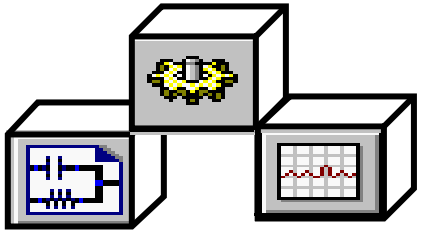
Topic 3:

DC Simulations and sub-circuit modeling



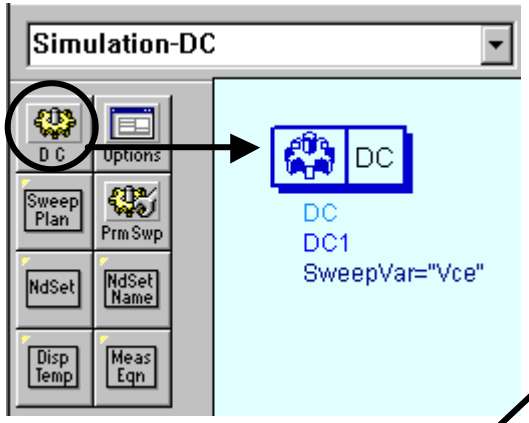
DC Simulation

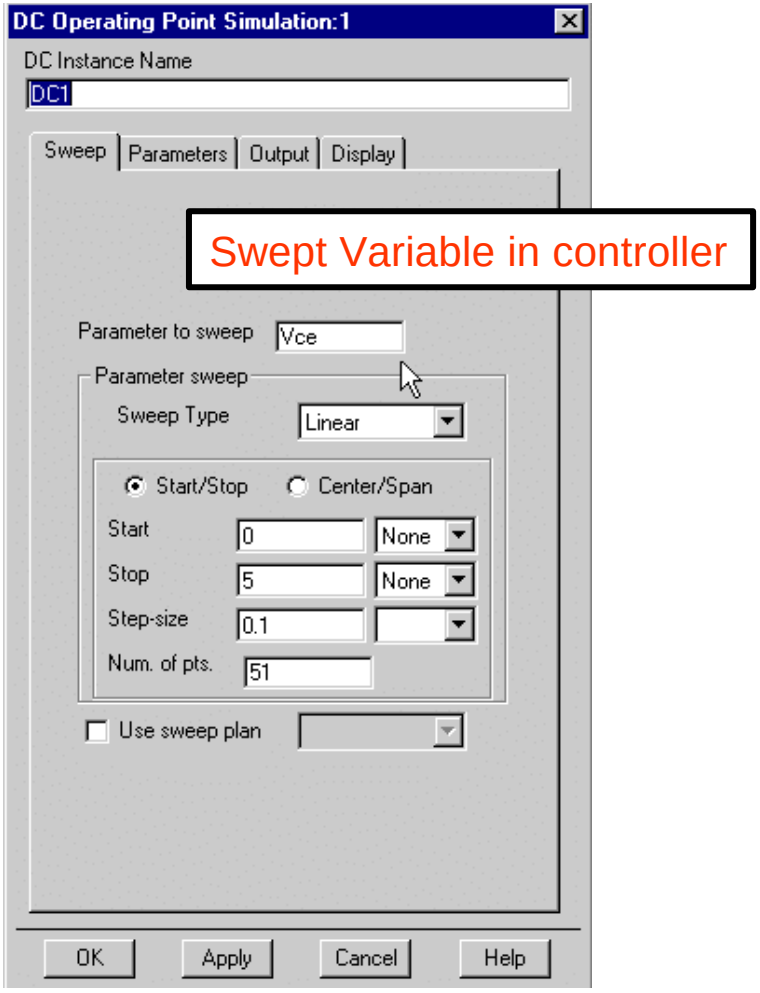
You get steady-state DC voltages and currents according to Ohm's Law: V= IR

- Capacitors = treated as ideal open circuits
- Inductors = treated as ideal short circuits
- Topology check: dc path to ground (if not => error message)
- Kirchoff's Law satisfied: sum of node current = 0
- Convergence simulator algorithms (modes) can be set

DC simulation controller

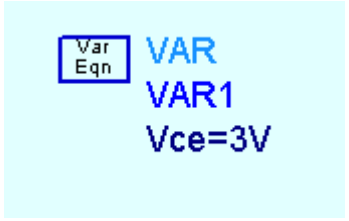
Palette and editor (dialog box)



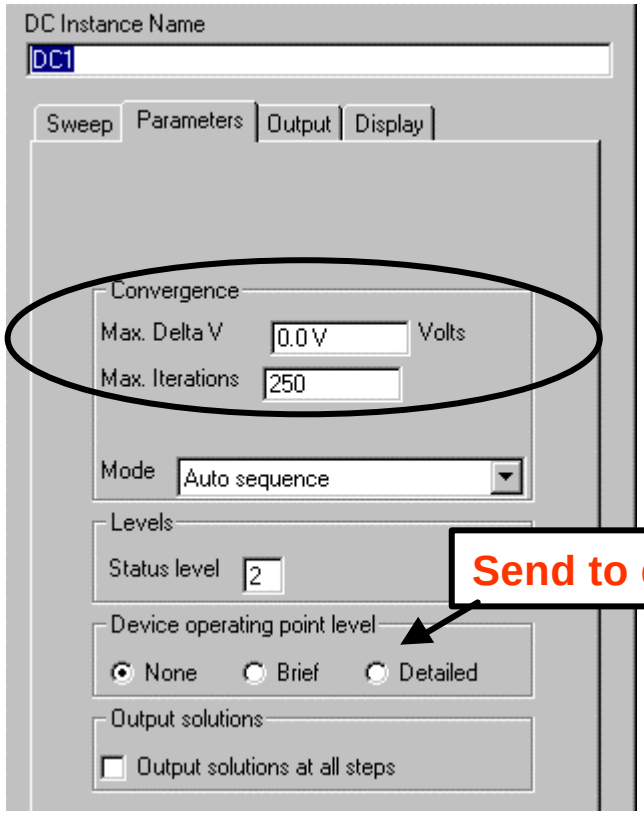


Sweep: allows you to sweep a parameter but it must be declared as a variable. Note the dialog entry automatically puts quotes on the controller (screen) entry.

VAR ⇨



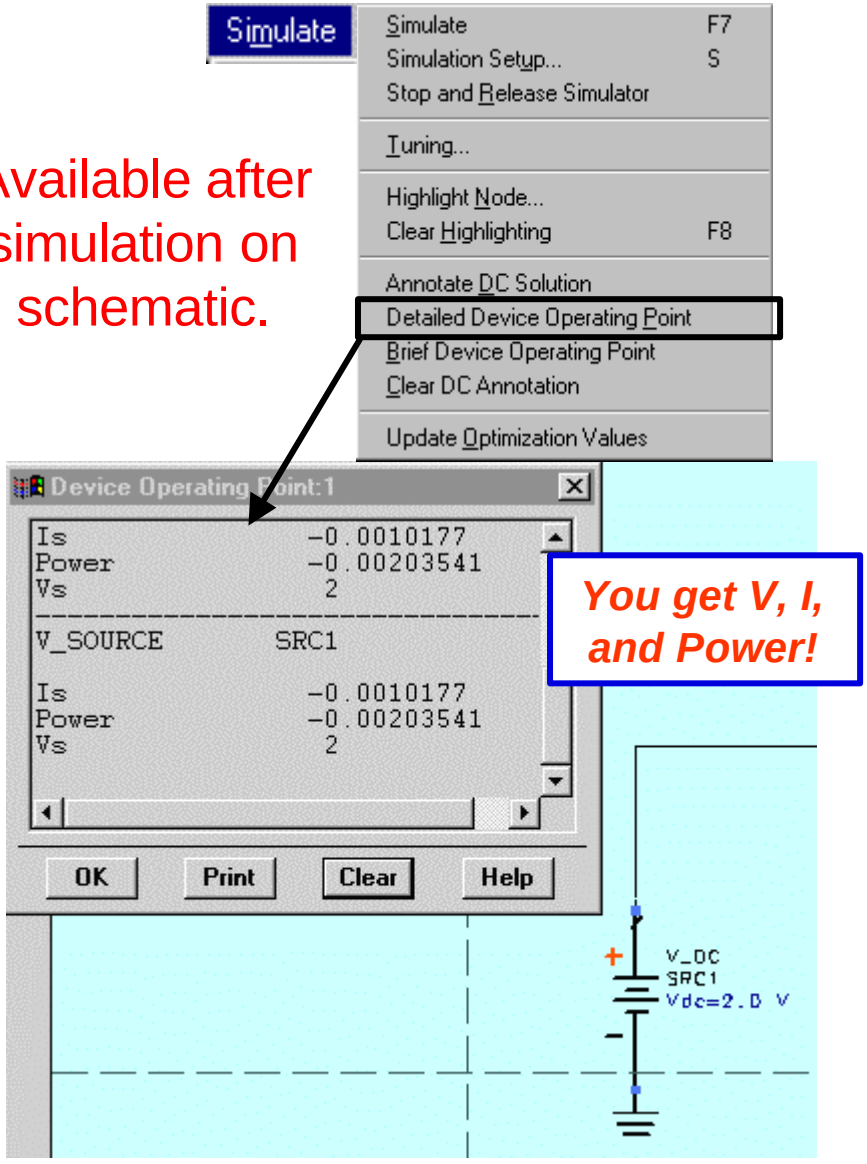
...more on DC



Send to dataset.

Convergence: increase V or iterations or change mode if you don't converge.

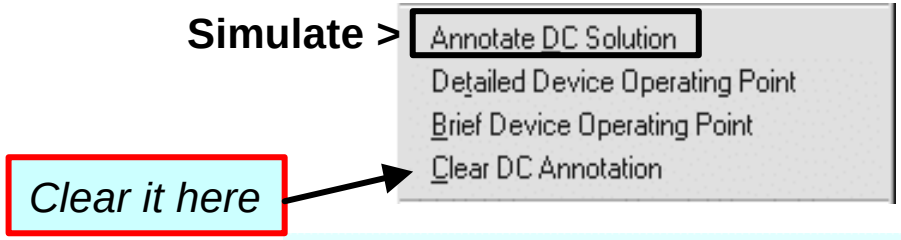
Available after simulation on schematic.

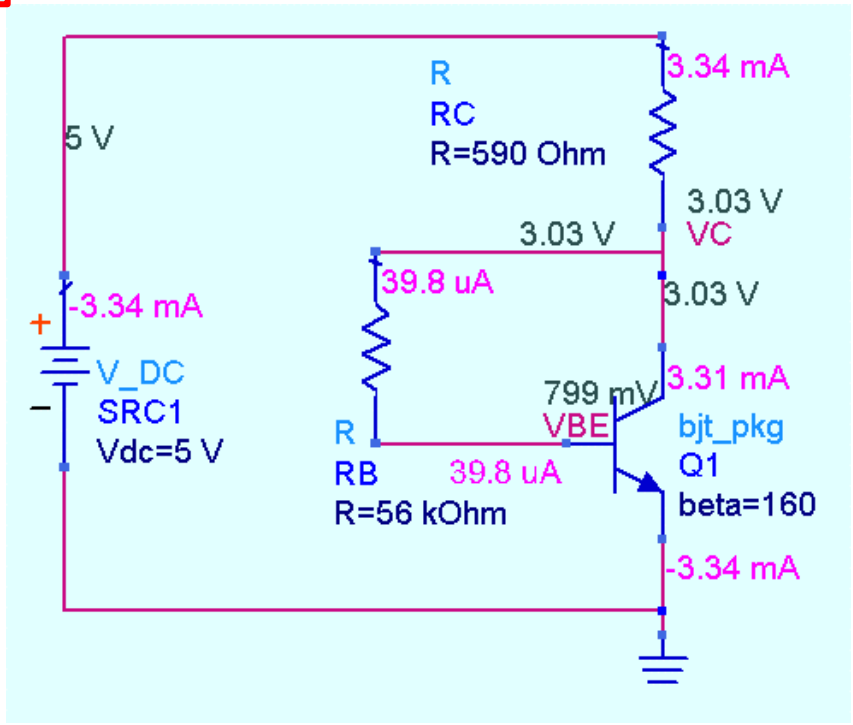


You get V, I, and Power!

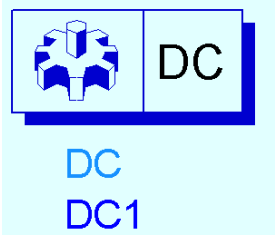
Schematic Annotation of DC values

Immediately after DC simulation, click: **Simulate > Annotate DC Solution**.





No controller settings necessary!



Minus sign used for current flowing out of a connection. Otherwise, current flows into a connection or device.

DC Simulation Controller is required in all simulations if you want DC annotation.

Class Exercise

Set these before you start the next exercise!

HOT KEYS and Schematic Preferences

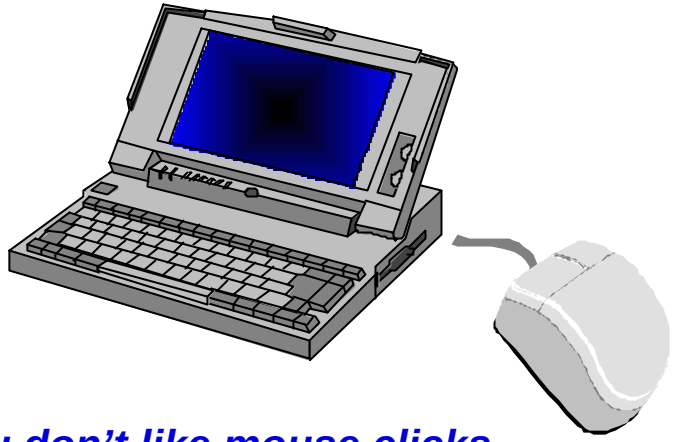
Default ***Hot Keys*** for commands

Pre-configured keys:

F7 = Simulate
F5 = Move Component Text

Ctrl+R = Rotate 90
Ctrl+M = Move
Ctrl+C = Copy
Ctrl+Z = Undo

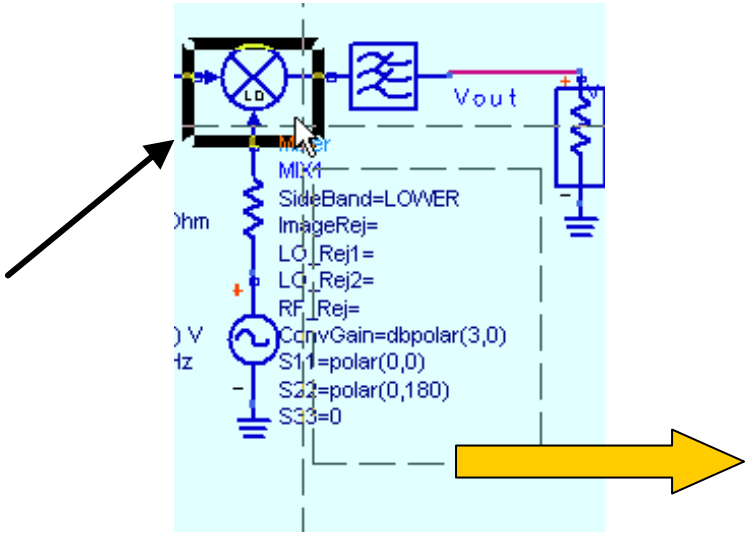
plus more...



If you don't like mouse clicks, HOT KEY your keyboard. Its global for all projects

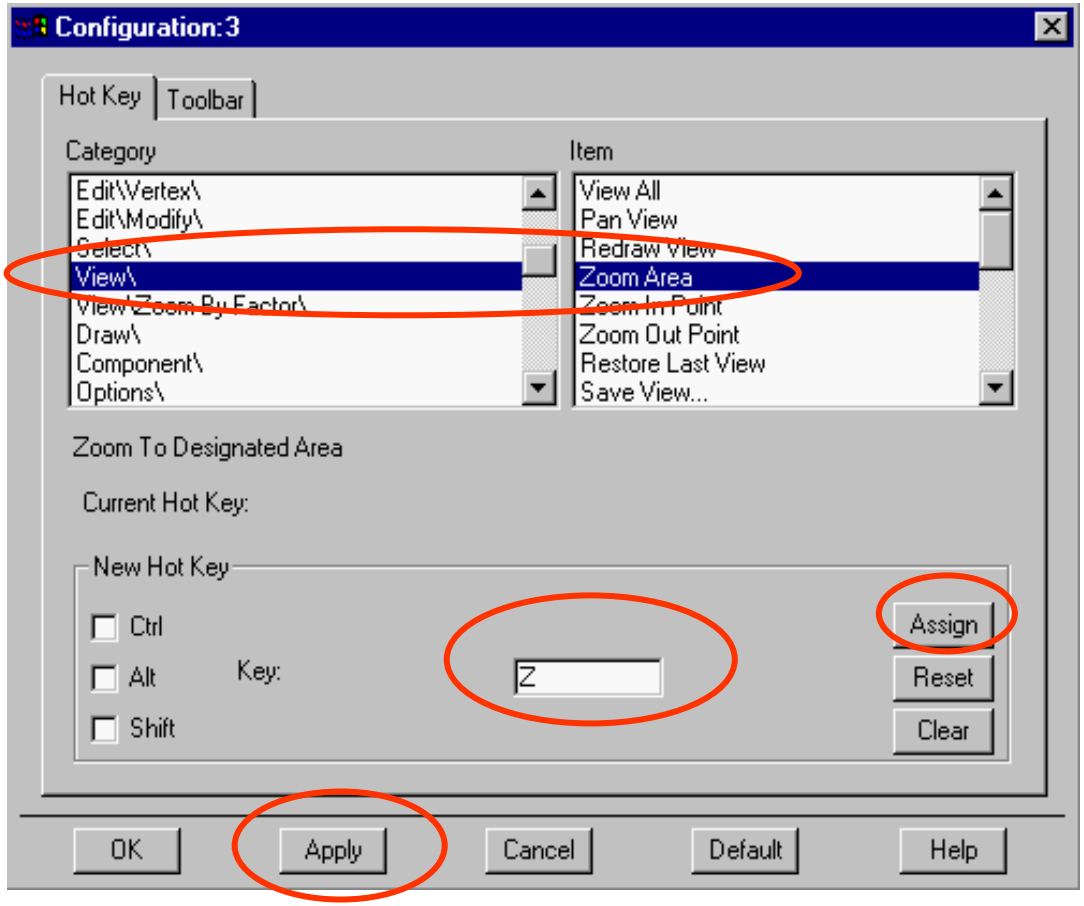
Try this now: *click the* **F5** *key, select the* Mixer component, *move the cursor and the text will follow!*

Hot Keys are global for all projects!



Set your own Hot Keys

Now, click: **Options > Hot Key / Toolbar Configurations...**



Follow these steps to set Zoom Area command:

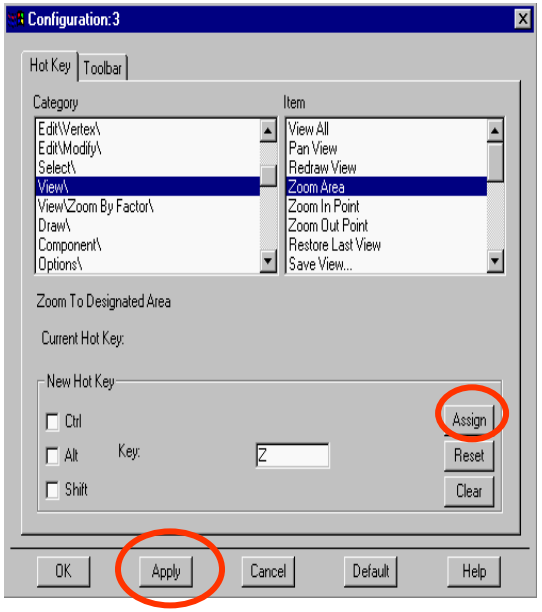
1. Select the command
2. Type in a letter: z (not case sensitive)
3. Click: **Assign**
4. Click: **Apply**
5. Now, try the Z hot key to verify it works.

Set a few more Hot Keys

Options > Hot Key / Toolbar Configurations...

S = Simulate > Setup
A = Activate
D = Deactivate
X = Edit > Move > Move & Disconnect

and any others you want ...

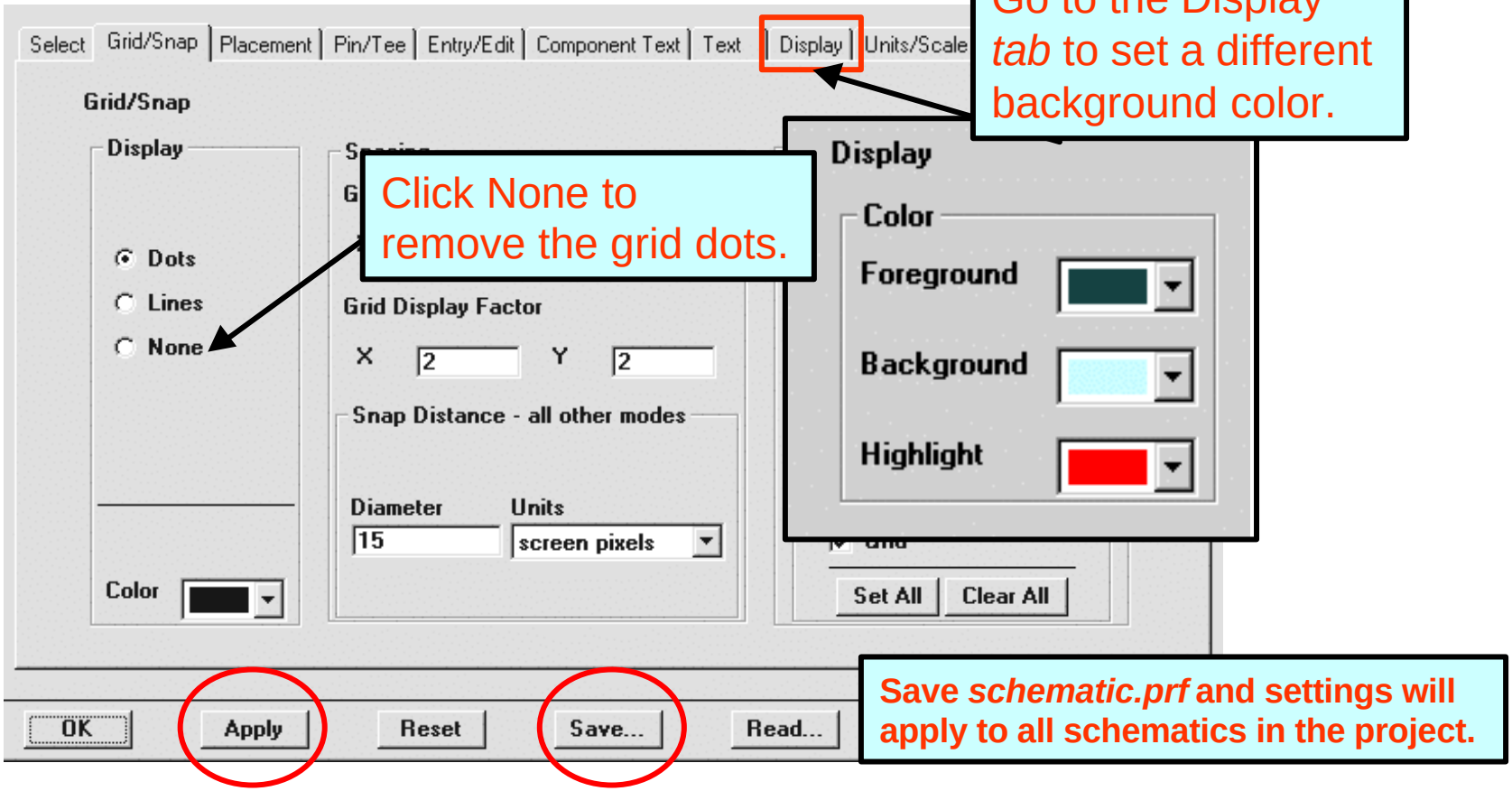


You will be able to use these hotkeys for all the labs in this project.

When everyone has finished, continue

If desired, set *Schematic Preferences*

Click: Options > Preferences



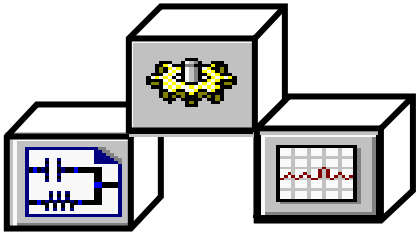
NOTE: Set wire color in: Options > Layers.

End of class exercise.

What the lab is about ...

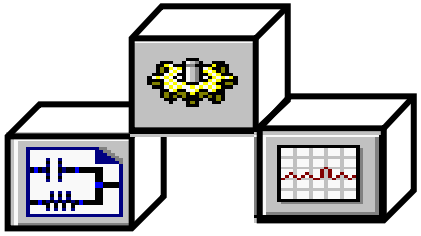
Lab 3:

DC Simulations and modeling the sub-circuit



Steps in the Design Process

You are here:



- *Design the rf_sys behavioral model receiver*
- *Test conversion, budget gain, spectrum, etc.*
- ***Start amp_1900 design – subckt parasitics***
- ***Simulate amp DC conditions & bias network***
- *Simulate amp AC response - verify gain*
- *Test amp noise contributions – tune parameters*
- *Simulate amp S-parameter response*
- *Define amp matching topology and tune input*
- *Optimize the amp in & out matching networks*
- *Filter design – lumped 200MHz LPF - use E-Syn*
- *Filter design – microstrip 1900 MHz BPF*
- *Transient and Momentum filter analysis*
- *Amp spectrum, delivered power, Zin - HB*
- *Test amp comp, distortion, two-tone, TOI*
- *CE basics for spectrum and baseband*
- *CE for amp_1900 with GSM source*
- *Replace amp and filters in rf_sys receiver*
- *Test conversion gain, NF, swept LO power*
- *Final CDMA system test CE with fancy DDS*
- *Co-simulation of behavioral system*

Start with some specifications...

***AMP** with max gain & low noise:*

Available voltage: 5 volts
Device: Generic BJT (Gummel-Poon)
Collector current: about 3.25 mA
Frequency: RF = 1900 MHz
Gain: > 15 dB (or much more with this model)
50 ohm match: input and output

Later labs: matching and testing the AMP for TOI, distortion, noise, compression, GSM & CDMA modulation response, and more.

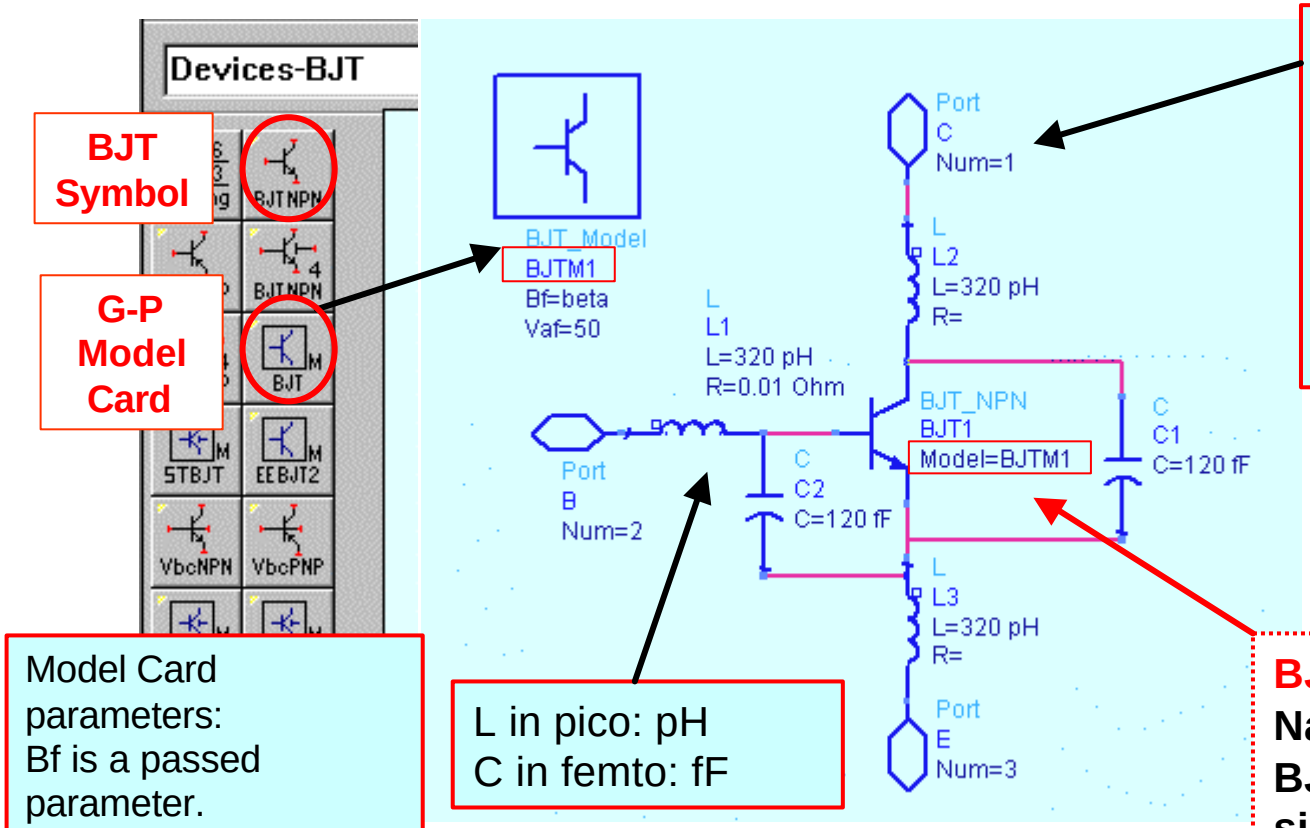
Filters: also, build 1900 MHz BPF for the input and a LPF for the IF output

***YOUR JOB**: Build, test, and refine the circuits to meet specifications.*

Start by modeling the sub-circuit...

Device with package parasitics

G-P Model Card, BJT symbol, parasitics, and ports.



BJT Symbol

G-P Model Card

Port connector numbers: **Num=** must be set in specific order as shown - this is necessary to correctly use BJT symbol.

Model Card parameters:
Bf is a passed parameter.
Vaf is changed as shown.

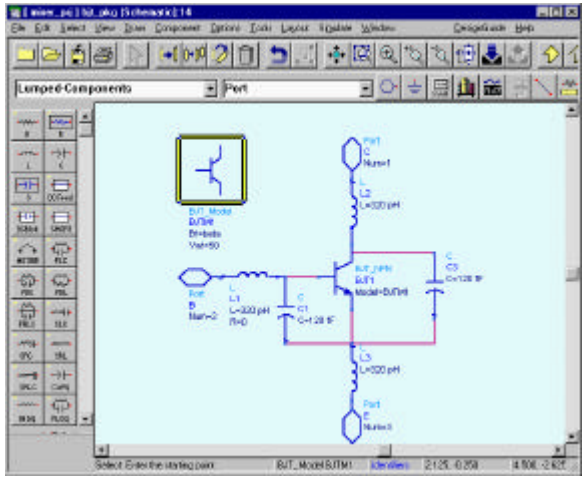
L in pico: pH
C in femto: fF

BJT Symbol:
Name of model card BJTM1 used for simulation. Library devices do not require this mapping.

Viewing and creating a schematic symbol

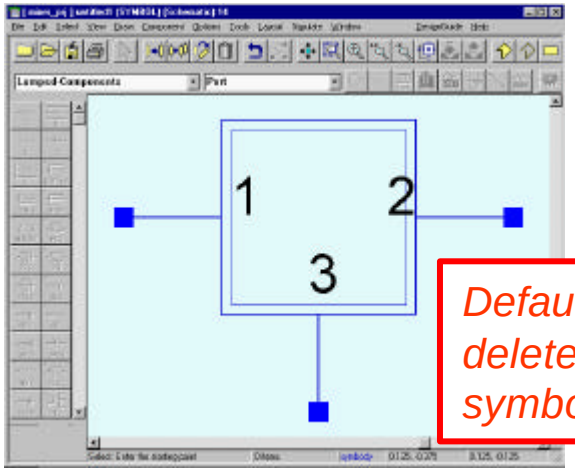
Schematic view

View → Create/Edit Schematic Symbol

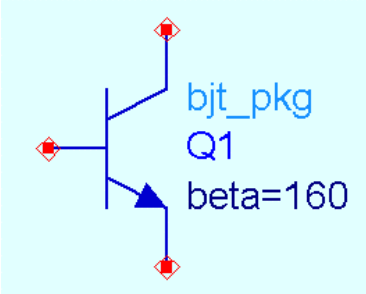


Symbol view

View → Create/Edit Schematic



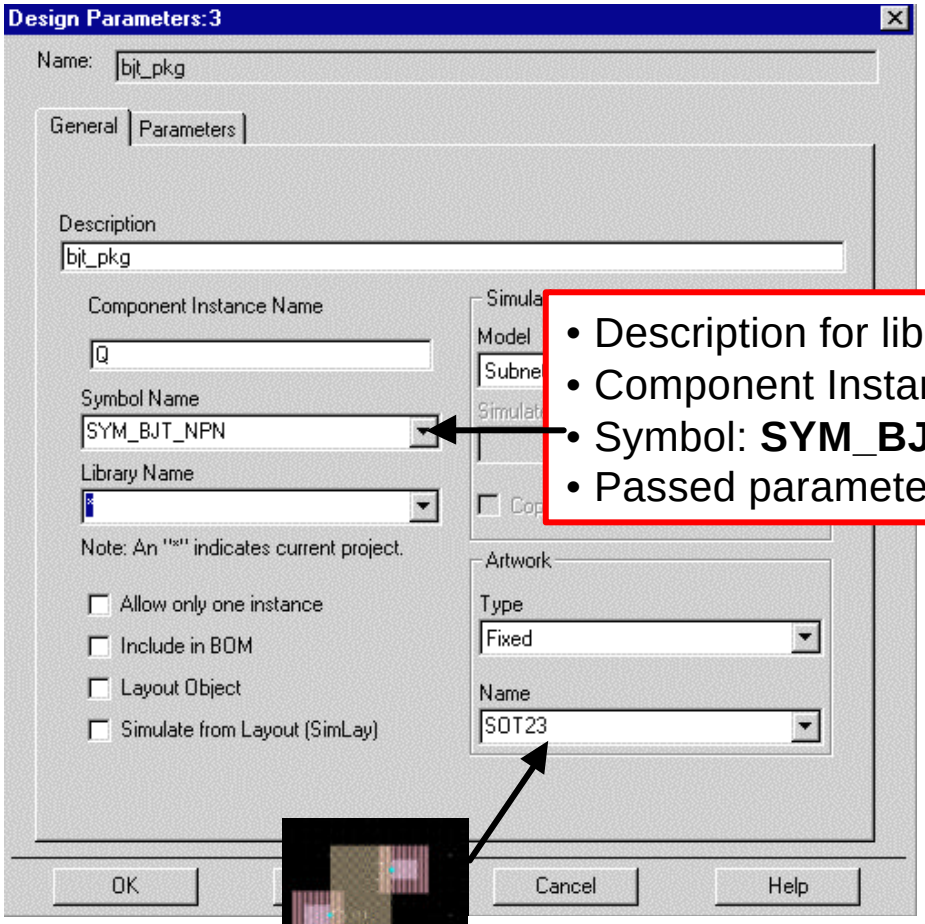
Default Symbol or delete and draw a symbol.



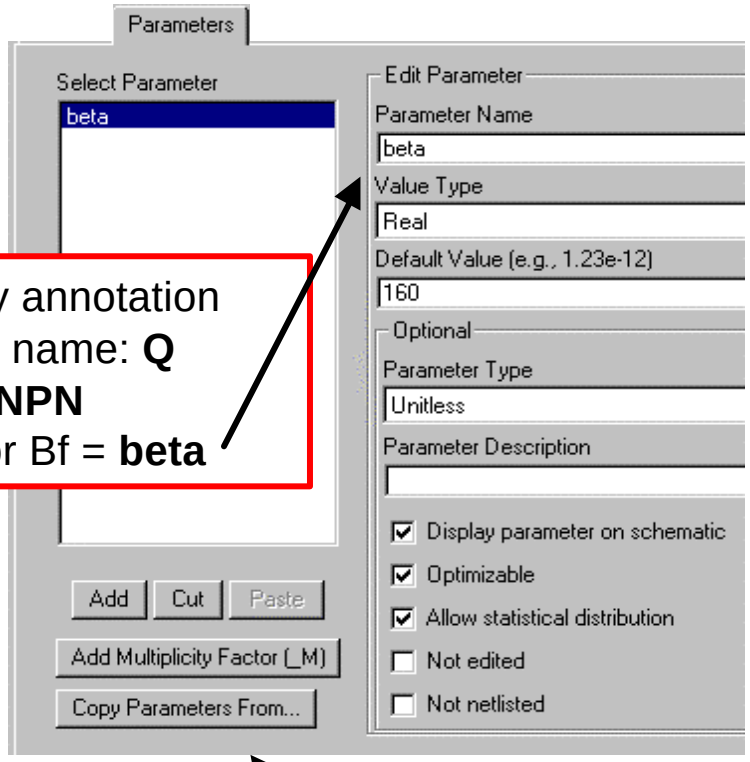
Or, to get this NPN BJT symbol with your annotation, use: **File > Design Parameters**.

Design Parameters for your schematic

Click: *File > Design / Parameters*



- Description for library annotation
- Component Instance name: **Q**
- Symbol: **SYM_BJT_NPN**
- Passed parameter for Bf = **beta**

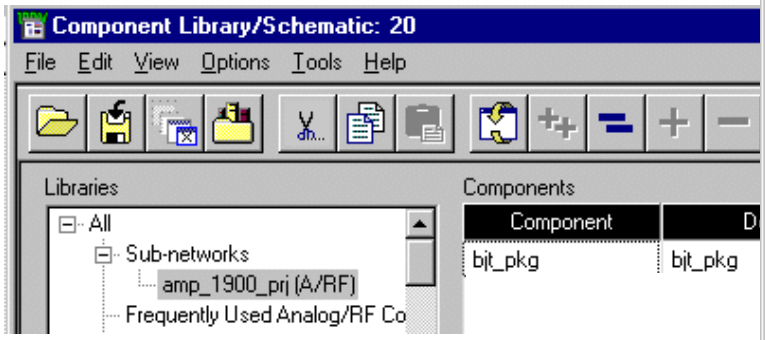


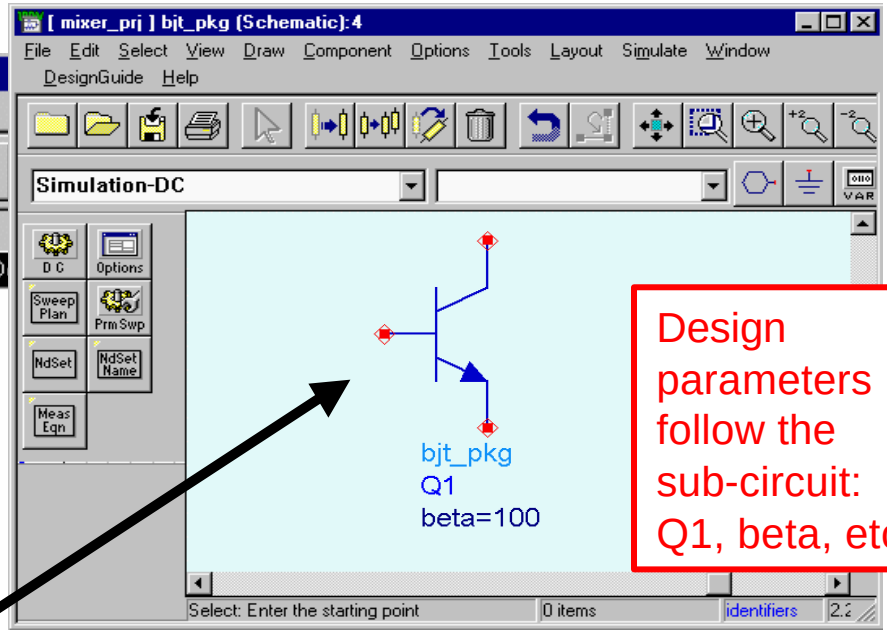
You can specify a layout: built-in sot23

You can copy parameters from other library models.

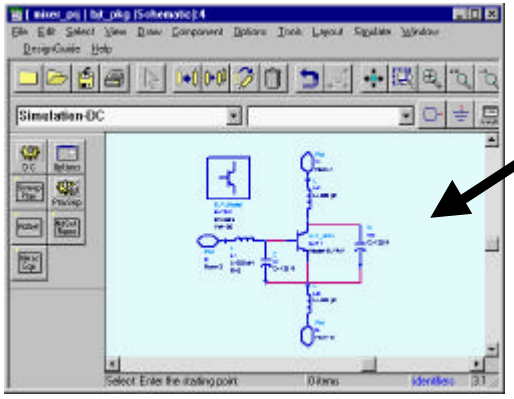
Insert the model in a new schematic

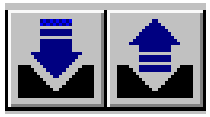
Insert the sub-circuit from the library.





Design parameters follow the sub-circuit: Q1, beta, etc.

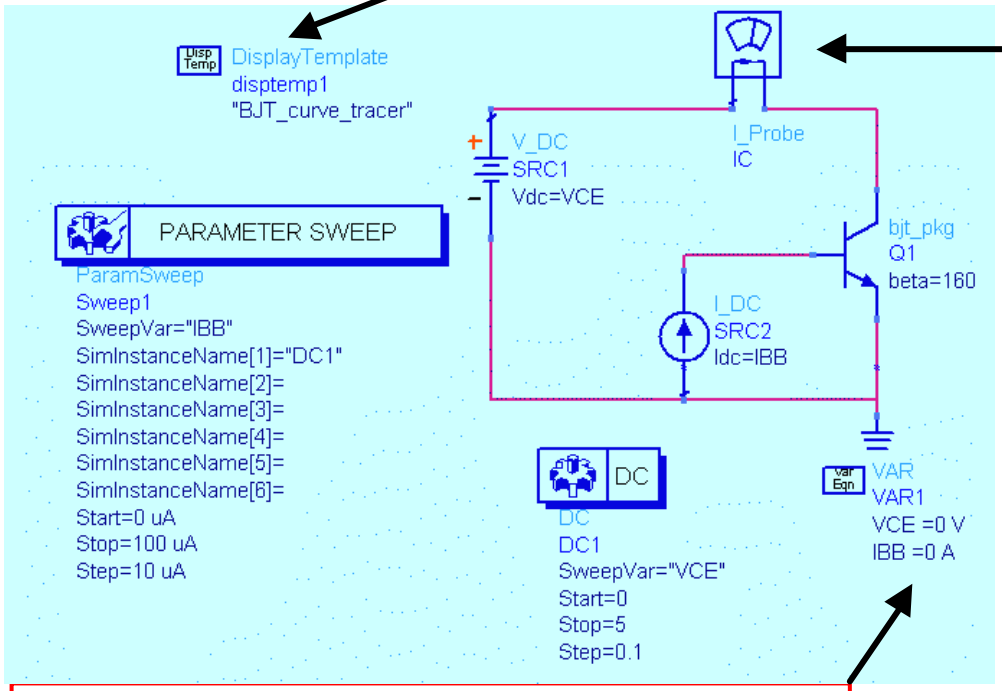




ICONS: Push into and Pop out of the hierarchy.

Set up a DC curve sweep with a template

This template also has a data display template.

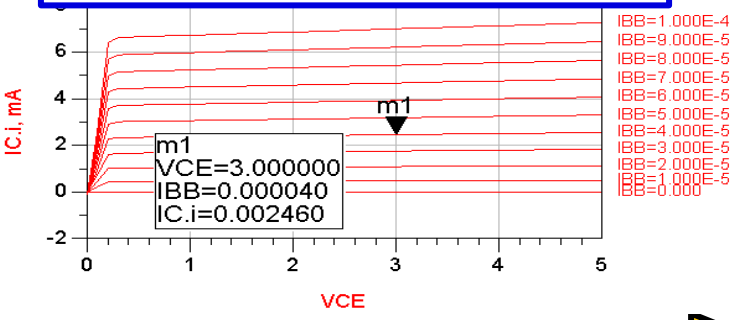


Probe give current current in dataset.

Your model (**bjt_pkg**) with annotation and passed parameter: beta.

Initialized VARs: VCE=0V & IBB=0A

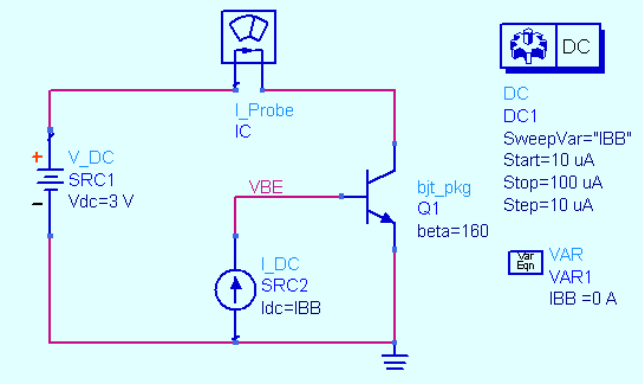
Data Display template gives curve tracer results:



NOTE: DC controller sweeps the X-axis and the Parameter Sweep, sweeps the Y-axis.

Finally, calculate and test a bias network

Setup a new sweep: then use voltage and current to calculate R values for the DC specs for Ib and Ic.



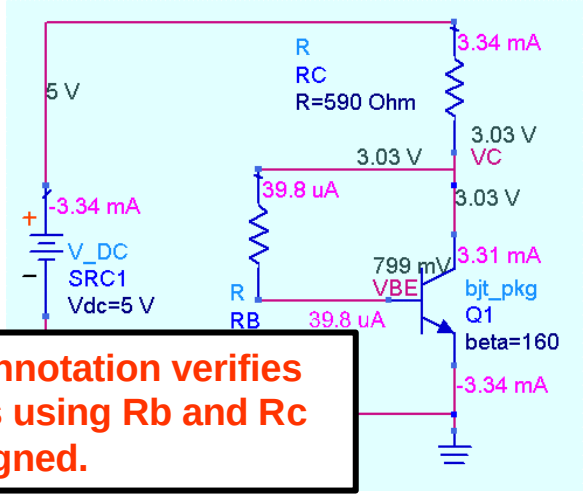
IBB	VBE	IC.i
1.000E-5	754.8mV	599.8uA
2.000E-5	777.1mV	1.430mA
3.000E-5	789.9mV	2.349mA
4.000E-5	798.8mV	3.325mA
5.000E-5	805.7mV	4.341mA
6.000E-5	811.3mV	5.389mA
7.000E-5	815.9mV	6.462mA
8.000E-5	819.9mV	7.557mA
9.000E-5	823.5mV	8.669mA
1.000E-4	826.6mV	9.798mA

Eqn Rb=(3-VBE) / IBB

Eqn Rc= 2 / (IC.i + IBB)

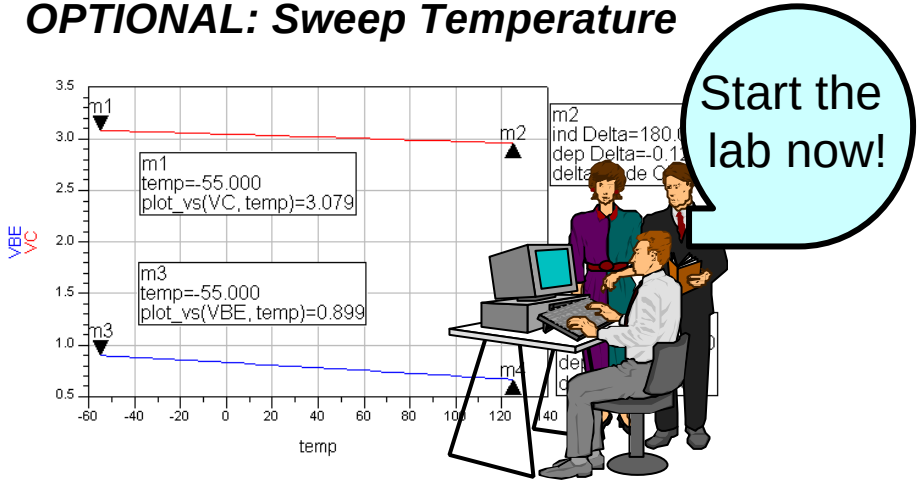
Bias Resistor Calculations

Rb[3]	Rc[3]
55029.037	594.350



Back Annotation verifies DC bias using Rb and Rc as designed.

OPTIONAL: Sweep Temperature



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。

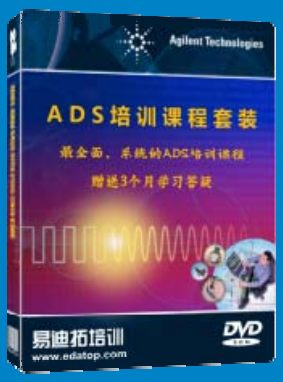
易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

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…

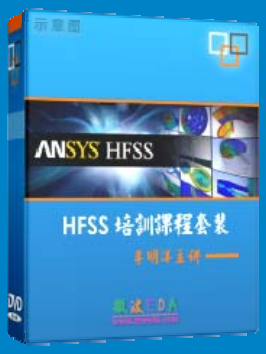
课程网址：http://www.edatop.com/peixun/rfe/110.html

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