

基于OrCAD的
电子线路计算机辅助设计

第四讲器件与建模

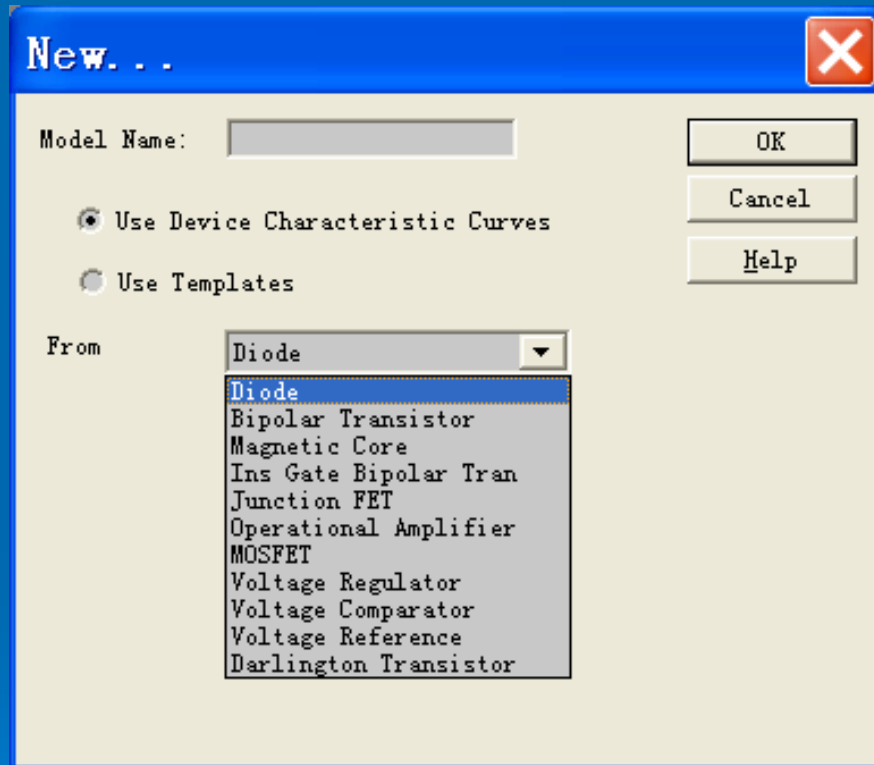
关于器件模型

- 用于仿真的器件由模型支持
- 系统提供的模型库
- 创建模型
- 从网上下载模型

建模方法

- 通过模型编辑器创建：
 - 基于器件特性曲线
 - 基于器件模板
 - 通过子电路网表创建
 - 基于Capture绘图，创建子电路网表，进行精调。
 - ABM器件
 - 模型内容
 - 可由以上几种分别、组合、共同组成
- 模型的应用：创建新器件、同类器件选用新模型。

基于器件特性曲线建模



- 打开模型编辑器Model Editor，执行命令File/New，Mode/New，弹出创建新器件模型对话框：

建模平台

Untitled3.lib:diode - PSpice Model Editor - [Forward Current]

File Edit View Model Plot Tools Window Help

Models List

Mode...	Type	Modifie...
diode*	Diode	12/20/0...

Forward Current

To include this spec in the model extraction please enter two or

#	Vfwd	Ifwd
1		
2		
3		
4		
5		
6		
7		

Parameters

Parameter Name	Value	Minimum	Maximum	Default	Active	Fixed
IS	1e-014	1e-020	0.1	1e-014	<input checked="" type="checkbox"/>	<input type="checkbox"/>
N	1	0.2	5	1	<input checked="" type="checkbox"/>	<input type="checkbox"/>
RS	0.001	1e-006	100	0.001	<input checked="" type="checkbox"/>	<input type="checkbox"/>
IKF	0	0	1000	0	<input checked="" type="checkbox"/>	<input type="checkbox"/>
XTI	3	-100	100	3	<input type="checkbox"/>	<input type="checkbox"/>
EG	1.11	0.1	5.51	1.11	<input type="checkbox"/>	<input type="checkbox"/>
CJO	1e-012	1e-020	0.001	1e-012	<input type="checkbox"/>	<input type="checkbox"/>
M	0.3333	0.1	10	0.3333	<input type="checkbox"/>	<input type="checkbox"/>
VJ	0.75	0.3905	10	0.75	<input type="checkbox"/>	<input type="checkbox"/>
FC	0.5	0.001	10	0.5	<input type="checkbox"/>	<input type="checkbox"/>
ISR	1e-010	1e-020	0.1	1e-010	<input type="checkbox"/>	<input type="checkbox"/>
NR	2	0.5	5	2	<input type="checkbox"/>	<input type="checkbox"/>
BV	100	0.1	1000000	100	<input type="checkbox"/>	<input type="checkbox"/>
IBV	0.0001	1e-009	10	0.0001	<input type="checkbox"/>	<input type="checkbox"/>
TT	5e-009	1e-016	0.001	5e-009	<input type="checkbox"/>	<input type="checkbox"/>

Ready NUM

输入数据表信息(例)

模型特性名	参数值
forward current	(1.3, 0.2)
junction capacitance	(1m, 120p) (1, 73p) (3.75, 45p)
reverse leakage	(6, 20n)
reverse breakdown	($V_z=7.5$, $I_z=20m$, $Z_z=5$)
reverse recovery	没有变化 (取缺省值)

括号中数据点的为 x 、 y 坐标值

激活新的数据表信息，

- 按数据表信息设置各页面参数后，执行 Tools/Extrat Parameter命令，各窗口的曲线图型及下方的Parameters信息同时随新参数变为新值，从而生成新的模型参数。如图：

The screenshot shows the PSpice Model Editor interface for a diode model. The main window is titled "mydiode.lib:diode - PSpice Model Editor". It features a menu bar (File, Edit, View, Model, Plot, Tools, Window, Help) and a toolbar. The interface is divided into several panes:

- Models ...**: A list of models, with "diode" selected.
- Junction Capacitance**: A graph showing the junction capacitance (Cj) versus reverse voltage. The y-axis ranges from 0F to 200pF, and the x-axis ranges from 0V to 1.0V. The curve shows a decreasing trend.
- Reverse Recovery**: A graph showing reverse current versus time. The y-axis ranges from -20mA to 20mA, and the x-axis ranges from 0s to 20ns. The curve shows a sharp negative peak followed by a decay.
- Forward Current**: A graph showing forward current (Ifwd) versus forward voltage. The y-axis ranges from 10fA to 1.0A, and the x-axis ranges from 1.0V to 3.0V. The curve shows an exponential increase.
- Reverse Breakdown**: A graph showing reverse current versus reverse voltage. The y-axis ranges from 100uA to 1.0A, and the x-axis ranges from 7.0V to 8.0V. The curve shows a sharp increase in current at the breakdown voltage.
- Reverse Leakage**: A graph showing reverse current versus reverse voltage. The y-axis ranges from 0A to 100nA, and the x-axis ranges from 25V to 75V. The curve shows a very small, nearly constant current.

At the bottom, the **Parameters** table is visible, listing various model parameters and their values:

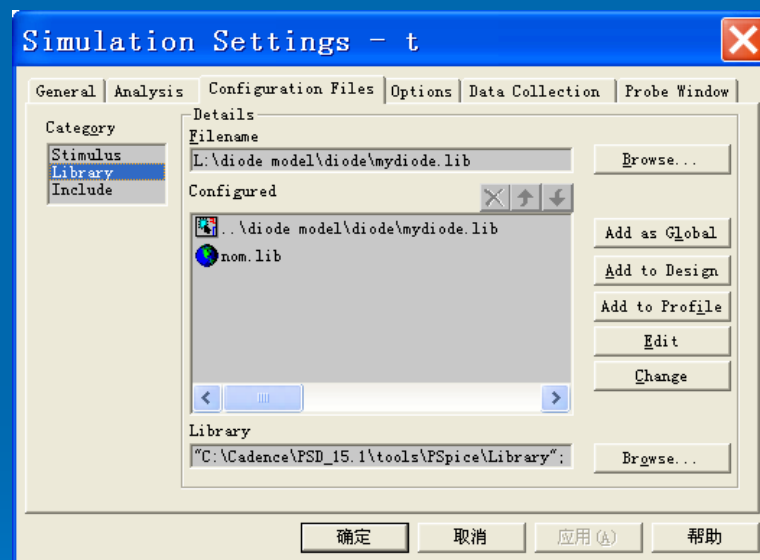
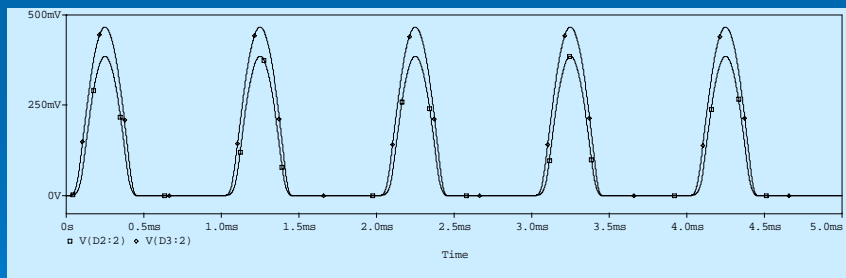
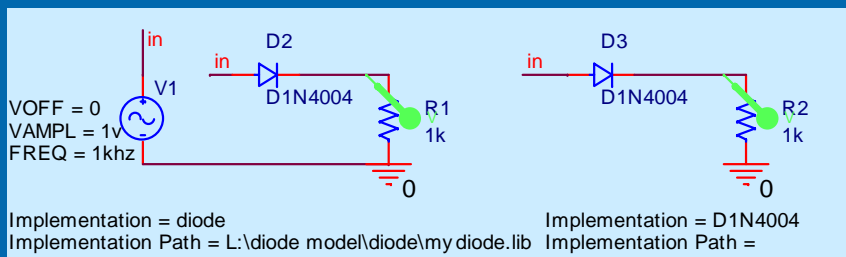
Parameter Name	Value	Minimum	Maximum	Default	Active	Fixed
CJO	1.2011e-010	1e-020	0.001	1e-012	<input checked="" type="checkbox"/>	<input type="checkbox"/>
M	0.47055	0.1	10	0.3333	<input checked="" type="checkbox"/>	<input type="checkbox"/>
VJ	0.5318	0.3905	10	0.75	<input checked="" type="checkbox"/>	<input type="checkbox"/>
FC	0.5	0.001	10	0.5	<input type="checkbox"/>	<input type="checkbox"/>

导出模型文件

- 执行Model/Export命令 导出当前模型参数，保存为.MOD文件。
 - .Mod文件可以用标准文本编辑器打开，可以编辑，可以加入模型库。本例内容及格式如下：
 - .MODEL diode D
 - + IS=10.010E-21
 - + RS=1.0000E-3
 - + CJO=120.11E-12
 - + M=.47065
 - + VJ=.5318
 - + ISR=6.1426E-9
 - + BV=7.5662
 - + IBV=.25869
 - + TT=5.0000E-9
- 执行File/Save命令，保存当前模型到当前模型库.lib。
 - 执行File/Save as命令，将当前模型保存为新的模型库或到选择的模型库。
 - 用mode/import命令向模型库追加新模型。

应用模型

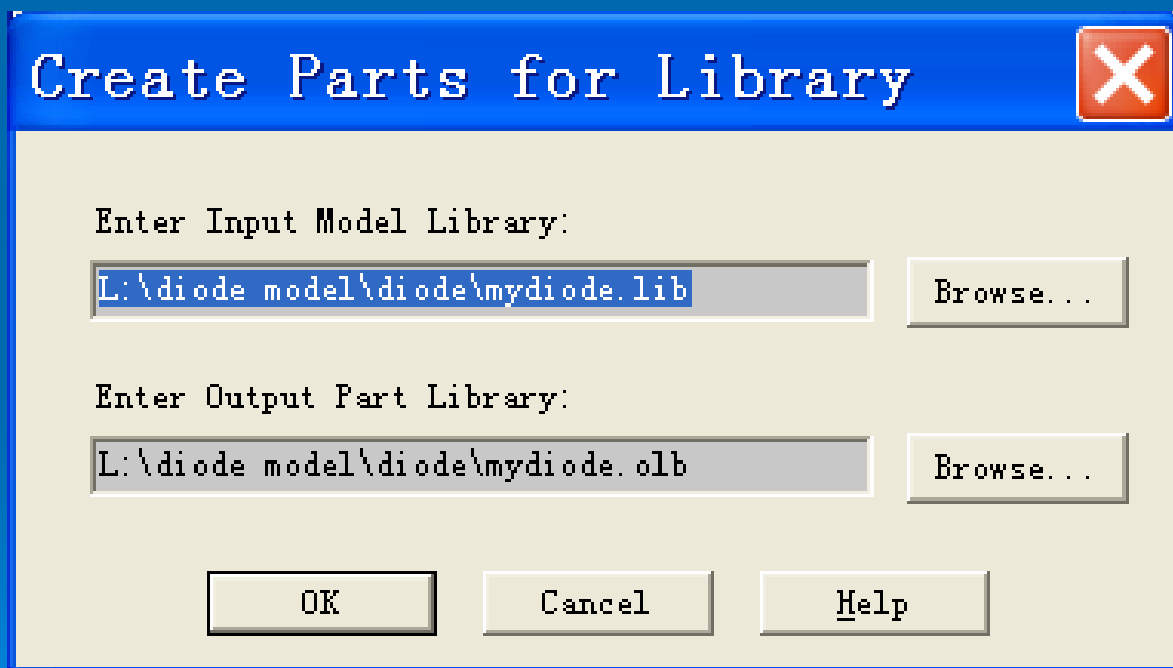
- 从器件库取出同类模型建立电路
- 为器件赋予新的模型文件
- 配置模型库文件




	Graphic	ID	Implementation	Implementation Path	Implementation Type	Part Reference	Reference	Value
+ SCHEMATIC1 : PAGE1 : D2	D1N4004.Normal		diode	L:\diode model\diode\my ...	PSpice Model	D2	D2	D1N4004

创建新器件

- 在模型编辑器执行File/Export to Capture Part Library...命令，创建新器件至器件库.olb。
- 调用新器件绘图
- 配置模型库



基于器件模板的模型

New... 

Model Name:

Use Device Characteristic Curves

Use Templates

From:

Technology

BJT JFET

Input Stage

NPN PNP

Compensation

Internal External

运放的器件模板

The screenshot displays the PSpice Model Editor interface for a model named 'OPA-LOCAL_3'. The 'Simulation Parameters' table is the central focus, listing various electrical and operational characteristics of the op-amp model.

Property Name	Description	Value	Default	Unit	D
VOS	Offset voltage	1e-6	1e-6	V	
IB	Input bias current	100e-12	100e-12	A	
IBOS	Input offset current	1e-12	1e-12	A	
A0	Open-loop gain	200000	200000	V/V	
GBW	Gain-BW product	1e+6	1e+6	Hz	
SRP	Positive slew rate	0.527e+6	0.527e+6	V/sec	
SRM	Negative slew rate	0.527e+6	0.527e+6	V/sec	
CMRR	Common-mode reject.	100000	100000	V/V	
PO	GBW excess phase	0	0	deg	
PD	Quies. power dissip.	50e-3	50e-3	W	
ISCP	I short(+ sink)	25e-3	25e-3	A	
ISCM	I short(- source)	-25e-3	-25e-3	A	
ROUT	D.C. output res	75	75	ohms	
ROAC	A.C. output res	20	20	ohms	

Below the simulation parameters, the 'Model Text (Read Only)' window shows the following text:

```
* created using Model Editor release 10.1.0 on 10/28/05 at 22:29
*DEVICE=OPA-LOCAL_3,OPAMP,NPN,INT
.subckt OPA-LOCAL_3 PIN NIN OUT PVSS NVSS
+ params:
XinstOPA-LOCAL_3 PIN NIN OUT PVSS NVSS model1639a_3
+ params:
```

模型参数

Property Name	Description	Value	Default	Unit	Distribution	Postol	Hegtol	Editable
VOS	Offset voltage	1e-6	1e-6	V				<input checked="" type="checkbox"/>
IB	Input bias current	100e-12	100e-12	A				<input checked="" type="checkbox"/>
IBOS	Input offset current	1e-12	1e-12	A				<input checked="" type="checkbox"/>
A0	Open-loop gain	200000	200000	V/V				<input checked="" type="checkbox"/>
GBW	Gain-BW product	1e+6	1e+6	Hz				<input checked="" type="checkbox"/>
SRP	Positive slew rate	0.527e+6	0.527e+6	V/sec				<input checked="" type="checkbox"/>
SRM	Negative slew rate	0.527e+6	0.527e+6	V/sec				<input checked="" type="checkbox"/>
CMRR	Common-mode reject.	100000	100000	V/V				<input checked="" type="checkbox"/>
PO	GBW excess phase	0	0	deg				<input type="checkbox"/>
PD	Quies. power dissip.	50e-3	50e-3	W				<input type="checkbox"/>
ISCF	I short(+ sink)	25e-3	25e-3	A				<input type="checkbox"/>
ISCM	I short(- source)	-25e-3	-25e-3	A				<input type="checkbox"/>
ROUT	D.C. output res	75	75	ohms				<input type="checkbox"/>
ROAC	A.C. output res	20	20	ohms				<input type="checkbox"/>
VFDIFF	Pos. output diff.	1	1	V				<input type="checkbox"/>
VMDIFF	Neg. output diff.	1	1	V				<input type="checkbox"/>
VCC	Supply volt +	15	15	V				<input type="checkbox"/>
VSS	Supply volt -	-15	-15	V				<input type="checkbox"/>
CINDM	Diff-mode input cap	0	0	F				<input type="checkbox"/>

- 各列属性：属性名、参数表述、参数值、缺省值、单位、误差分布类型、正向公差、负向公差、激活选项。
- 参数属性可编辑，激活**editable**选项对应的参数，保存后修改的信息反映在模型文本中，并且以修改的值覆盖缺省值。
- **Editable**激活的参数可在图形编辑界面修改。
- 执行Model/Export命令提取模型参数是针对选中激活选项的参数。

修改参数（例）

仿真属性名	参数值	公差分布类型	正向公差 (Postol)	负向公差 (Negtol)
VOS	$1e^{-7}$	Flat	10%	10%
IB	default	Flat	10%	10%
IBOS	default	Flat	10%	10%
A0	1000000	Flat	10%	10%
GBW	default	Flat	10%	10%
SRP	$1.0e^{+6}$	Flat	10%	10%
SRM	$1.0e^{+6}$	Flat	10%	10%
CMRR	default	Flat	10%	10%

修改参数后的模型

The screenshot shows the PSpice Model Editor interface. The main window is titled "mylocal_lib.lib:myopa_3 - PSpice Model Editor - [Simulation Parameters]". The menu bar includes File, Edit, View, Model, Plot, Tools, Window, and Help. The toolbar contains various icons for file operations and simulation control.

On the left, there is a "Models List" panel with the following entries:

Model...	Type	Modi
myopa_3	0...	12/2
myopa_2	0...	12/2
myopa_1	0...	12/2

The main area displays the "Simulation Parameters" table:

Property Name	Value	Default	Unit	Distribution	Postol	Neqtol	Editable
VOS	1e-7	1e-6	V	FLAT	10%	10%	<input checked="" type="checkbox"/>
IB	100e-12	100e-12	A	FLAT	10%	10%	<input checked="" type="checkbox"/>
IBOS	1e-12	1e-12	A	FLAT	10%	10%	<input checked="" type="checkbox"/>
A0	100000	200000	V/V	FLAT	10%	10%	<input checked="" type="checkbox"/>
GBW	1e+6	1e+6	Hz	FLAT	10%	10%	<input checked="" type="checkbox"/>
SRP	1e+6	0.527e+6	V/sec	FLAT	10%	10%	<input checked="" type="checkbox"/>
SRM	1e+6	0.527e+6	V/sec	FLAT	10%	10%	<input checked="" type="checkbox"/>
CMRR	100000	100000	V/V	FLAT	10%	10%	<input checked="" type="checkbox"/>
FO	0	0	deg				<input type="checkbox"/>
PD	50e-3	50e-3	W				<input type="checkbox"/>
ISCP	25e-3	25e-3	A				<input type="checkbox"/>
ISCM	-25e-3	-25e-3	A				<input type="checkbox"/>
ROUT	75	75	ohms				<input type="checkbox"/>
ROAC	20	20	ohms				<input type="checkbox"/>
VPDIFF	1	1	V				<input type="checkbox"/>
VMDIFF	1	1	V				<input type="checkbox"/>
VCC	15	15	V				<input type="checkbox"/>
VSS	-15	-15	V				<input type="checkbox"/>
CINDM	0	0	F				<input type="checkbox"/>

At the bottom, the "Model Text (Read Only)" panel shows the following text:

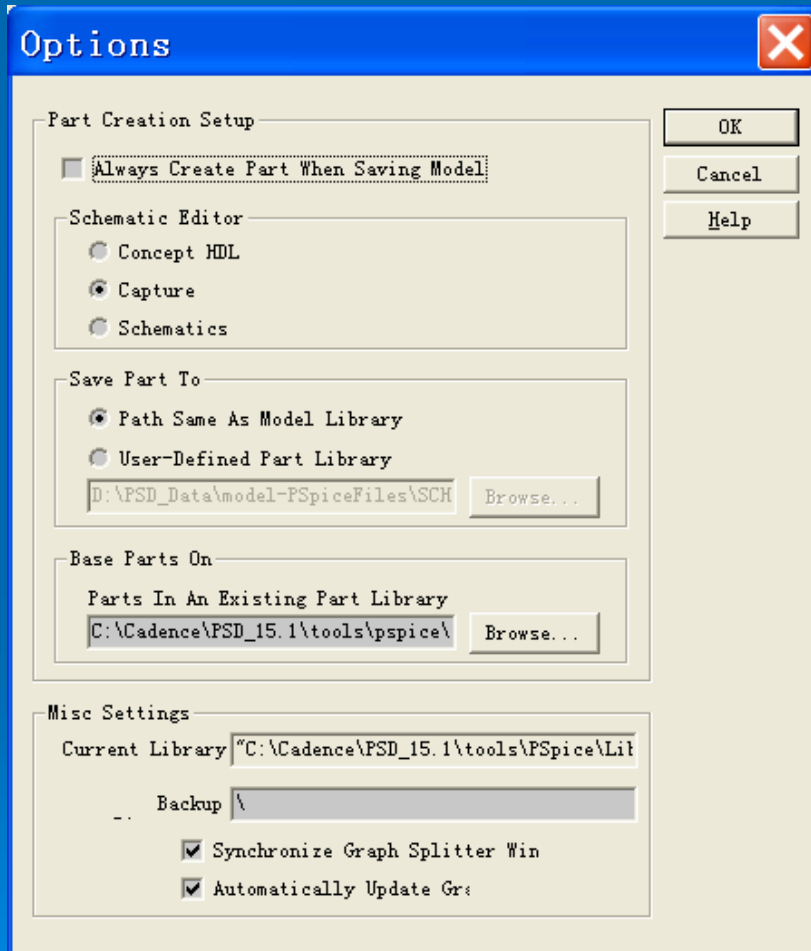
```
* updated using Model Editor release 10.1.0 on 12/23/05 at 13:07
*DEVICE=myopa_3,OPAMP,NPN,INT
.subckt myopa_3 PIN NIN OUT PVSS NVSS
+ params:
+ VOS=1e-7
+ IB=100e-12
+ IBOS=1e-12
+ A0=100000
+ GBW=1e+6
+ SRP=1e+6
+ SRM=1e+6
+ CMRR=100000
Xinstmyopa_3 PIN NIN OUT PVSS NVSS model639a_3
```

The status bar at the bottom left shows "Ready" and the bottom right shows "NUM".

模型应用

- 在Captue绘图页面上安放同种类型的Pspice器件，通过器件的Implementation属性，将模型名称赋给该属性并配置模型库。
- 创建相应的器件，应用时用创建的器件绘图，再配置模型库
- 对于多极模型，可创建单极或多级模型的器件

创建单级模型参数的器件



- 打开模型编辑器，执行 **Tools Option** 命令
- 在弹出的选项框中选中 **Always Create Part When Saving Model** 选项
- 执行 **Save** 命令保存模型的同时生成单级器件。

单级器件模型

- 其中包含仿真参数和Smoke参数。

The screenshot displays two windows from a simulation software. The 'Smoke Parameters' window on the left shows a 'Test Node Mapping' table. The 'Simulation Parameters' window on the right shows a table of simulation parameters.

Smoke Parameters - Test Node Mapping

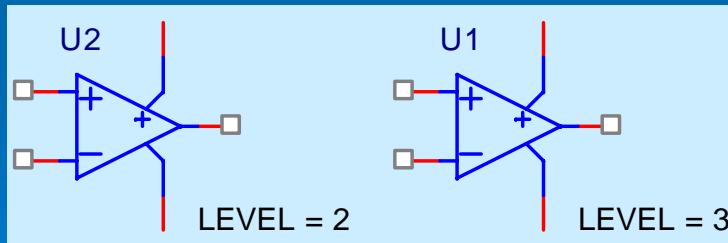
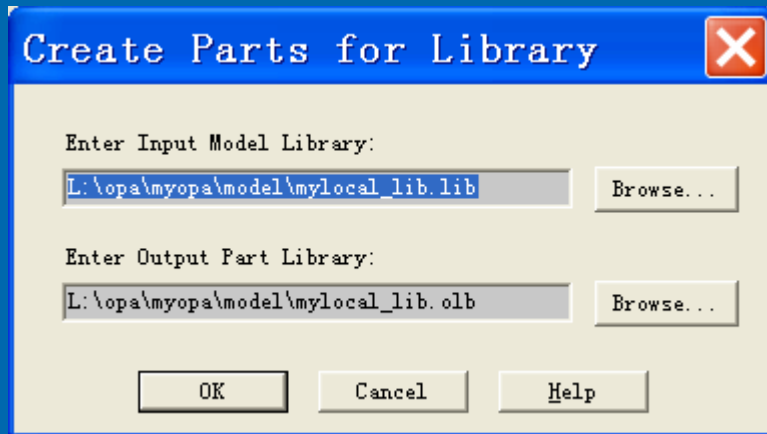
This is the Nodes and Port Mapping. This mapping is non-editable

Node	Port
NODE_POS	PIN
NODE_NEG	NIN
NODE_VCC	PVss
NODE_VEE	NVss
NODE_GND	0
TERM_POS	PIN
TERM_NEG	NIN
TERM_OUT	OUT

Simulation Parameters

Property Name	Description	Value	Default	Unit	Distribution	Postol	Hegto
GBW	Gain-BW product	1e+6	1e+6	Hz			
SRP	Positive slew rate	0.527e+6	0.527e+6	V/sec			
SRM	Negative slew rate	0.527e+6	0.527e+6	V/sec			
CMRR	Common-mode reject.	100000	100000	V/V			
PO	GBW excess phase	0	0	deg			
PD	Quies. power dissip.	50e-3	50e-3	W			
ISCP	I short(+ sink)	25e-3	25e-3	A			
ISCM	I short(- source)	-25e-3	-25e-3	A			
ROUT	D.C. output res	75	75	ohms			
ROAC	A.C. output res	20	20	ohms			
VPDIFF	Pos. output diff.	1	1	V			
VMDIFF	Neg. output diff.	1	1	V			
VCC	Supply volt +	15	15	V			
VSS	Supply volt -	-15	-15	V			
CINDM	Diff-mode input cap	0	0	F			
RINDM	Diff-mode input res	10e+9	10e+9	ohms			
enw	Equi inp white noise	10e-9	10e-9	V/sqrt(Hz)			
PSRR	Power supply reject.	1e+5	1e+5	V/V			
RLOADP	Load res for VPDIFF	2000	2000	ohms			

创建多级模型参数的器件



- 执行模型编辑器的 File/Export to Capture Part Library 命令，
- 选择与模型库同路径同名称命名器件，OK后生成新器件。
- 器件含有全部级别模型
- 设置器件属性level的级别数，level=2或level=3。
- 只拥有单级模型参数的器件不能改变模型级别。

创建完成的多级器件模型

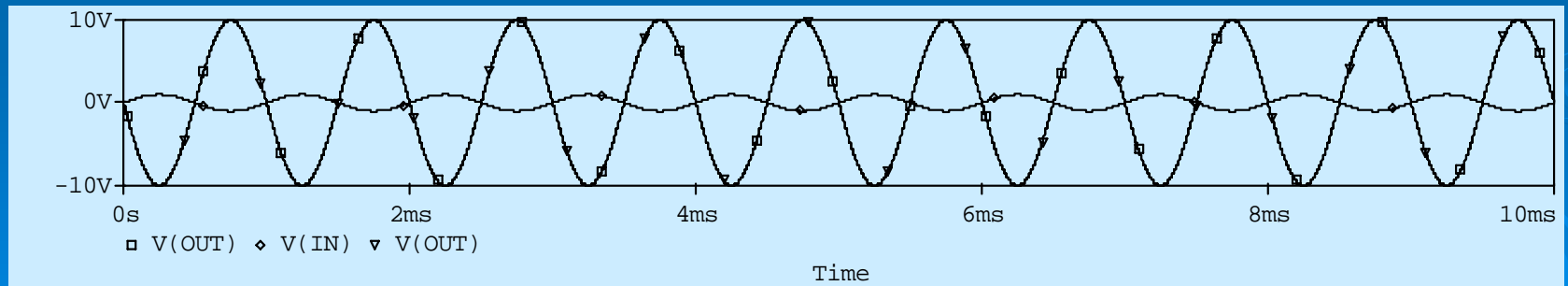
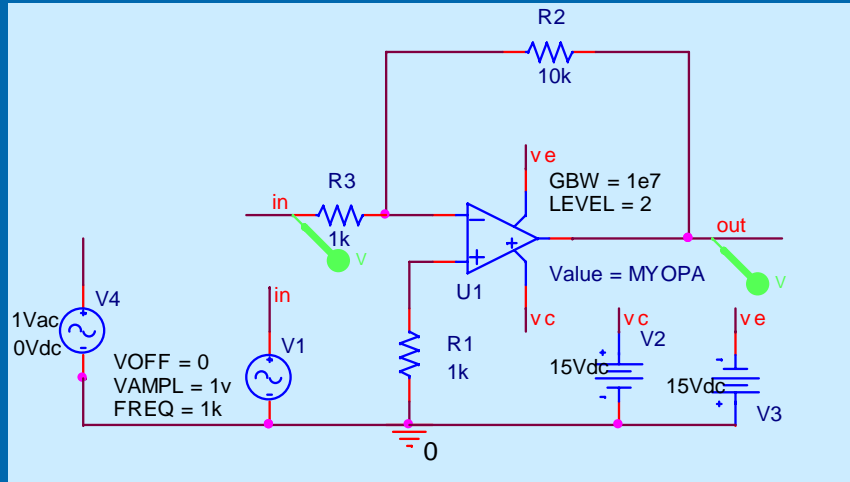
➤ 多级器件包
含全部模型

The screenshot displays the PSpice Model Editor interface for a multi-level device model. The main window is titled "mylocal_lib.lib:myopa_3 - PSpice Model Editor - [Simulation Parameters]". The "Simulation Parameters" table is the central focus, listing various properties and their values. The "Model Text" window at the bottom shows the model definition code.

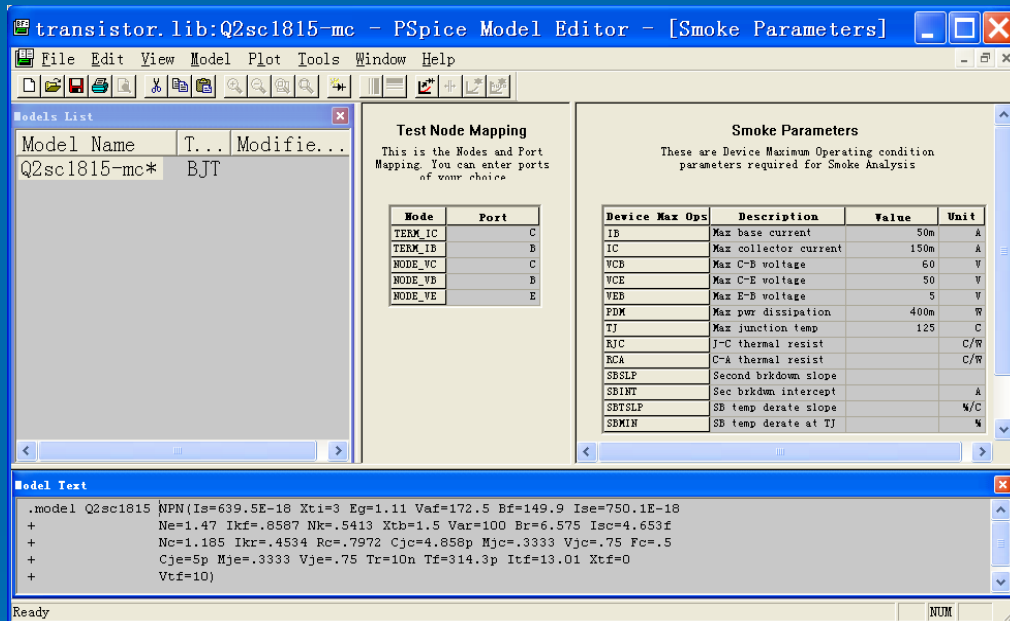
Property Name	Description	Value	Default	Unit	Distribution	Postol	Negto
VOS	Offset voltage	1e-7	1e-6	V	FLAT	10%	
IB	Input bias current	100e-12	100e-12	A	FLAT	10%	
IBOS	Input offset current	1e-12	1e-12	A	FLAT	10%	
A0	Open-loop gain	100000	200000	V/V	FLAT	10%	
GBW	Gain-BW product	1e+6	1e+6	Hz	FLAT	10%	
SRP	Positive slew rate	1e+6	0.527e+6	V/sec	FLAT	10%	
SRM	Negative slew rate	1e+6	0.527e+6	V/sec	FLAT	10%	
CMRR	Common-mode reject.	100000	100000	V/V	FLAT	10%	
PO	GBW excess phase	0	0	deg			
PD	Quies. power dissip.	50e-3	50e-3	W			
ISCP	I short(+ sink)	25e-3	25e-3	A			
ISCM	I short(- source)	-25e-3	-25e-3	A			
ROUT	D.C. output res	75	75	ohms			
ROAC	A.C. output res	20	20	ohms			
VPDIFF	Pos. output diff.	1	1	V			
VMDIFF	Neg. output diff.	1	1	V			
VCC	Supply volt +	15	15	V			
VSS	Supply volt -	-15	-15	V			
CINDM	Diff-mode input cap	0	0	F			

```
* updated using Model Editor release 10.1.0 on 12/23/05 at 12:51
*DEVICE=myopa_3,OPAMP,NPN,INT
.subckt myopa_3 PIN NIN OUT PVSS NVSS
+ params:
+ VOS=1e-7
+ IB=100e-12
+ IBOS=1e-12
+ A0=100000
+ GBW=1e+6
```

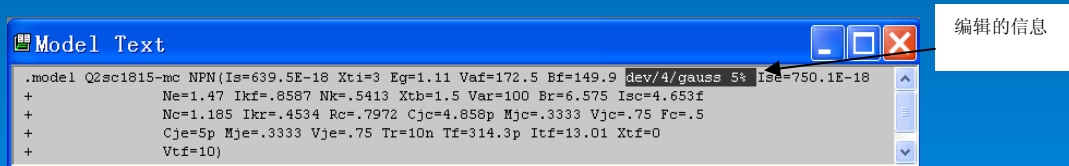
应用



编辑模型文本



- 编辑模型文本只用于基于器件特性曲线建立的模型。
- 在Capture页面选中器件，本例为Q2sc1815。
- 执行主菜单或鼠标右键命令Edit PSpice Model，打开模型编辑器并同时加载该模型参数信息。
- 编辑文本，本例加入BF值的公差信息
- 编辑文本后保存



在Capture的绘图页面编辑器件参数

The screenshot shows the PSpice Model Editor interface for a device named 'myopa_3'. The main window is titled 'Simulation Parameters' and contains a table of parameters. A 'Models List' panel on the left shows three models: myopa_3, myopa_2, and myopa_1. A 'Model Text (Read Only)' panel at the bottom displays the SPICE model code.

Property Name	Value	Default	Unit	Distribution	Postol	Negtol	Editable
VOS	1e-7	1e-6	V	FLAT	10%	10%	<input checked="" type="checkbox"/>
IB	100e-12	100e-12	A	FLAT	10%	10%	<input checked="" type="checkbox"/>
IBOS	1e-12	1e-12	A	FLAT	10%	10%	<input checked="" type="checkbox"/>
AO	100000	200000	V/V	FLAT	10%	10%	<input checked="" type="checkbox"/>
GBW	1e+6	1e+6	Hz	FLAT	10%	10%	<input checked="" type="checkbox"/>
SRP	1e+6	0.527e+6	V/sec	FLAT	10%	10%	<input checked="" type="checkbox"/>
SRM	1e+6	0.527e+6	V/sec	FLAT	10%	10%	<input checked="" type="checkbox"/>
CMRR	100000	100000	V/V	FLAT	10%	10%	<input checked="" type="checkbox"/>
PO	0	0	deg				<input type="checkbox"/>
PD	50e-3	50e-3	W				<input type="checkbox"/>
ISCP	25e-3	25e-3	A				<input type="checkbox"/>
ISCM	-25e-3	-25e-3	A				<input type="checkbox"/>
ROUT	75	75	ohms				<input type="checkbox"/>
ROAC	20	20	ohms				<input type="checkbox"/>
VPDFIFF	1	1	V				<input type="checkbox"/>
VMDIFF	1	1	V				<input type="checkbox"/>
VCC	15	15	V				<input type="checkbox"/>
VSS	-15	-15	V				<input type="checkbox"/>
CINDM	0	0	F				<input type="checkbox"/>

```
* updated using Model Editor release 10.1.0 on 12/23/05 at 13:07
*DEVICE=myopa_3,OPAMP,NPN,INT
.subckt myopa_3 PIN NIN OUT PVSS NVSS
+ params:
+ VOS=1e-7
+ IB=100e-12
+ IBOS=1e-12
+ AO=100000
+ GBW=1e+6
+ SRP=1e+6
+ SRM=1e+6
+ CMRR=100000
Xinstmyopa_3 PIN NIN OUT PVSS NVSS model1639a_3
```

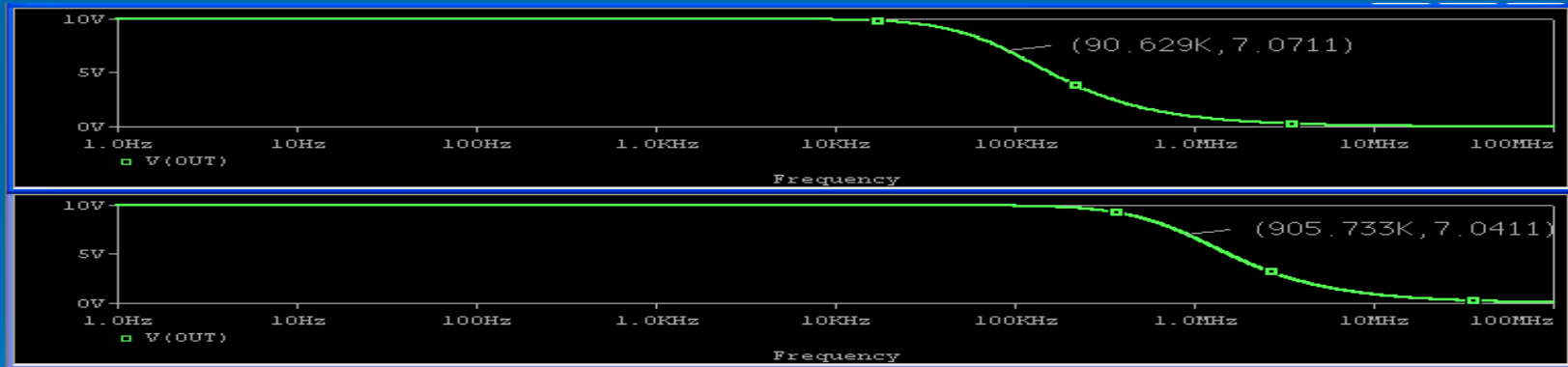
改变GWB

Property Editor

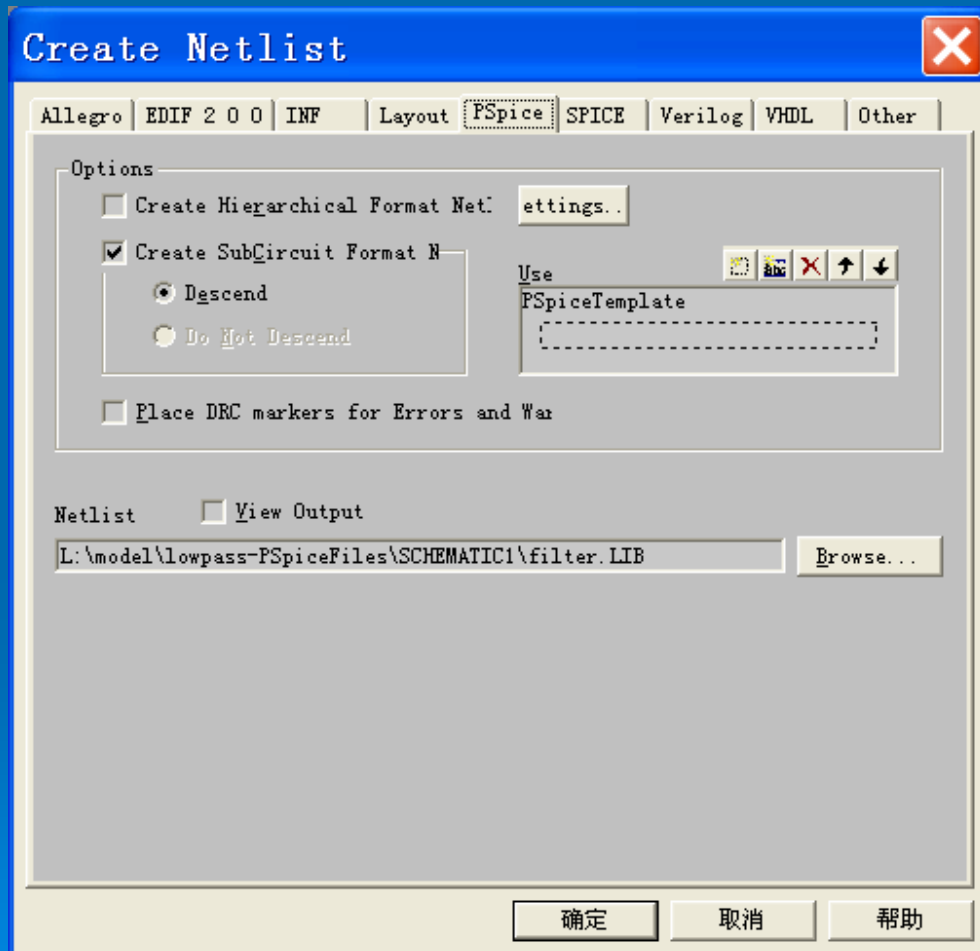
New Column... Apply Display... Delete Property Filter Orcad-PSpice Help

	PSpiceOnly	Reference	Value	GBW	BiasValue	Power	LEVEL	Source Part	TOL_ON_OFF
1	<input checked="" type="checkbox"/>	SCHEMATIC1: PAGE1: U1	U1	MYOPA	1e7	50.99mW	2	MYOPA.Normal	ON

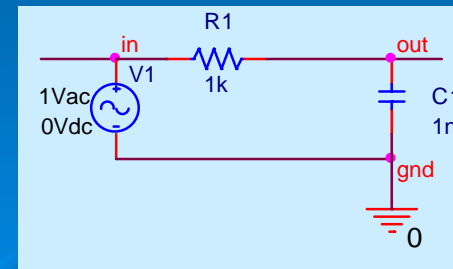
\Parts \ Schematic Nets \ Pins \ Title Bl



创建子电路网表



- 示例，创建一个简单的滤波器模型及对应的器件
- 在Capture 创建电路
- 选中项目管理器dsn文件夹，执行命令tools/create netlist...，进入PSpice页。
- 选中Create SubCircuit Format 栏
- 最后一栏选择模型库文件存放路径及文件名，确定后生成模型库文件，本例为filter.lib

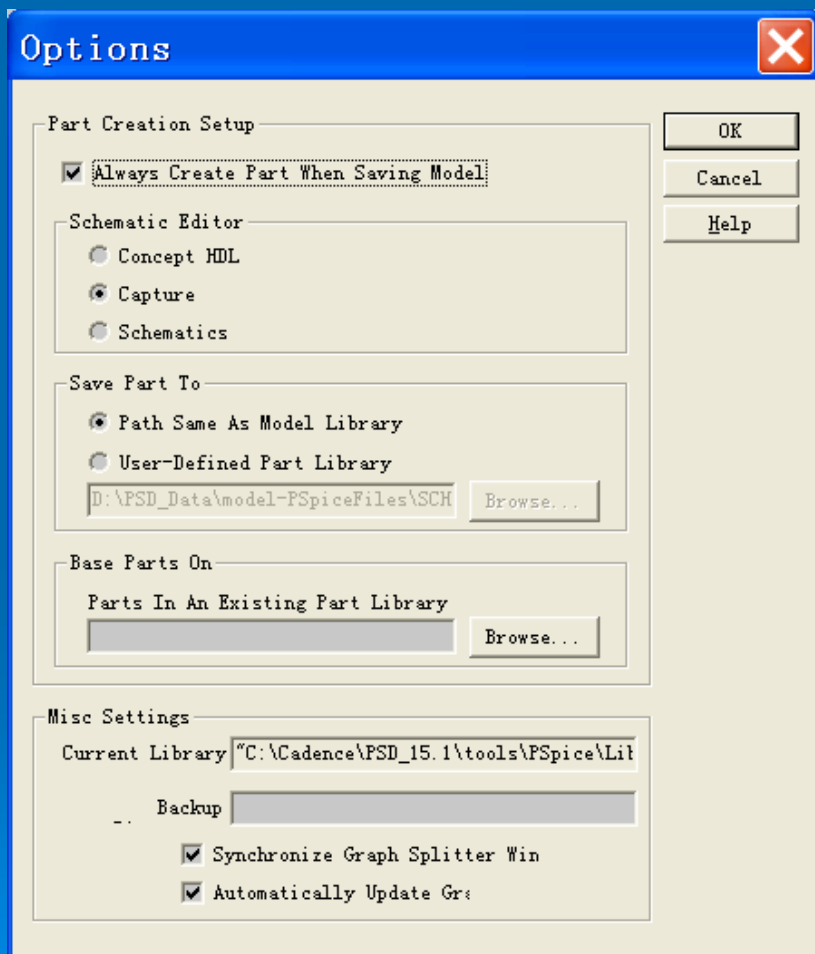


精调模型

- 用模型编辑器打开该模型库文件精调内容:
- `.SUBCKT SCHEMATIC1`
- `R_R1 IN OUT 1k`
- `C_C1 0 OUT 1n`
- `V_V1 IN 0 DC 0Vdc AC 1Vac`
- `.ENDS`
- 精调后的模型
- `.SUBCKT lowpass1 in out gnd`
- `R_R1 IN OUT 1k`
- `C_C1 gnd OUT 1n`
- `.ENDS`

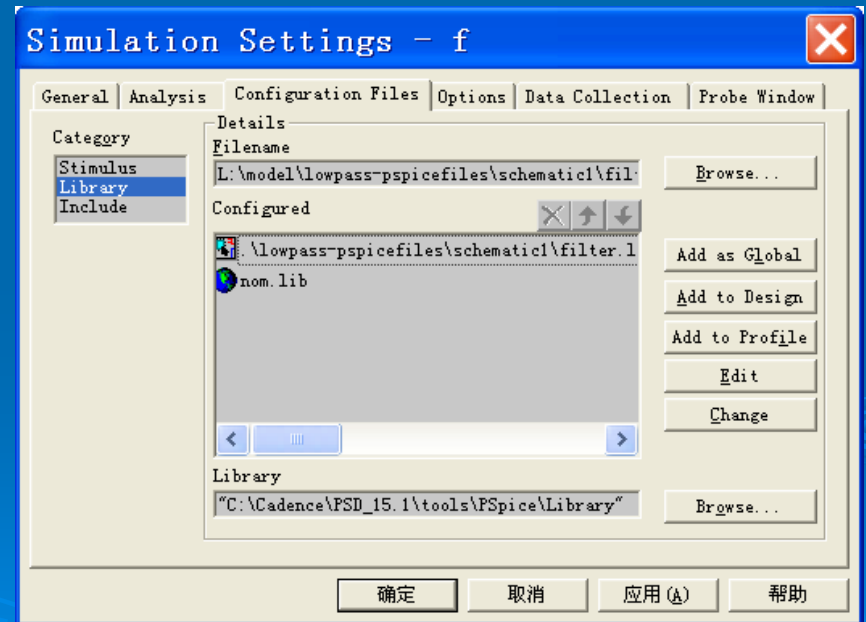
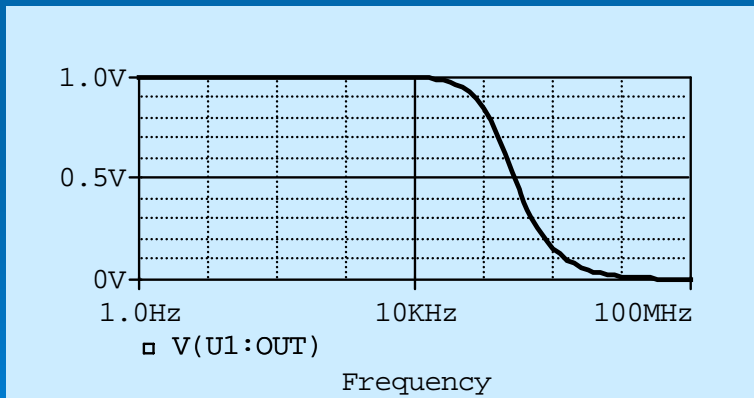
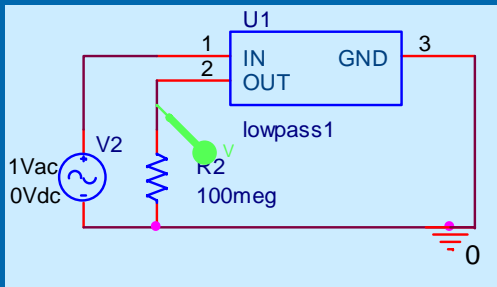
电路的网表也可以直接编辑，不必通过建立电路，然而后者会更可靠的确保网表的功能，特别是对于复杂的电路，还是以先调好电路为宜。

生成器件



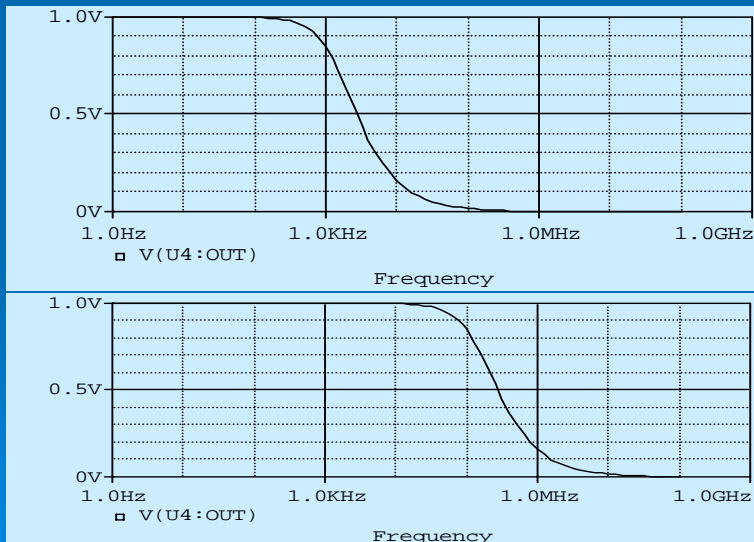
应用新器件

- 选用新器件绘图
- 如图配置器件库
- 仿真结果如图



创建器件的可编辑参数

- `.SUBCKT lowpass3 in out gnd`
- `.param valuer=1k valuec=100n`
- `R_R1 IN OUT {valuer}`
- `C_C1 gnd OUT {valuec}`
- `.ENDS`



用模型编辑器处理Smoke信息

The screenshot shows the PSpice Model Editor interface for the model 'D120NQ045'. The window title is 'commute0.lib:D120NQ045 - PSpice Model Editor - [Smoke Parameters]'. The interface is divided into several panes:

- Models List:** A table with columns 'Model Name' and 'Type'. The entry 'D120NQ045*' is listed with type 'S...'.

Model Name	Type
D120NQ045*	S...
- Test Node Mapping:** A section with the text 'This is the Nodes and Port Mapping. You can enter ports of your choice.' Below it is a table:

Node	Port
TERM_AN	A
NODE_AN	A
NODE_CAT	K
- Smoke Parameters:** A section with the text 'These are Device Maximum Operating condition parameters required for Smoke Analysis'. Below it is a table:

Device Max Ops	Description	Value	Unit
IF	Max forward current	120	A
VR	Peak reverse voltage	45	V
FDM	Max pwr dissipation	60	W
TJ	Max junction temp	150	C
RJC	J-C thermal resist	0.4	C/W
RCA	C-A thermal resist		C/W
- Model Text:** A text area containing the following content:

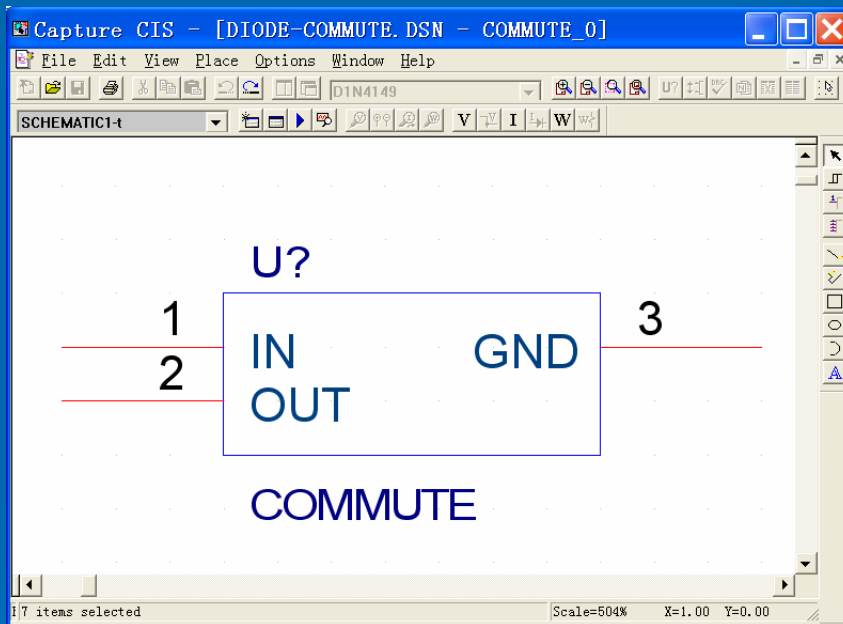
```
* created using Parts release 7.1p on 08/23/96 at 11:42
* per manufacturer est.: Vz(est)=45v@100ma; TT(est)=0s
.SUBCKT D120NQ045 1 2
D1 1 2 DFWD
D2 2 1 DLEAK
R1 1 2 228.722k

.MODEL DFWD D
+ IS=11.13717E-6
+ N=0.520057
+ RS=1.020740E-3
+ IKF=184.112E-6
+ XTI=2
+ EG=.6900
+ CJO=11.509E-9
+ M=.49488
+ VJ=.75
+ ISR=225.7639E-12
+ NR=2
+ BV=44.905
+ IBV=2.5867E-3
+ TT=0

.MODEL DLEAK D
+ IS = 186.0702E-12
```

The status bar at the bottom shows 'Ready' and a digital clock '16:27:33'.

编辑器件外形



- 在Capture的绘图页面上选中要编辑的器件，执行鼠标邮件命令Part Editor，进入该器件外形编辑界面。
- Capture的Option/DesignTemplate命令在Page Size页面确定光标落在格点上以及各点间距。
- 器件引脚必须落在格点上。
- 需要连续方式绘制图形，可直接点击菜单栏的Snap to grid图标。

编辑外形注意事项

- 在编辑器件图形时，要注意PSPICETEMPLATE定义中引脚或节点的标号、使用的指令是否正确，引脚名与器件的引脚名是否匹配等问题。
- 对于使用正常的器件，如果没有把握，在对器件进行编辑时，不要更改器件的引脚及节点名。

End