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

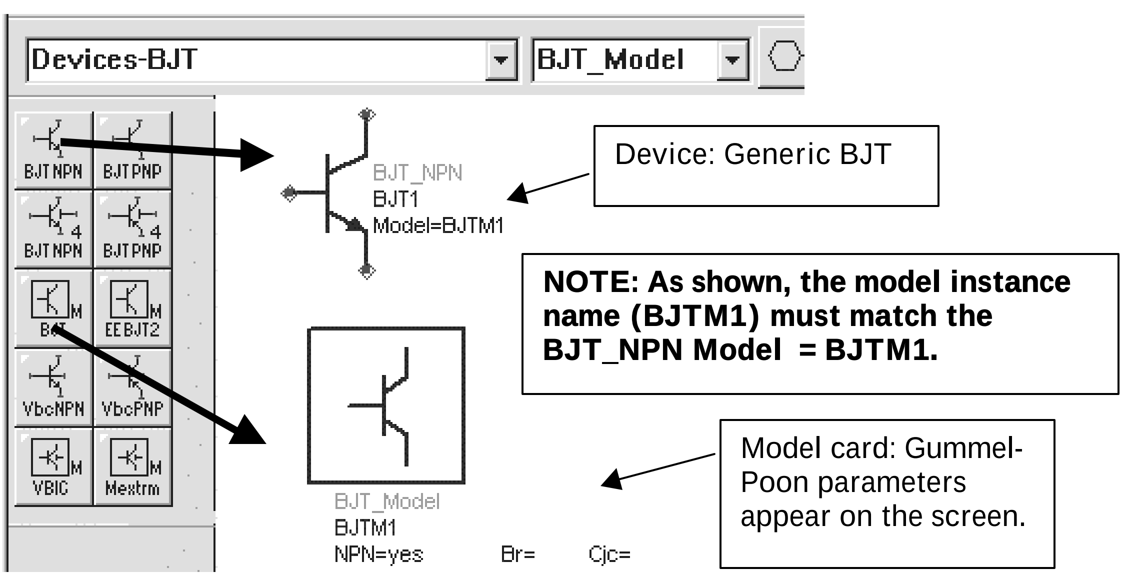
The following steps are for creating the mixer BJT sub-circuit with package parasitics and performing the dc simulations as part of the design process.

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

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

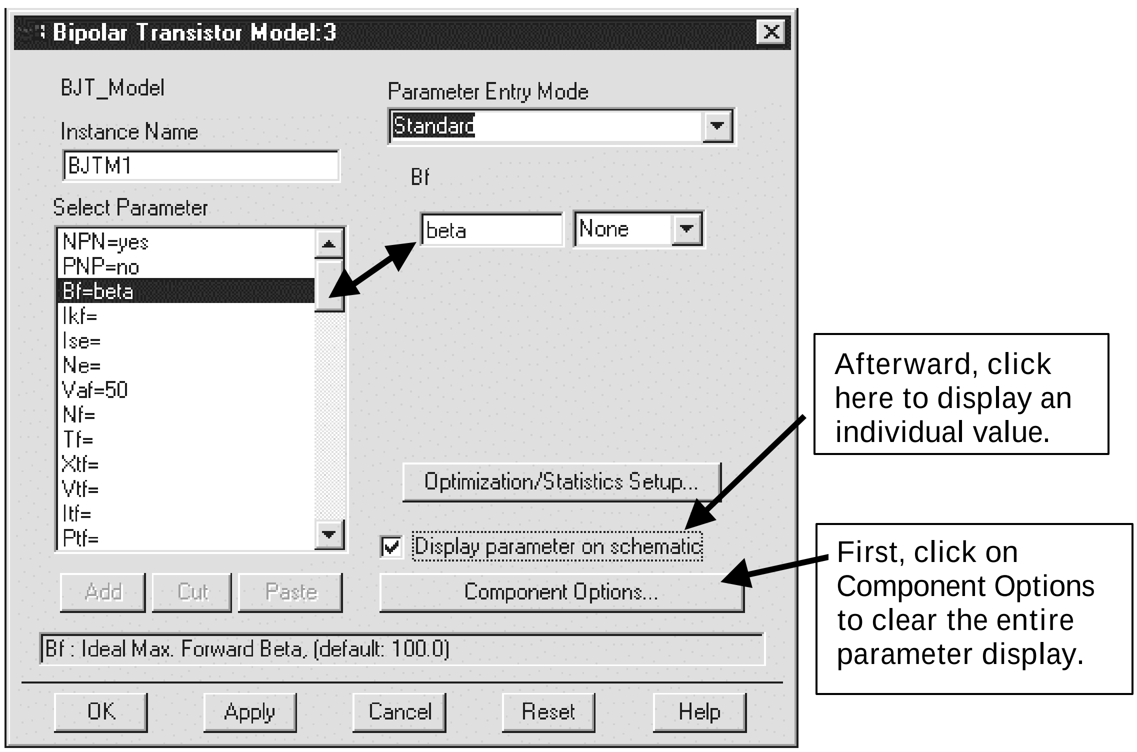
3. **Setup the BJT device and model:**

   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.

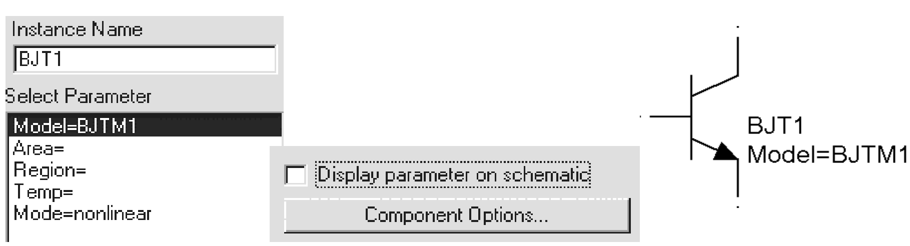
c. Assign Forward Beta = beta. Double click on the model card you just inserted. Select the **Bf** parameter and type in the word **beta** as shown here. Also, click the small box**: Display parameter on schematic** for **Bf** only and then click **Apply**. The numerical value of beta will be assigned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



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.

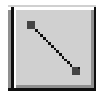
f. For the **BJT device** or any component, you can also remove the unwanted display parameters (Area, Region, Temp and Mode) by editing it in the same way.



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:

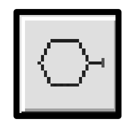
NOTE: Connect the components together or wire them as needed.

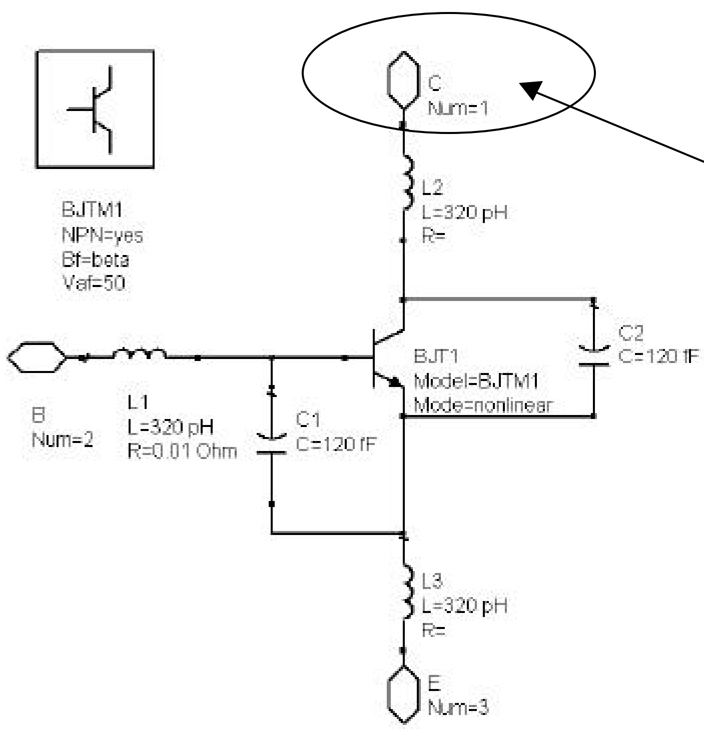


Add Wire icon.

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.





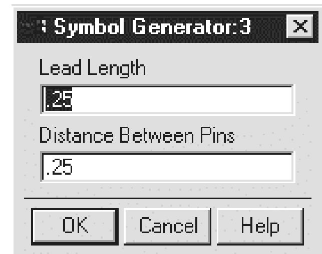
NOTE: You must number the ports (num=) exactly as shown or the device will not have the correct

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

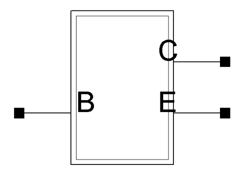
There are three ways to create a symbol for a circuit: 1) Use a default symbol, 2) Use a built-in symbol (a standard symbol), or 3) Create a new symbol by drawing one or modifying an existing one. For this lab you will use a built-in bjt symbol which looks better than the default three-port symbol. The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



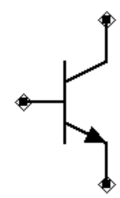
b. Next, a rectangle or square with three ports is generated:

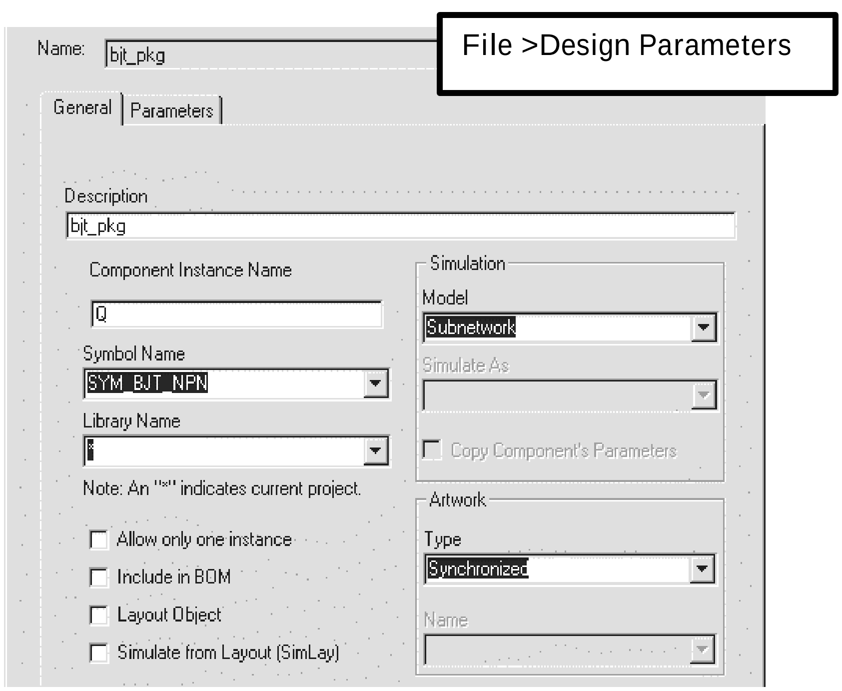
NOTE: You will be replacing the default symbol with a built-in BJT symbol in the next steps. As you do, you must assign the pin (port) numbers exactly as shown to match the built-in symbol for the emitter, base, and collector.



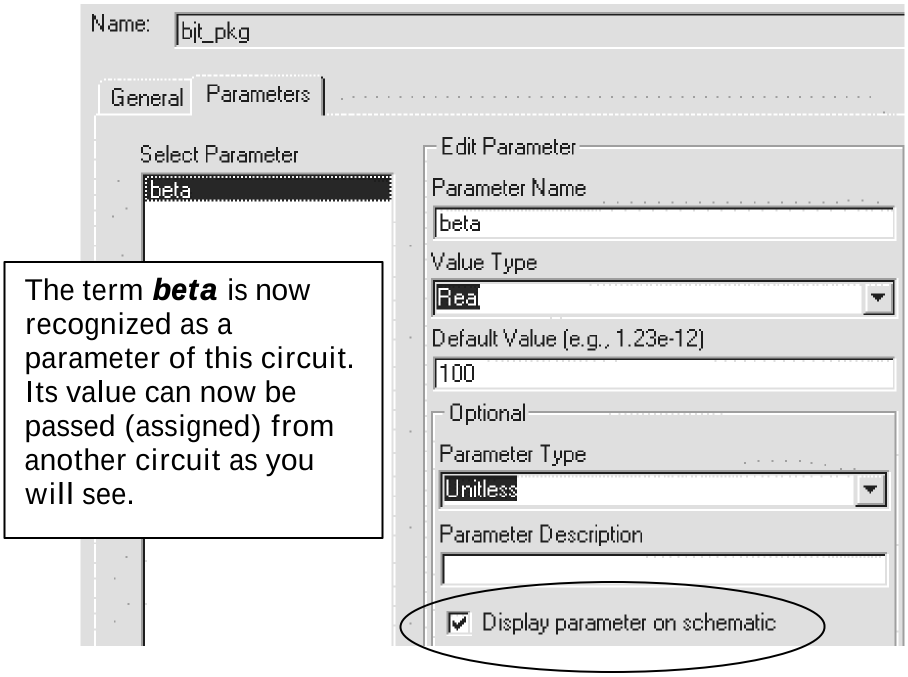
c. To change the symbol to a built-in symbol that looks like a transistor, delete the entire symbol you just created: **Select > Select All**. Then click the **trash can** icon to delete the symbol.

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**.

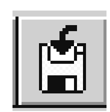




e. Set beta as a pass parameter: To do this, click the **Parameters** tab. In the Parameter Name area, type in **beta** and assign a default value of **100** by clicking the **Add** button. Be sure to click the **Display** button as shown in the picture. Click the **OK** button at the bottom (not shown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



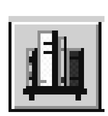
f. In the schematic window, **Save** the design to make sure all your work is save and **close the window**. You now have a sub-circuit that will be available for use in other designs and other projects.

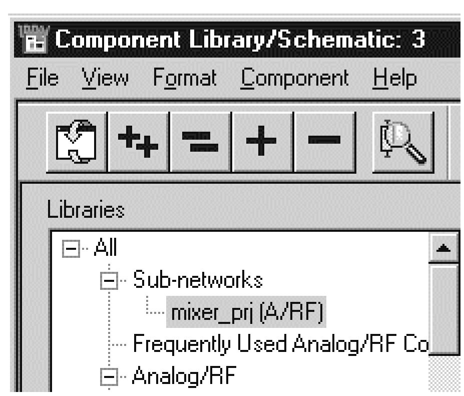


6. Create another circuit for DC simulations

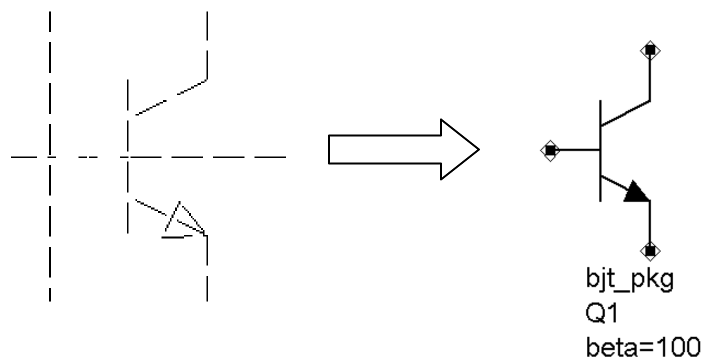
a. Open a new schematic from the Main window and save it as: **dc_curves**. This will be the upper level circuit.

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.





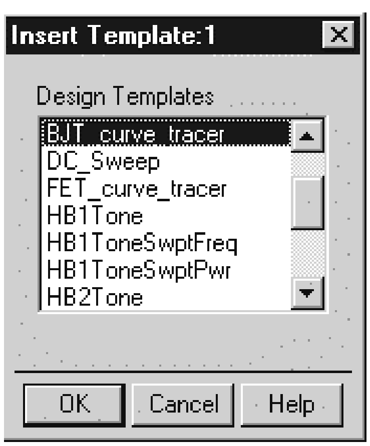
c. Select the **bjt_pkg component** and the npn transistor symbol will be appear on your cursor. Click in the dc_curves schematic to insert the **bjt_pkg**. You can now **close the library window** and save the dc_curves design (good practice to save often).

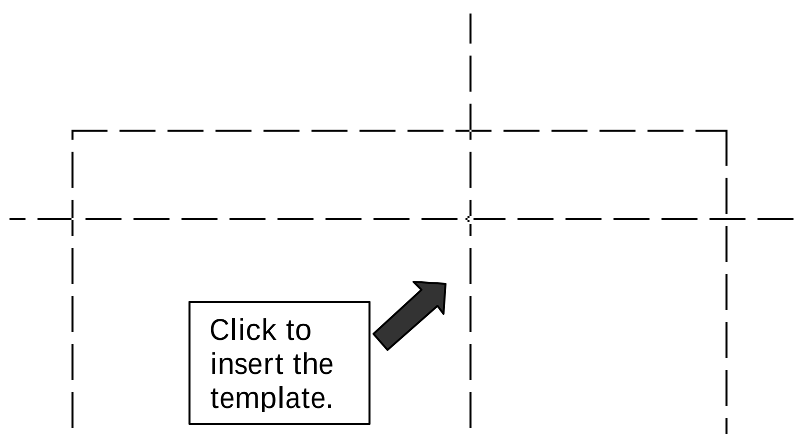


7. Set up a dc curve tracer

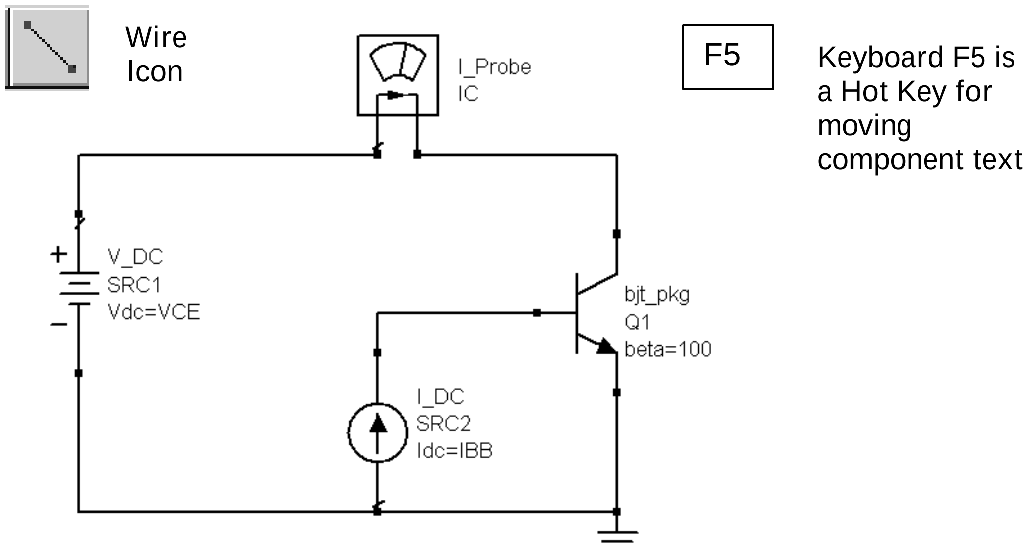
For this step you will use a template. ADS built-in templates make it easier to set up the simulation after the schematic is built. In this case, the dc curve tracer template is set up to sweep VCE within incremental values of base current IBB.

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.



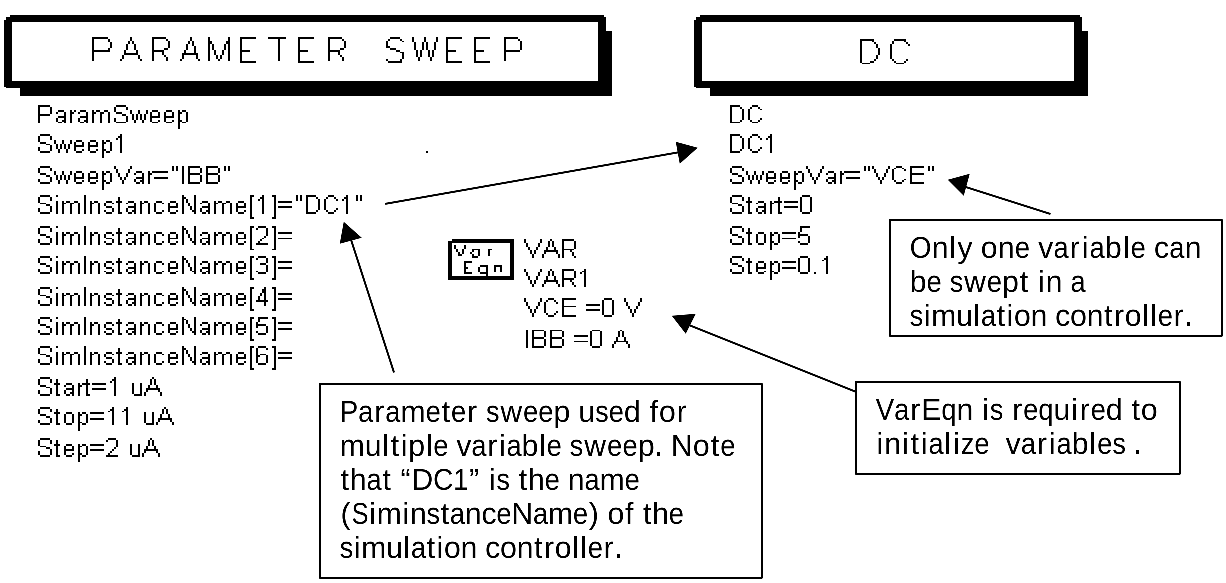


b. With the curve tracer template inserted, **wire the circuit together** so it looks like the shown here. Note that you can move the component text using the F5 key so that the schematic looks good.



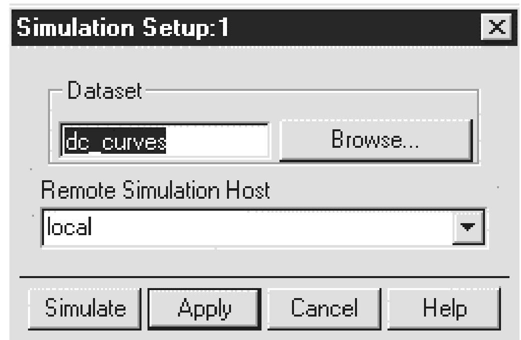
NOTE: If you did not use this Template, you would have to insert every component (the V_DC source, the I_Probe, the I_DC source, etc.) one at a time. Also, you would have to assign and set up the variables (IBB, VCE) for the swept simulation.

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.

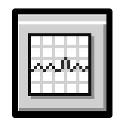


b. Click **Apply** and **Simulate**.

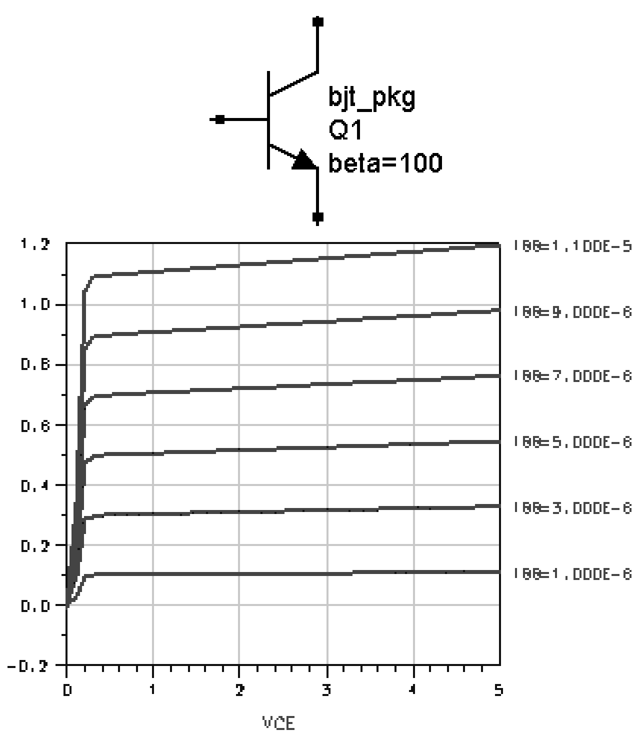
c. After the simulation is finished, click the **Cancel** button and the setup dialog will disappear. If you get an error message, check the simulation set up and repeat if necessary.

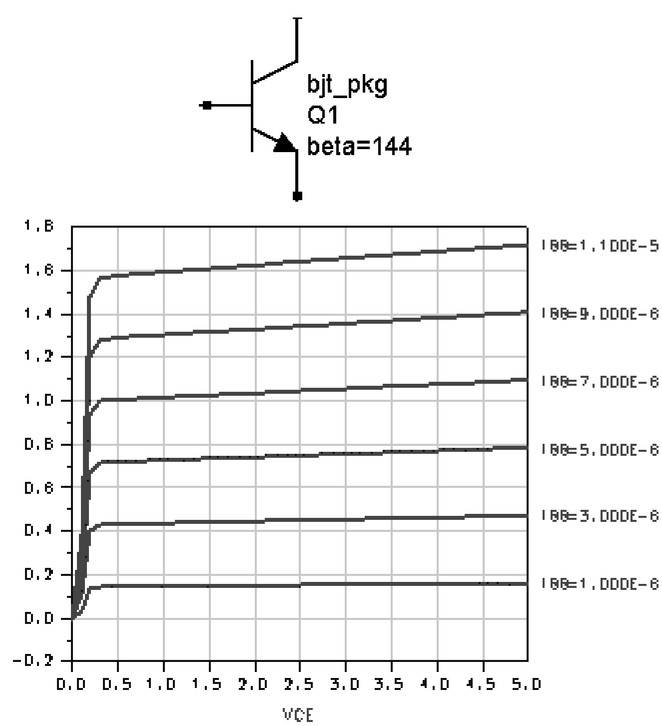
9. Display the results, change beta, and resimulate

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.

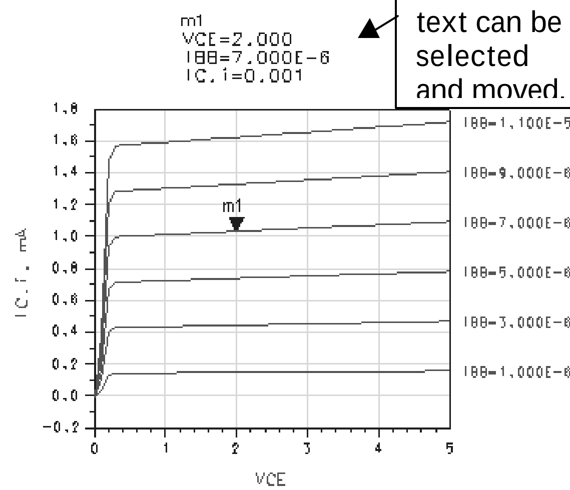


b. On the schematic, change the value of **beta = 144**. The value will automatically be passed down to the sub circuit that you set up in the previous steps. Simulate again and notice the change as shown here. NOTE: **You will use beta =144 for the remainder of the labs**.



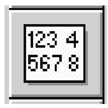


c. **Insert a marker** on the *dc_curves* trace (as shown here), where the initial specification of 1 mA at VCE corresponds to about 7 uA of base current.



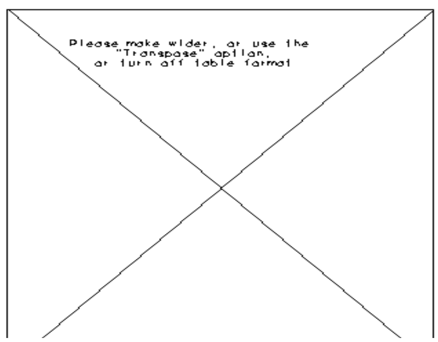
The marker text can be selected and moved.

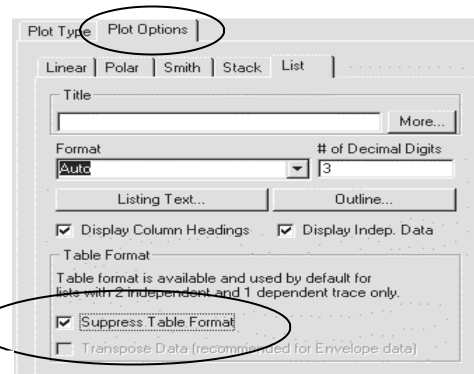
d. **Insert a list** (click the icon).



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.

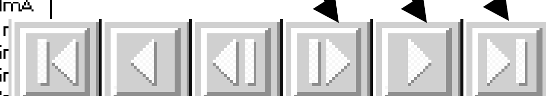




VCE	IC.i
5.000	781.0uA
IBB=7.000E-6	
0.000	-6.952uA
0.100	242.4uA
0.200	939.4uA
0.300	997.3uA
0.400	1.001mA
0.500	1.003mA
0.600	1.005mA
0.700	1.007mA
0.800	1.009mA
0.900	1.011r
1.000	1.013r
1.100	1.015r
1.200	1.017r
1.300	1.019mA
1.400	1.021mA
1.500	1.023mA
1.600	1.025mA
1.700	1.027mA
1.800	1.029mA
1.900	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.

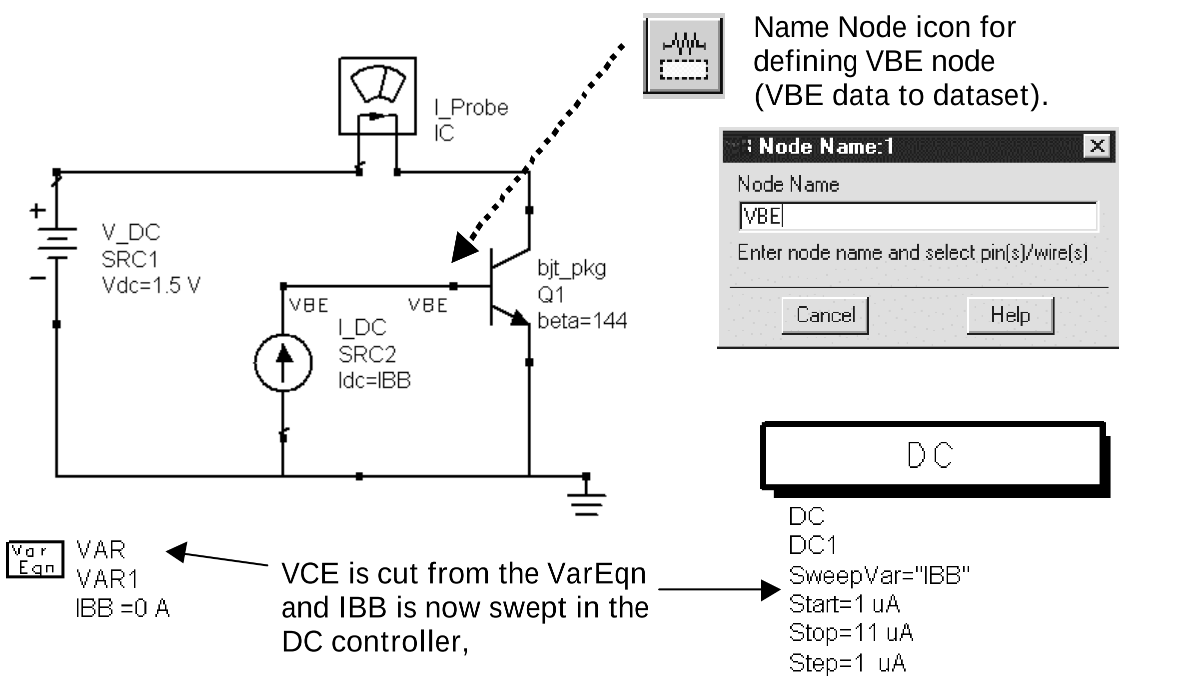
10. Create a new design to calculate bias values

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.

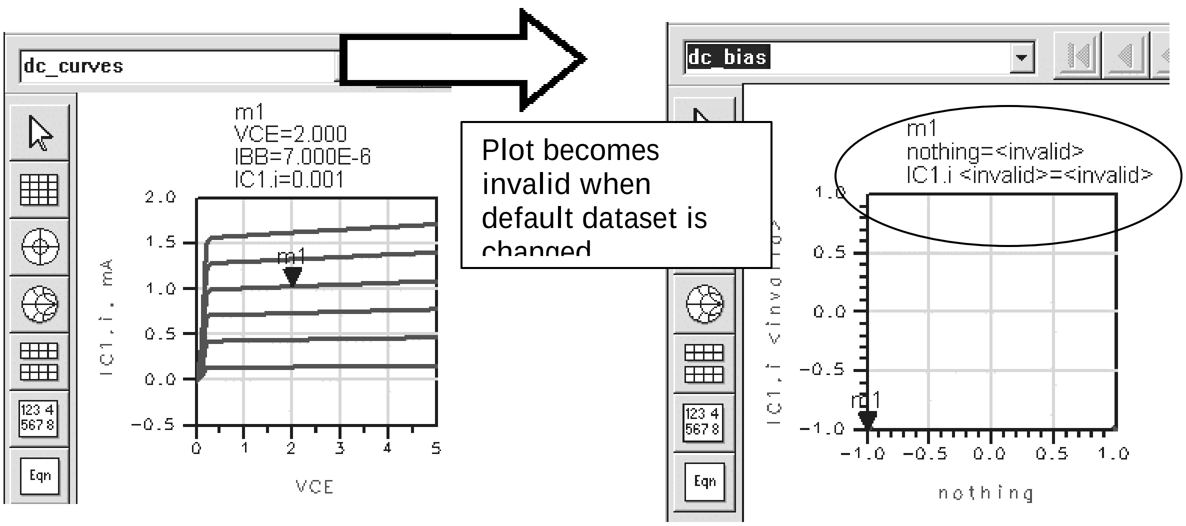


d. **Save and Simulate**: Save the new design by clicking the save icon – this is always good practice. Next, check the dataset name: **Simulation > Setup**) as in the previous simulation. Be sure it appears as: **dc_bias** and then **Simulate**.

11. Display the data (dc_bias) in a list

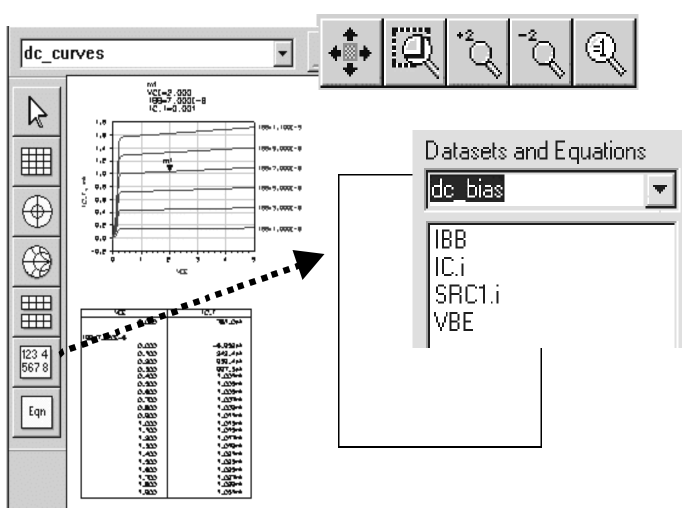
In this step, you will use the same data display window that contains the dc_curves data. In fact, you can plot numerous datasets in the display but you must explicitly define (dataset name..) the data to be displayed.

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**.

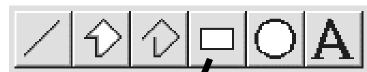


b. Now, in the current Data Display window, make room for the new data by using the **zoom and view icons**. Then **insert a new list**.

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.



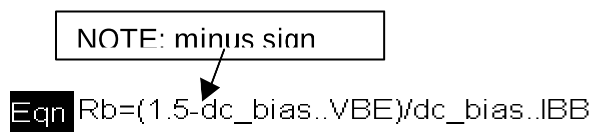
d. **Draw a box around the values** of interest as shown here. To do this, click the **rectangle icon** from the tool bar and draw it on the list. This is one way to highlight the data. Also, the data display window by using Save As and giving it a name like: dc_data.



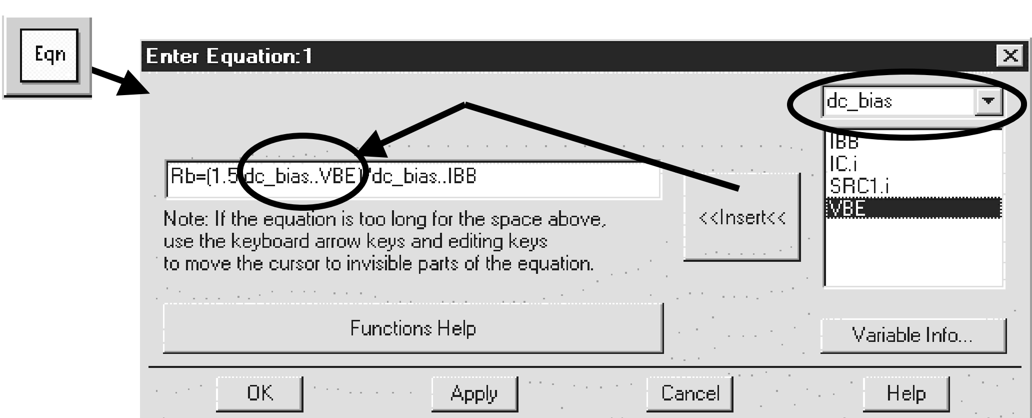
IBB	dc_bias..IC.i	dc_bias..VBE
1.000E-6	146.2uA	719.3mV
2.000E-6	292.4uA	737.1mV
3.000E-6	438.5uA	747.5mV
4.000E-6	584.6uA	754.9mV
5.000E-6	730.6uA	760.6mV
6.000E-6	876.7uA	765.3mV
7.000E-6	1.023mA	769.3mV
8.000E-6	1.169mA	772.7mV
9.000E-6	1.315mA	775.7mV
1.000E-5	1.461mA	778.4mV
1.100E-5	1.607mA	780.9mV

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:



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.

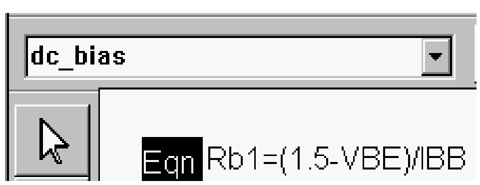
Eqn Rb1=(1.5-VBE)/IBB

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.



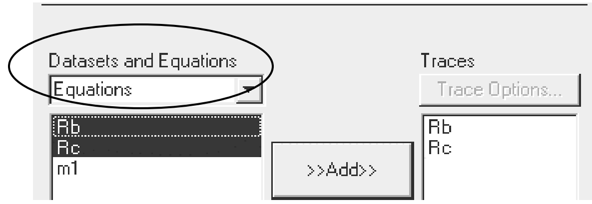
Equation is black (valid) because the default dataset corresponds to the equation..

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**.



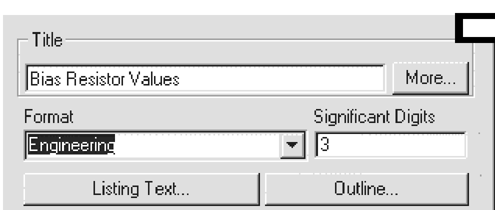
g. When the list appears, you will then see a table of values for Rb and Rc that correspond to the value of IBB. As a rule, you always get the independent variable (here IBB) when you list or plot data.

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.

IBB	Rb	Rc
1....	78...	33...
2....	38...	16...
3....	25...	11...
4....	18...	84...
5....	14...	67...
6....	12...	56...

IBB	Rb	Rc
1.000E-6	780714.794	3395.621
2.000E-6	381452.969	1698.402
3.000E-6	250829.474	1132.499
4.000E-6	186274.268	849.497
5.000E-6	147872.779	679.674
6.000E-6	122446.591	566.447
7.000E-6	104388.426	485.564
8.000E-6	90911.023	424.897
9.000E-6	80473.556	377.708
1.000E-5	72155.500	339.955
1.100E-5	65373.292	309.065

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

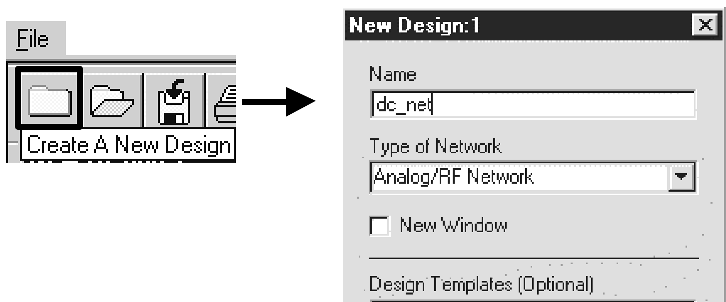
IBB	Rb	Rc
1.000u	780.7k	3.396k
2.000u	381.5k	1.698k
3.000u	250.8k	1.132k
4.000u	186.3k	849.5
5.000u	147.9k	679.7
6.000u	122.4k	566.4
7.000u	104.4k	485.6
8.000u	90.91k	424.9
9.000u	80.47k	377.7
10.00u	72.16k	340.0
11.00u	65.37k	309.1

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.

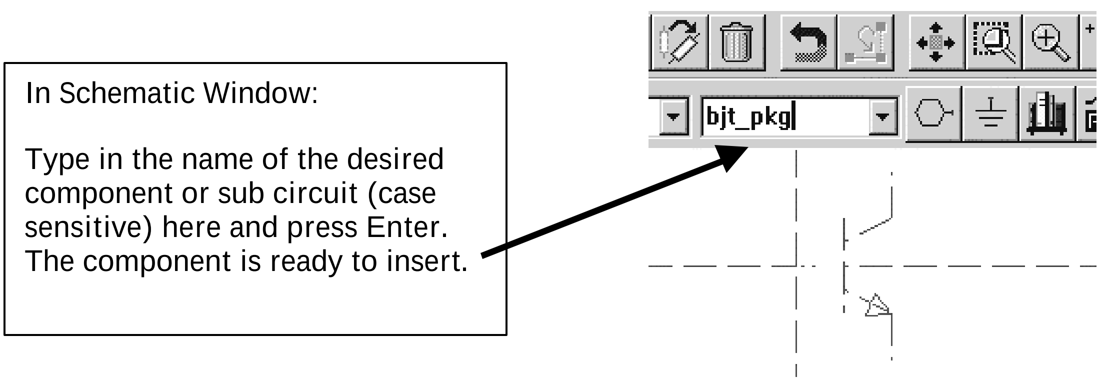
13. Set up a new design to test the bias network

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.

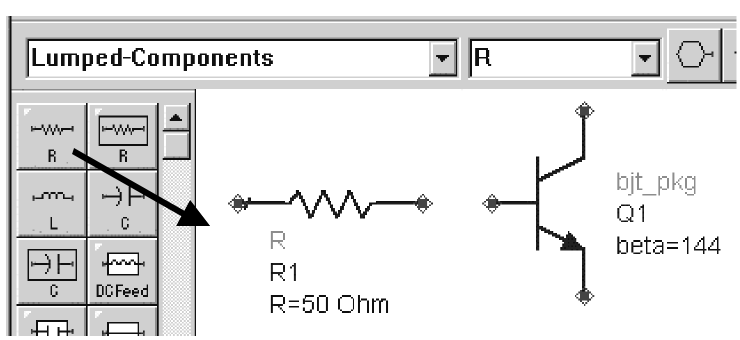


b. In the new schematic (dc_net), insert your sub circuit **bjt_pkg** by typing in the name in the component history list:

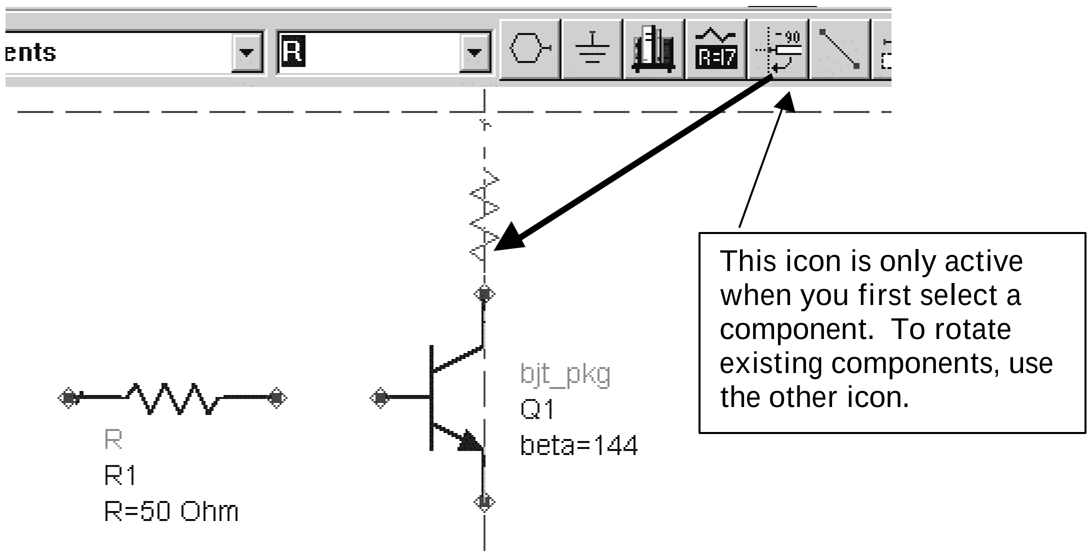


c. Set the value of **beta** to: **144**

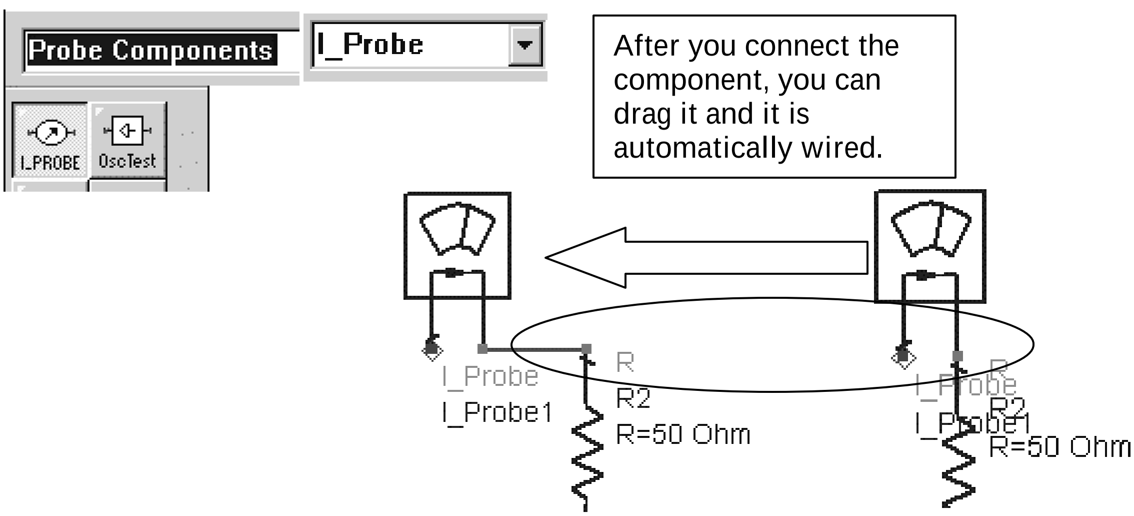
d. Goto **Lumped Components** palette and insert a **resistor** as the base resistor. Notice that "R" appears in the history list when you do this.



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.

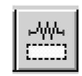


f. Insert a **current probe (I_Probe)** from the palette or type it in. Connect it to the top of the collector resistor.

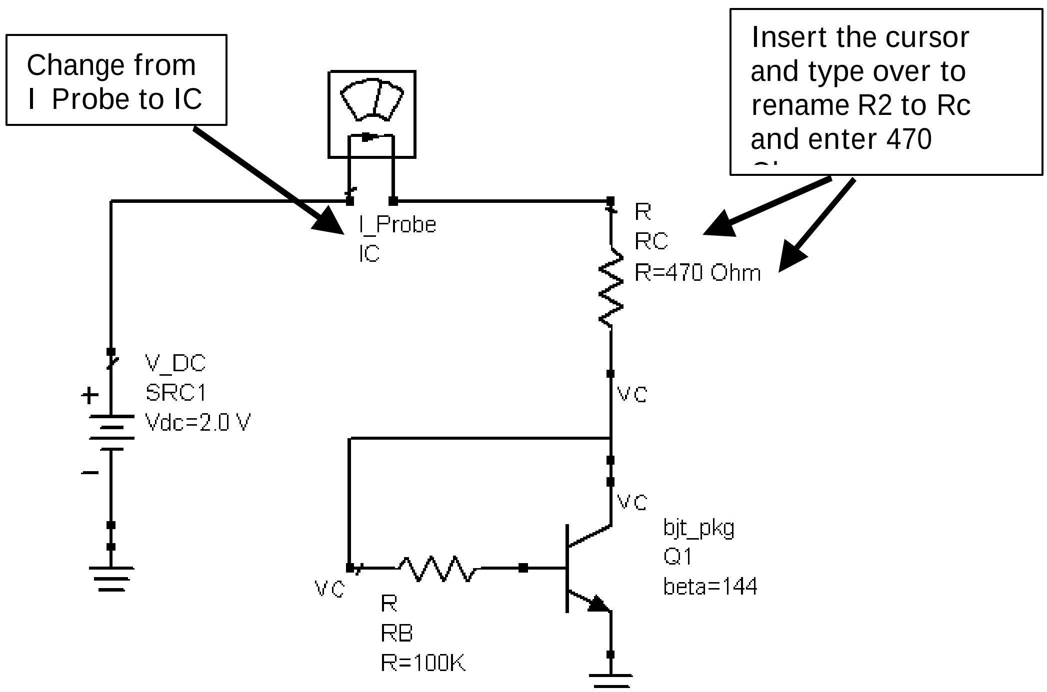


g. **Finish building the circuit as follows**:

- 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**.

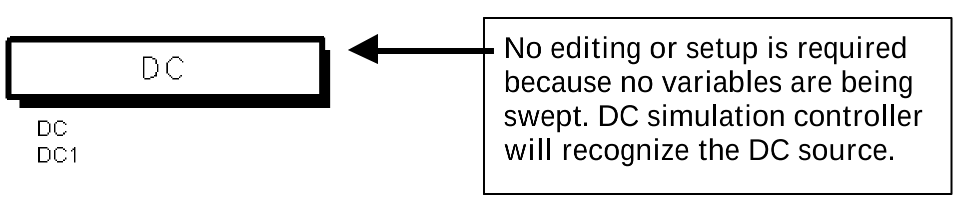


- Wire the circuit and organize it.



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**.**

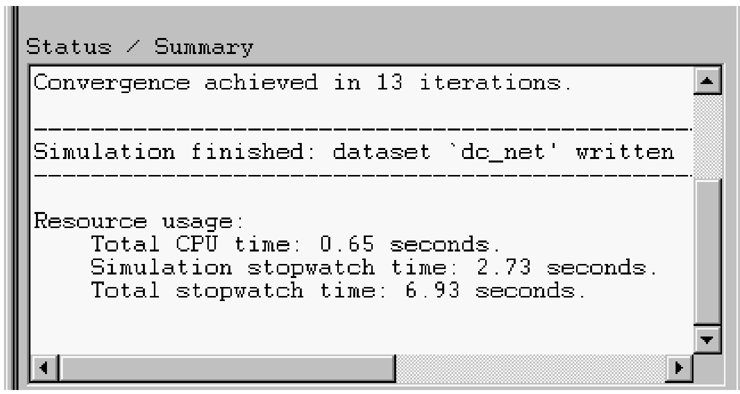
h. **Insert a DC simulation controller** (Simulation-DC palette).



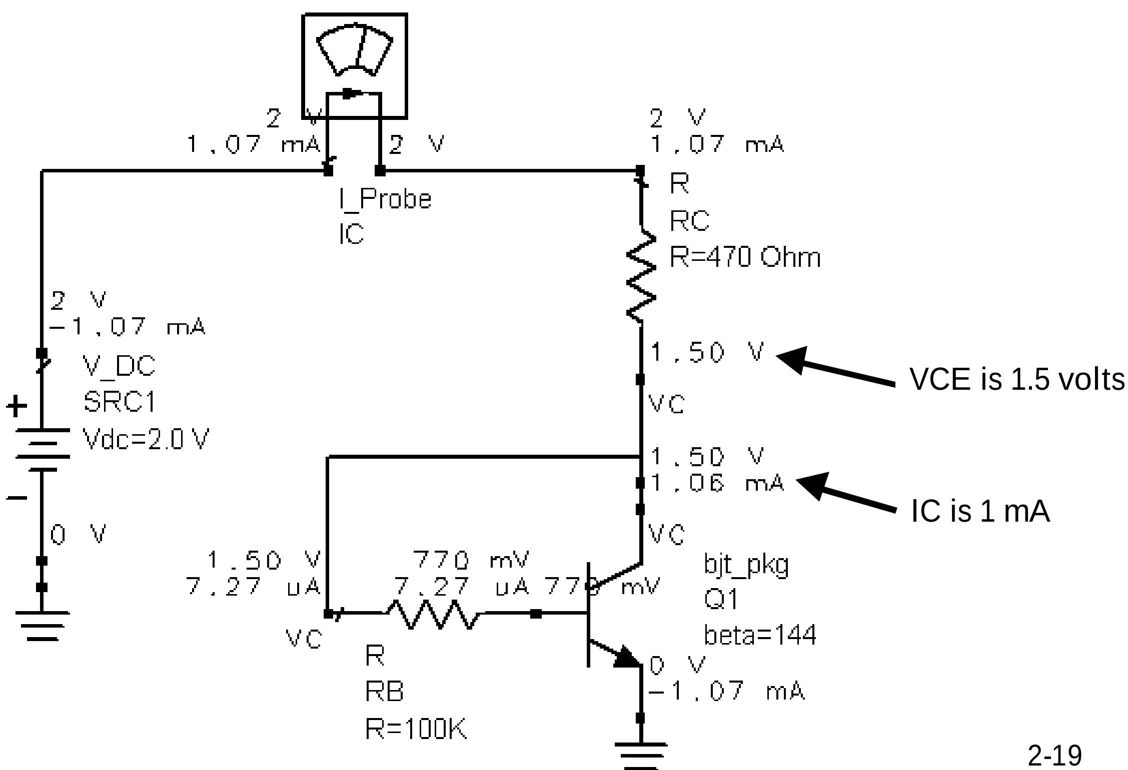
14. Simulate and verify the bias network conditions

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:

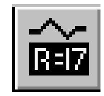


b. Annotate the current and voltage at the nodes. Click on the menu command: **Simulate > Annotate DC Solution**. Now you should see the voltage and currents at the nodes. Be sure that you have about 1 mA of collector current with VCE about 1.5 V. If not, check your work.

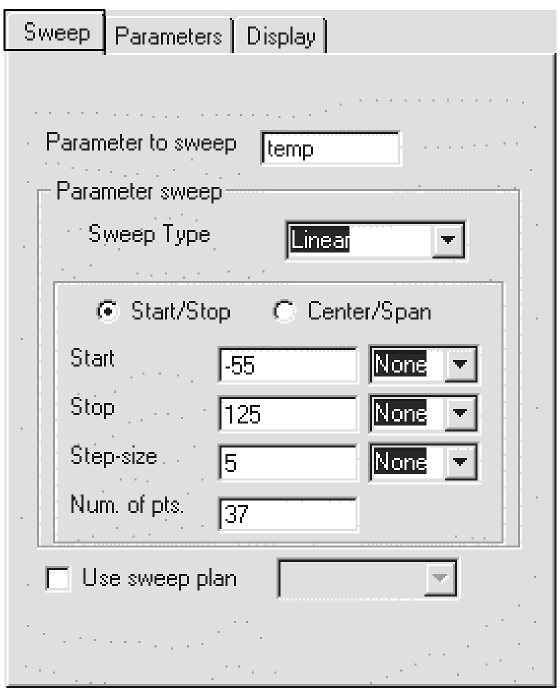


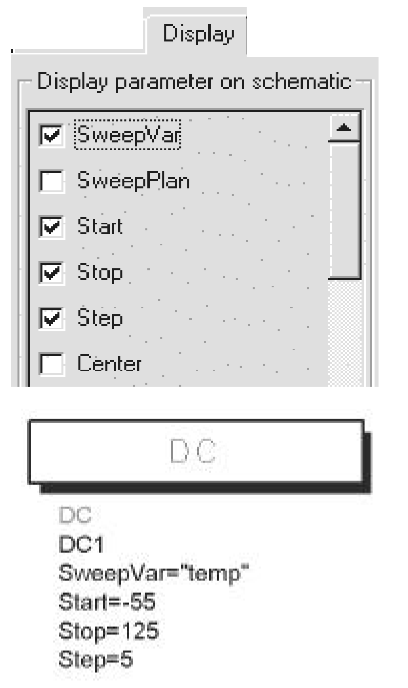
15. Sweep Temperature

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

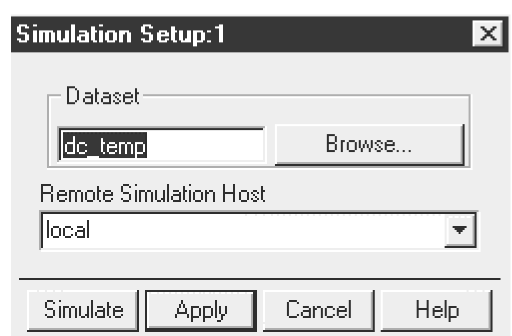


b. In the **Sweep tab**, enter the ADS global variable **temp** (default is Celsius) as shown here and enter the sweep range: **-55 to 125 @ 5 step**. Also, in the Display tab, click the boxes to display the annotation on the controller – click **Apply** to see it and **OK** to dismiss the dialog.

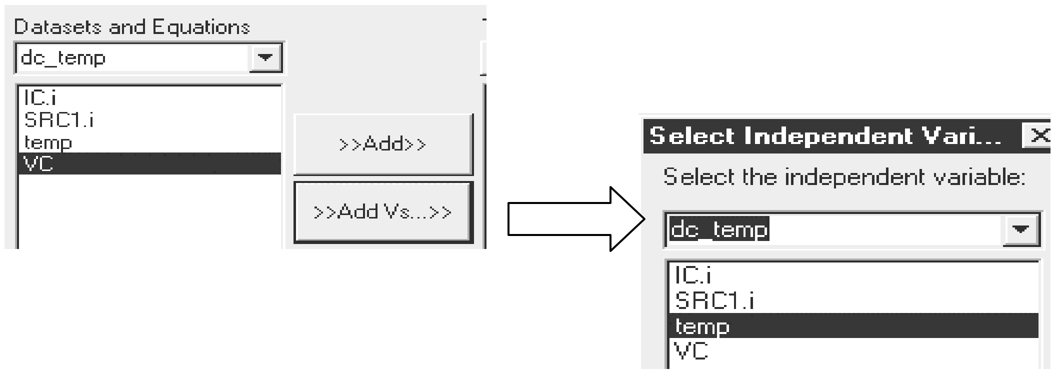




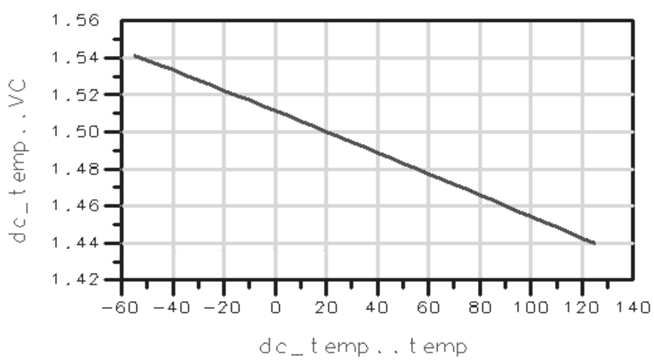
c. Set the simulation dataset name to **dc_temp**, click Apply to assign that dataset name, and then **Simulate**.



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.

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

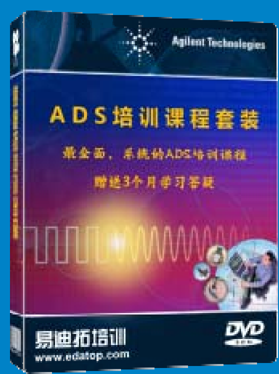
射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

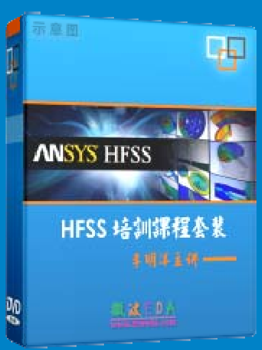
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...



课程网址： http://www.edatop.com/peixun/ads/13.html



HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的HFSS培训教程套装，可以帮助您从零开始，全面深入学习HFSS的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com