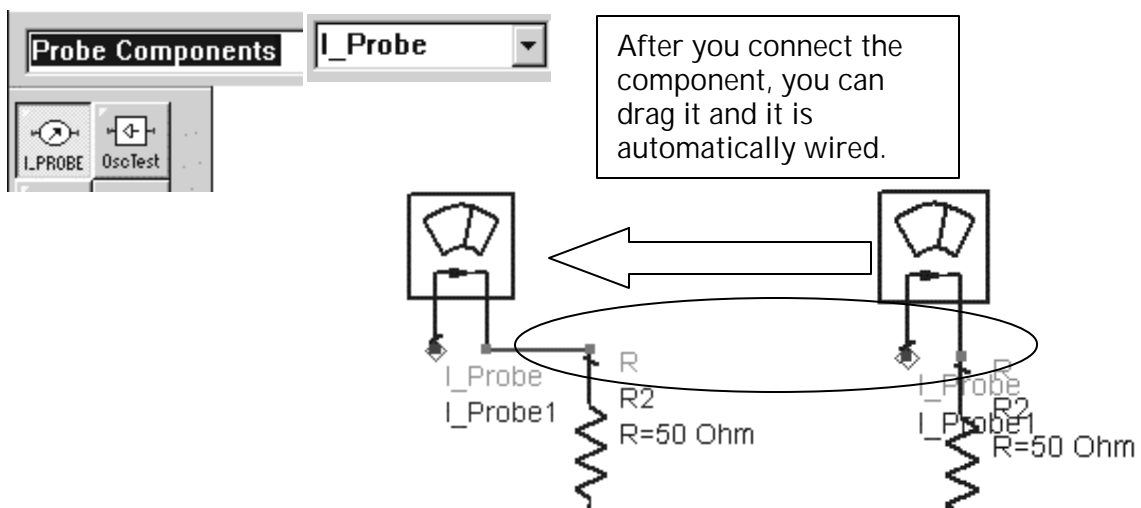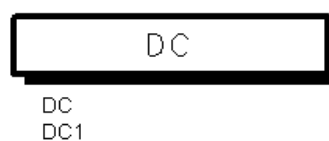
g. **Finish building the circuit as follows**:

- Rename and assign resistors: **Rb** = **100 K ohms** and **Rc** = **470 ohms**.

- Rename the I_Probe: **IC**

- Insert **V_DC** supply set at **2 V** from (Sources-Frequency Domain palette).

- Insert a node name at the collector as **VC**.

- Wire the circuit and organize it.
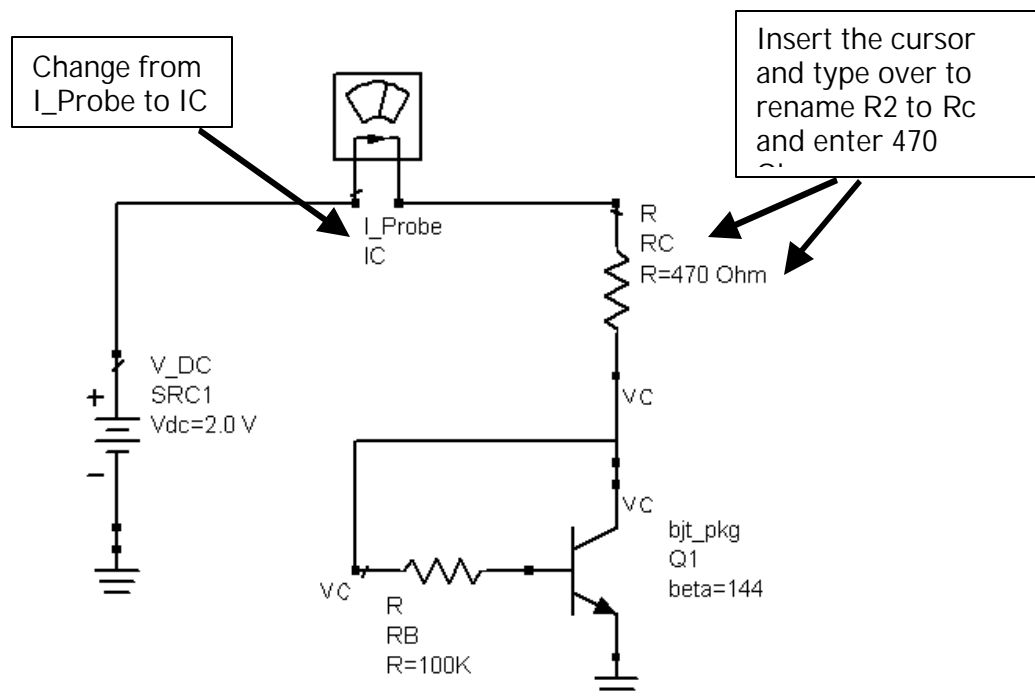
NOTE on Name Node: To remove a *named node,* click **Edit > Component > Remove Node Name** or you can rename the node with a blank (click the icon and try it). This step is to show how to remove a node name – you may need it later on**.**

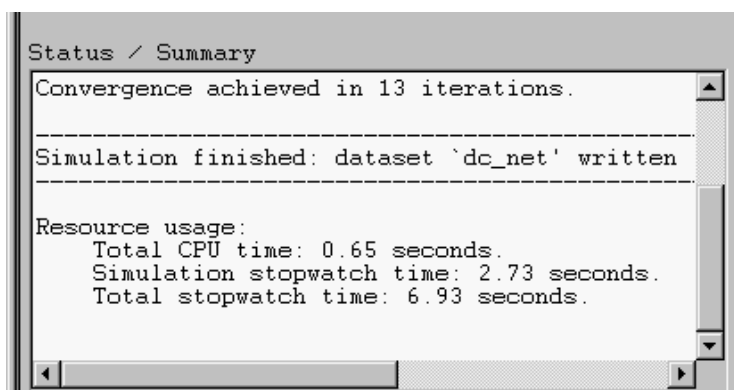h. **Insert a DC simulation controller** (Simulation-DC palette).

14. Simulate and verify the bias network conditions

For this you do not need to display the data. Instead, you will simply annotate the schematic to verify that IC meets the 1 mA specification and that bias design consideration (described earlier) is accurate.

a. Press the **F7** keyboard key and the simulation will be launched with the dataset name that is the same as the schematic – this is the default. You can verify this by reading the status window:

```
Status / Summary
Convergence achieved in 13 iterations.

------------------------------------------------------
Simulation finished: dataset `dc_net' written
------------------------------------------------------

Resource usage:
     Total CPU time: 0.65 seconds.
     Simulation stopwatch time: 2.73 seconds.
     Total stopwatch time: 6.93 seconds.
```

b. Annotate the current and voltage at the nodes. Click on the menu command: **Simulate > Annotate DC Solution**. Now you should see the voltage and currents at the nodes. Be sure that you have about 1 mA of collector current with VCE about 1.5 V. If not, check your work.
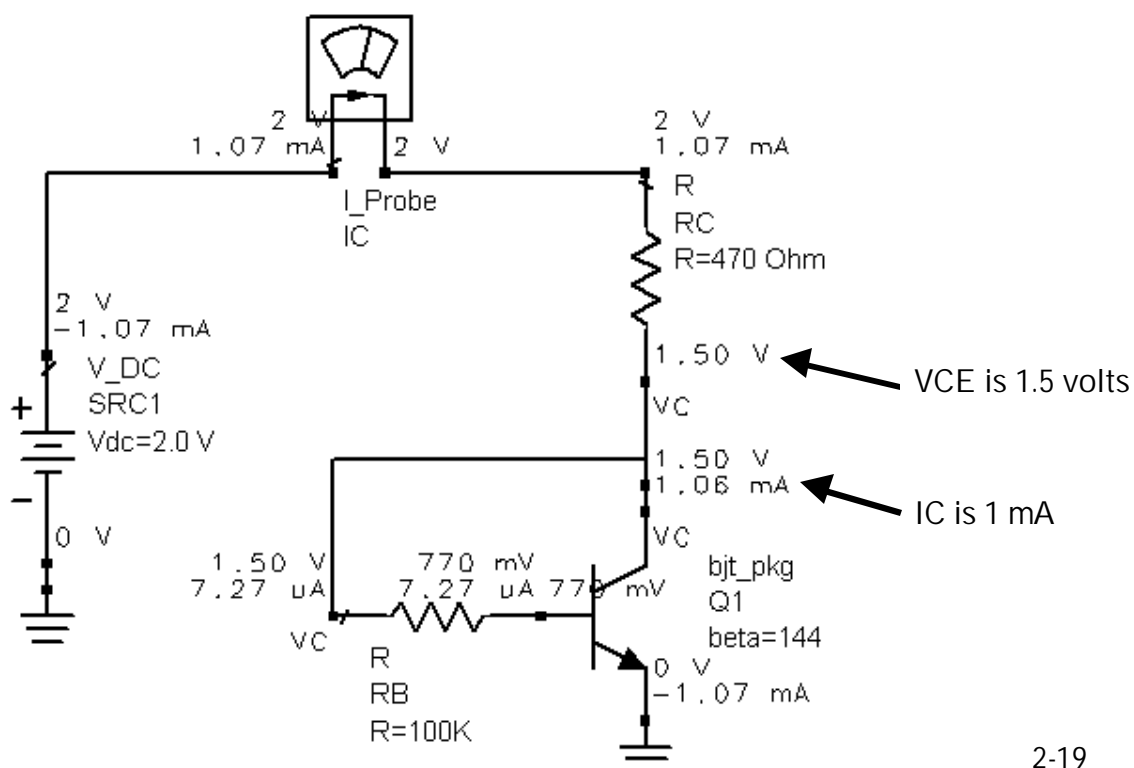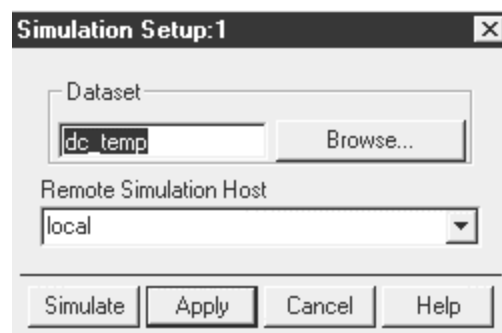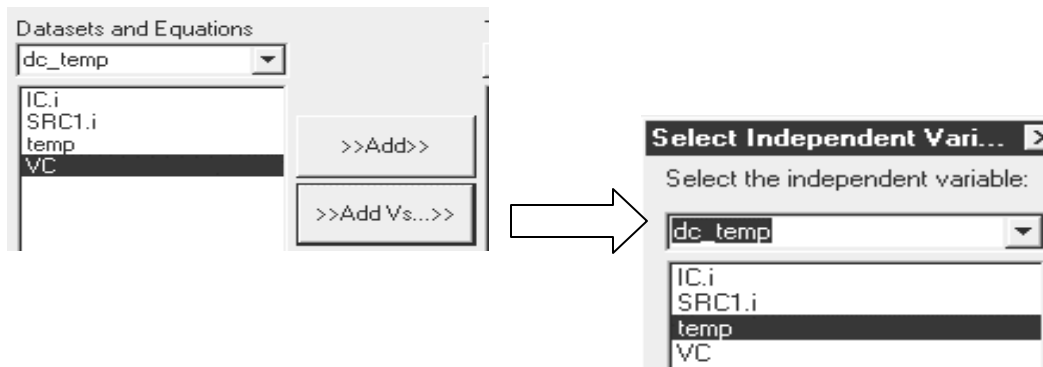
15. Sweep Temperature

a. Edit the **DC controller** – select it and click the edit icon.

b. In the **Sweep tab**, enter the ADS global variable **temp** (default is Celsius) as shown here and enter the sweep range: **-55 to 125 @ 5 step**. Also, in the Display tab, click the boxes to display the annotation on the controller – click **Apply** to see it and **OK** to dismiss the dialog.
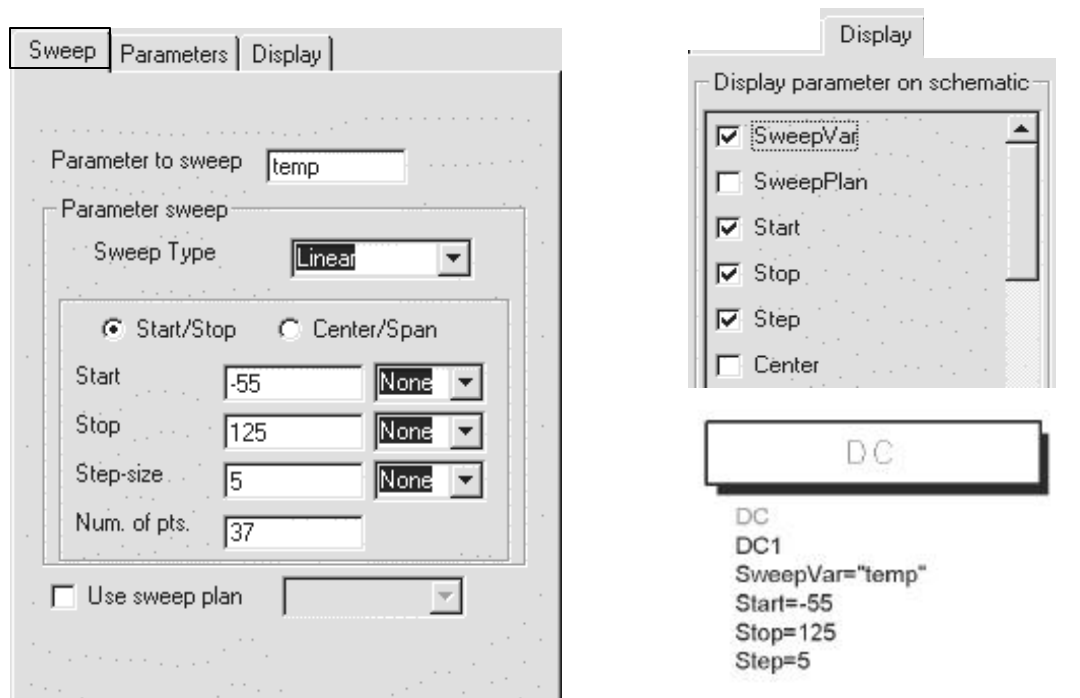

c. Set the simulation dataset name to **dc_temp**, click Apply to assign that dataset name, and then **Simulate**.

d. Plot the results in a rectangular plot as **VC vs temp** - you should be able to do this as shown:

The plot should look like the one shown here: collector voltage decreases as the temperature increases. You can use this temperature sweep method for any simulation in the future.
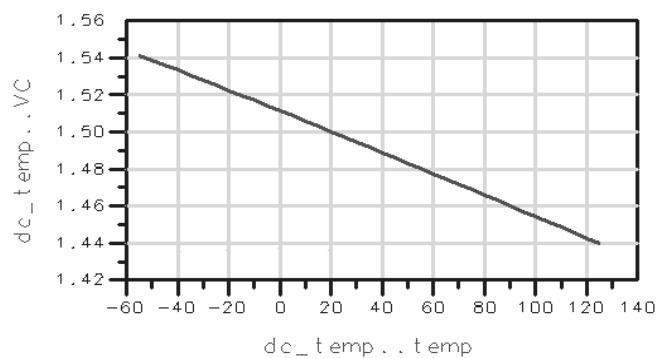
EXTRA EXERCISES

1. Plot current (probe) vs. temperature.

2. Try these commands:

   a. Select the bjt and click the command: Edit > Component > Break Connections. Reinsert the bjt and see what happens.

   b. Spend a few moments experimenting with the other Simulation menu commands: Highlight Node and Detailed Device Operating Point. These are only available after a dc simulation.

   c. Go to the data display: Use the right mouse button and experiment with the selections.

2. Replace the Gummel-Poon model card with another model (Mextram) and resimulate. Afterward, compare the results.