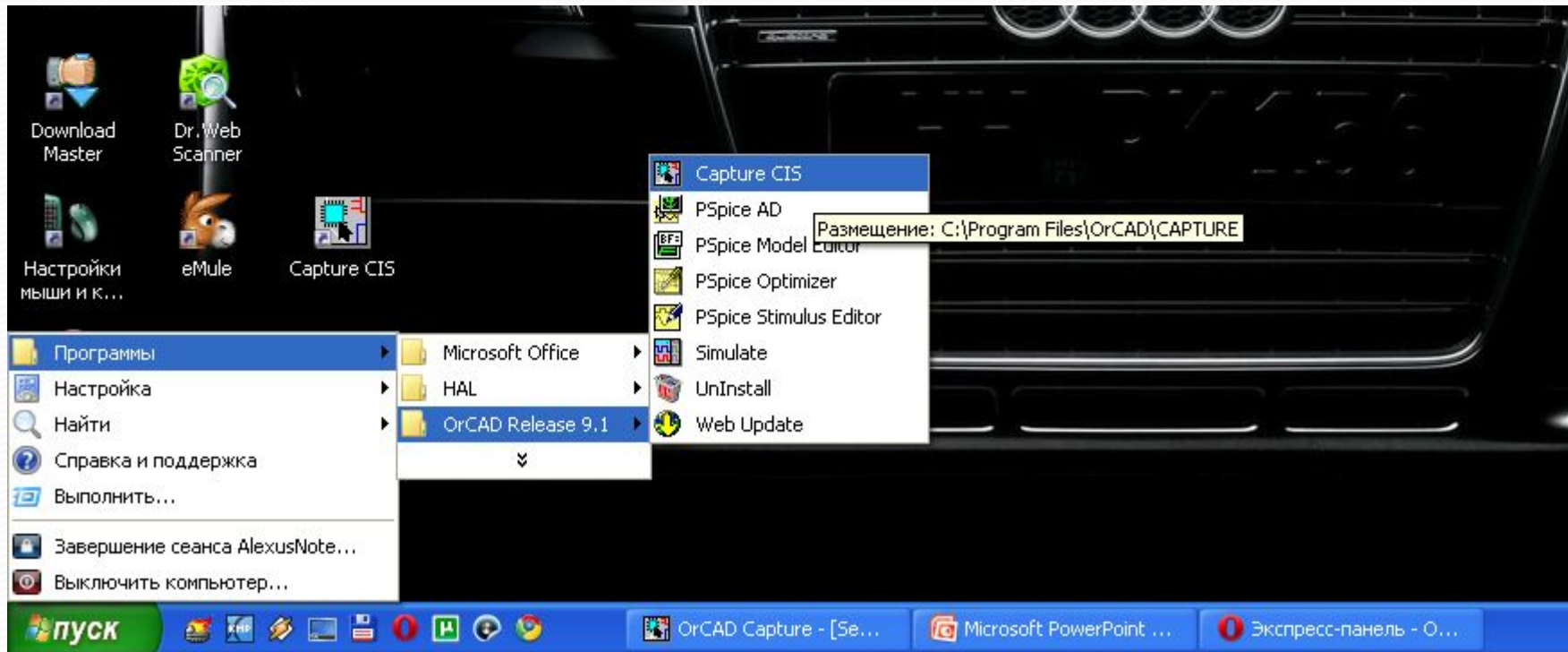


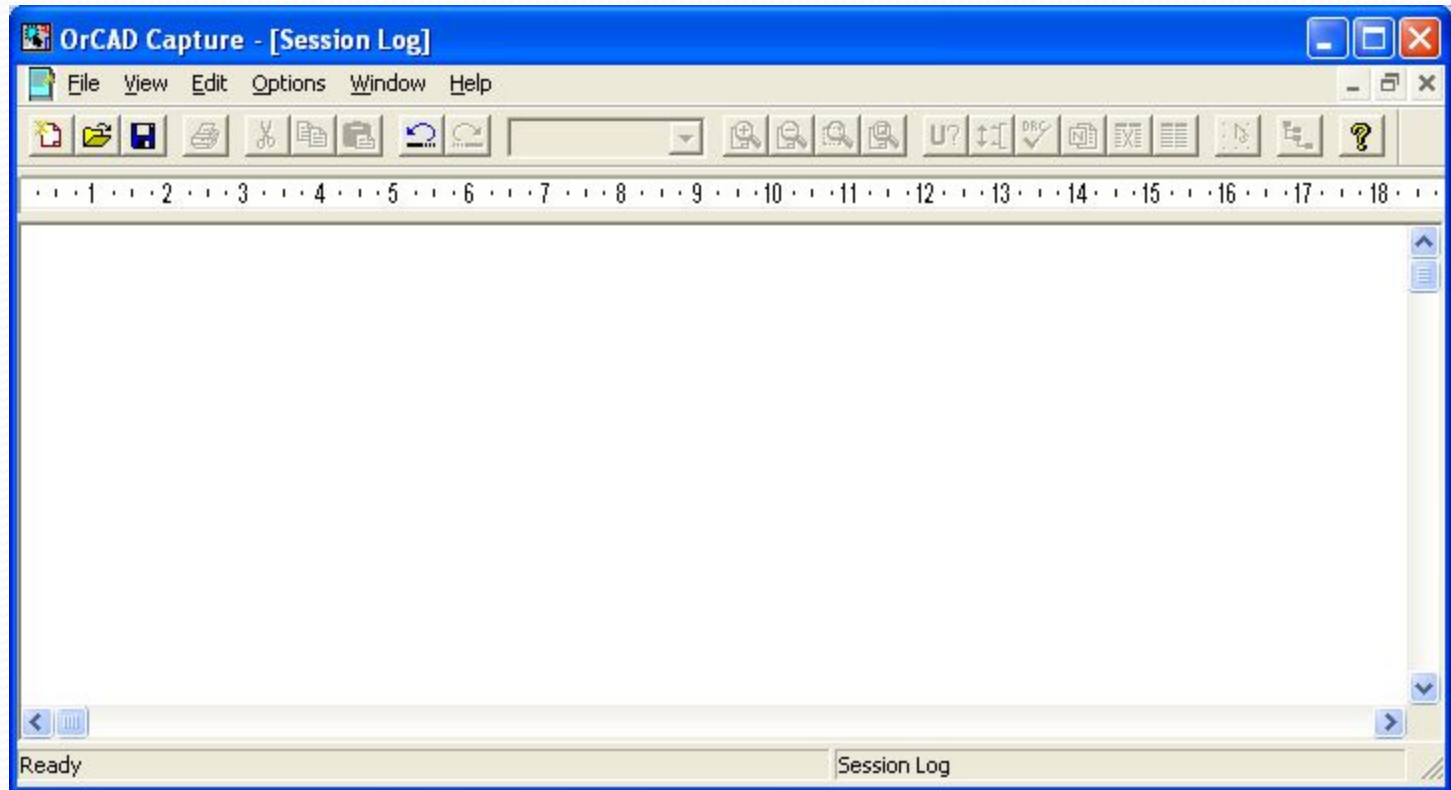
# Основы работы в OrCad

# Запуск программы и первое знакомство с интерфейсом

Запуск осуществляется при помощи ярлыка из меню ПУСК или Рабочего стола

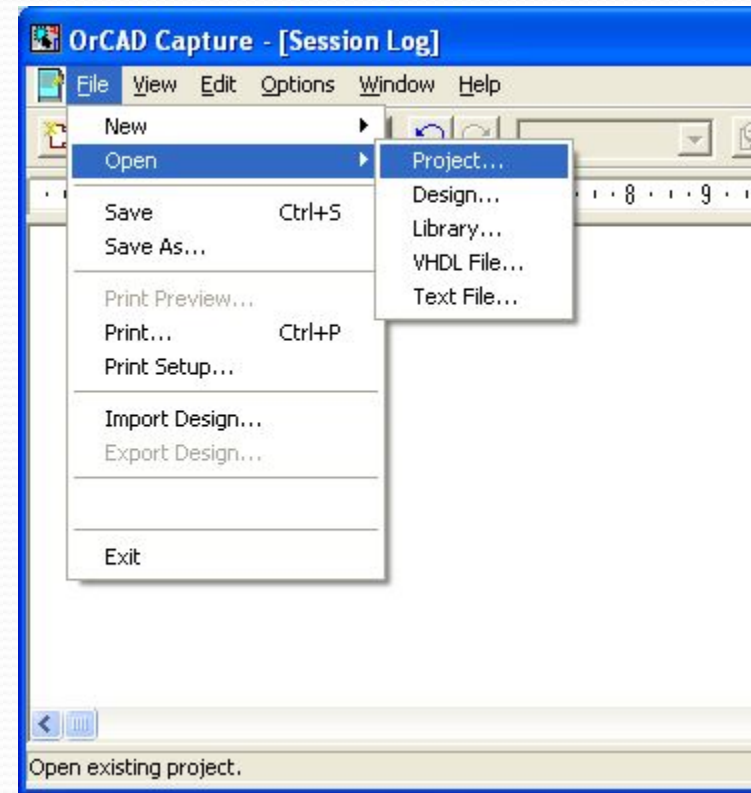
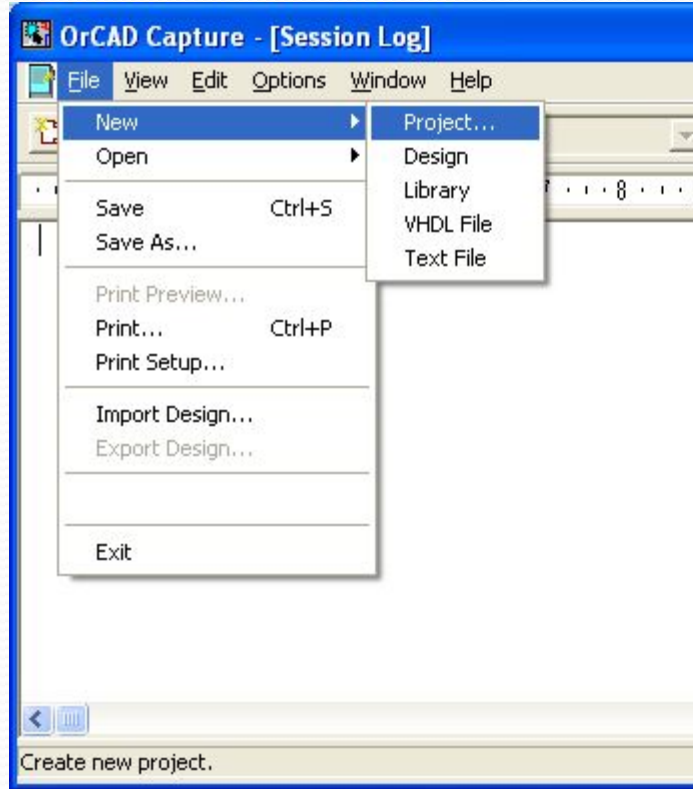


# Главное окно программы

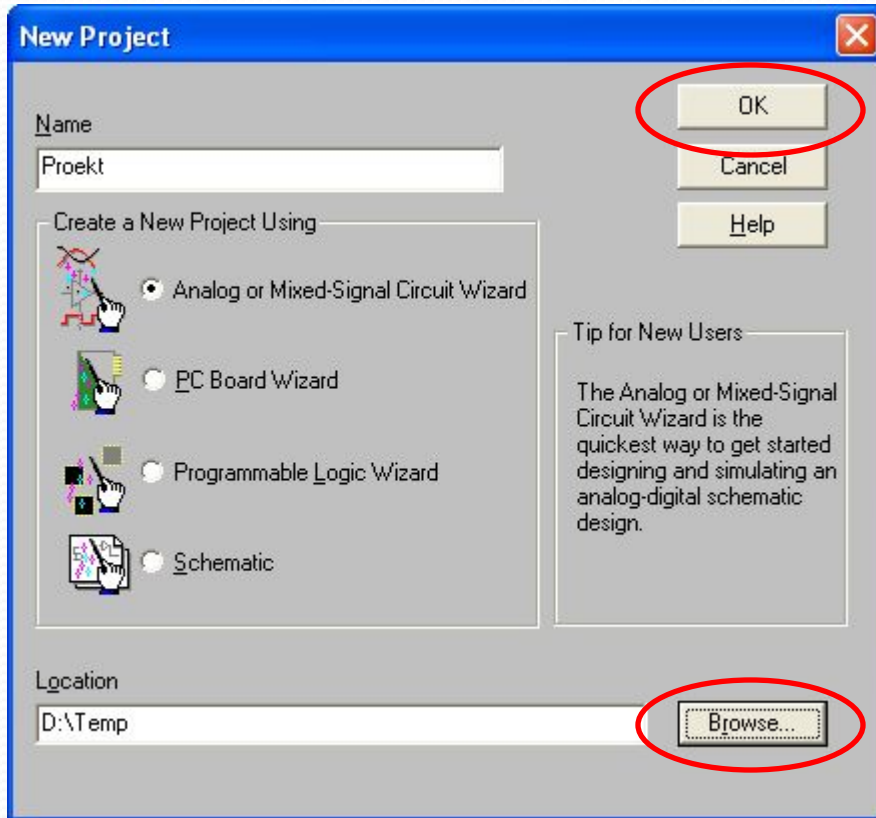


# Работа с Меню

Создание (открытие) нового проекта



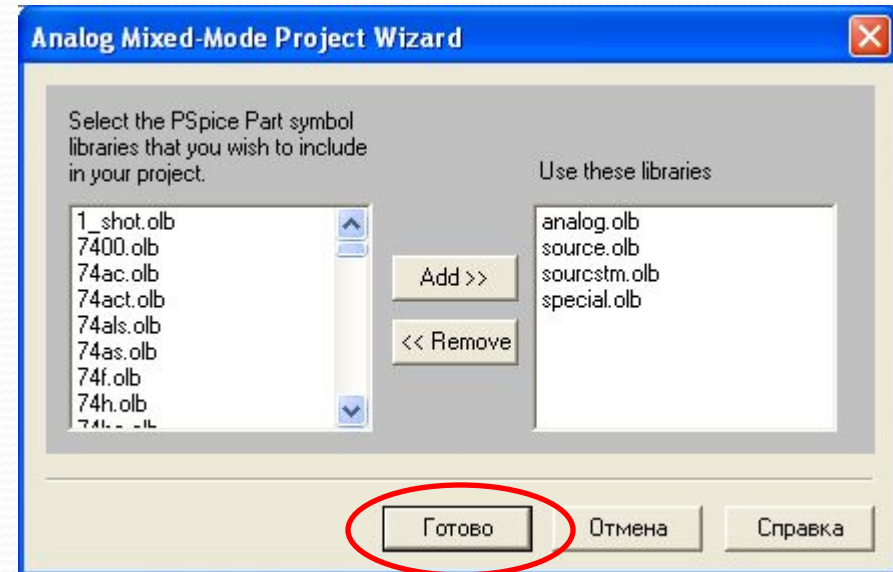
# Создание нового проекта



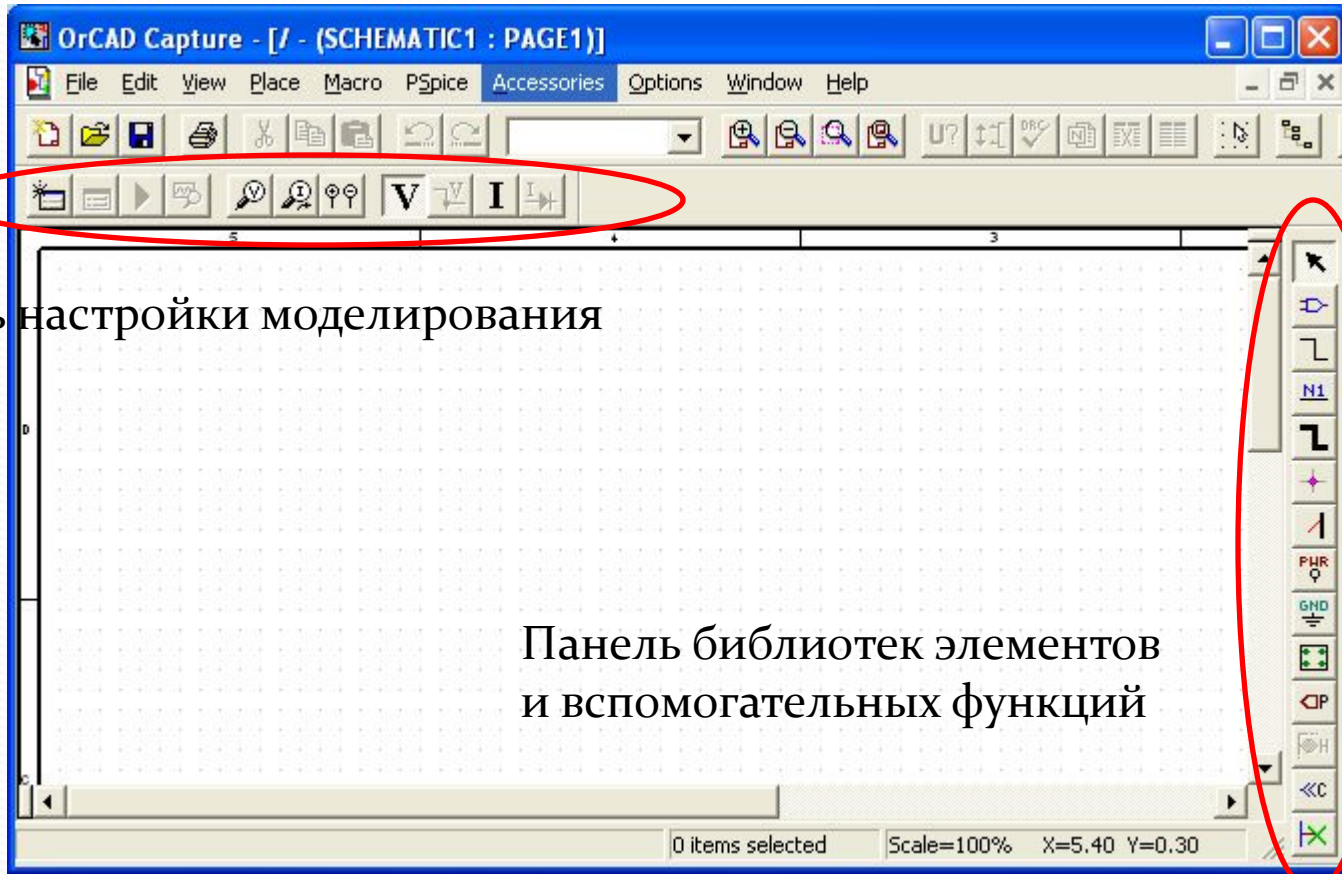
Выполняем процедуру представленную на предыдущем слайде, выбираем «Аналоговая или смешанная схема», указываем путь, нажимаем **ОК** (имя и путь **только латинскими** буквами)



После указания пути и нажатия ОК открывается окошко с открытыми по умолчанию библиотеками



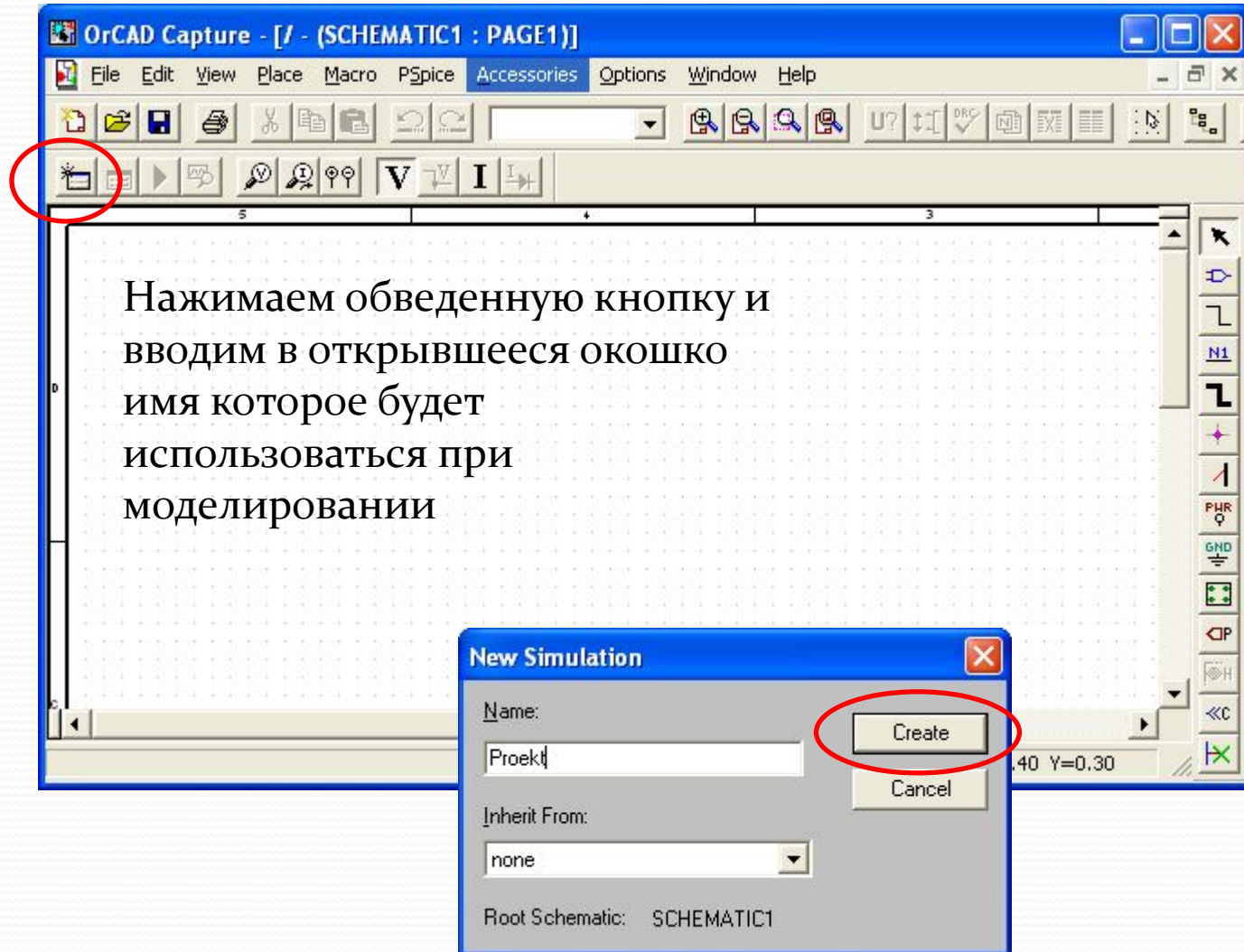
# Создание нового проекта



Панель настройки моделирования

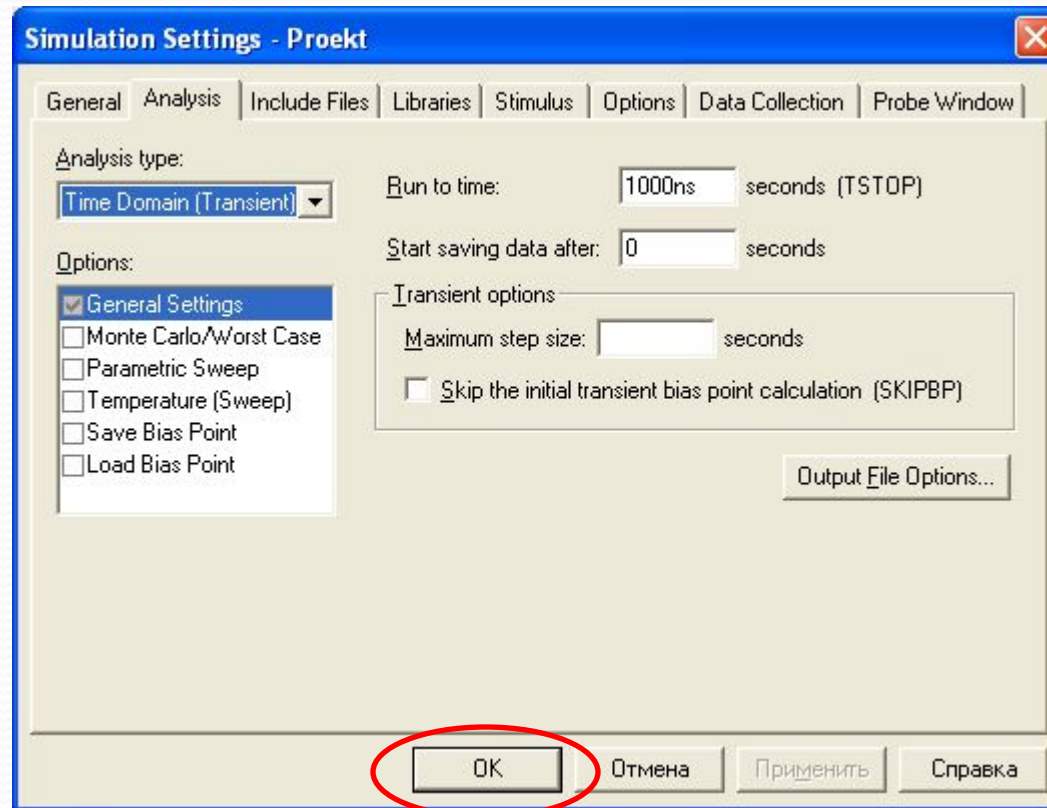
Панель библиотек элементов  
и вспомогательных функций

# Создание нового проекта



# Создание нового проекта

После задания имени профиля моделирования открывается окошко настройки параметров моделирования



Тип анализа

Время расчета

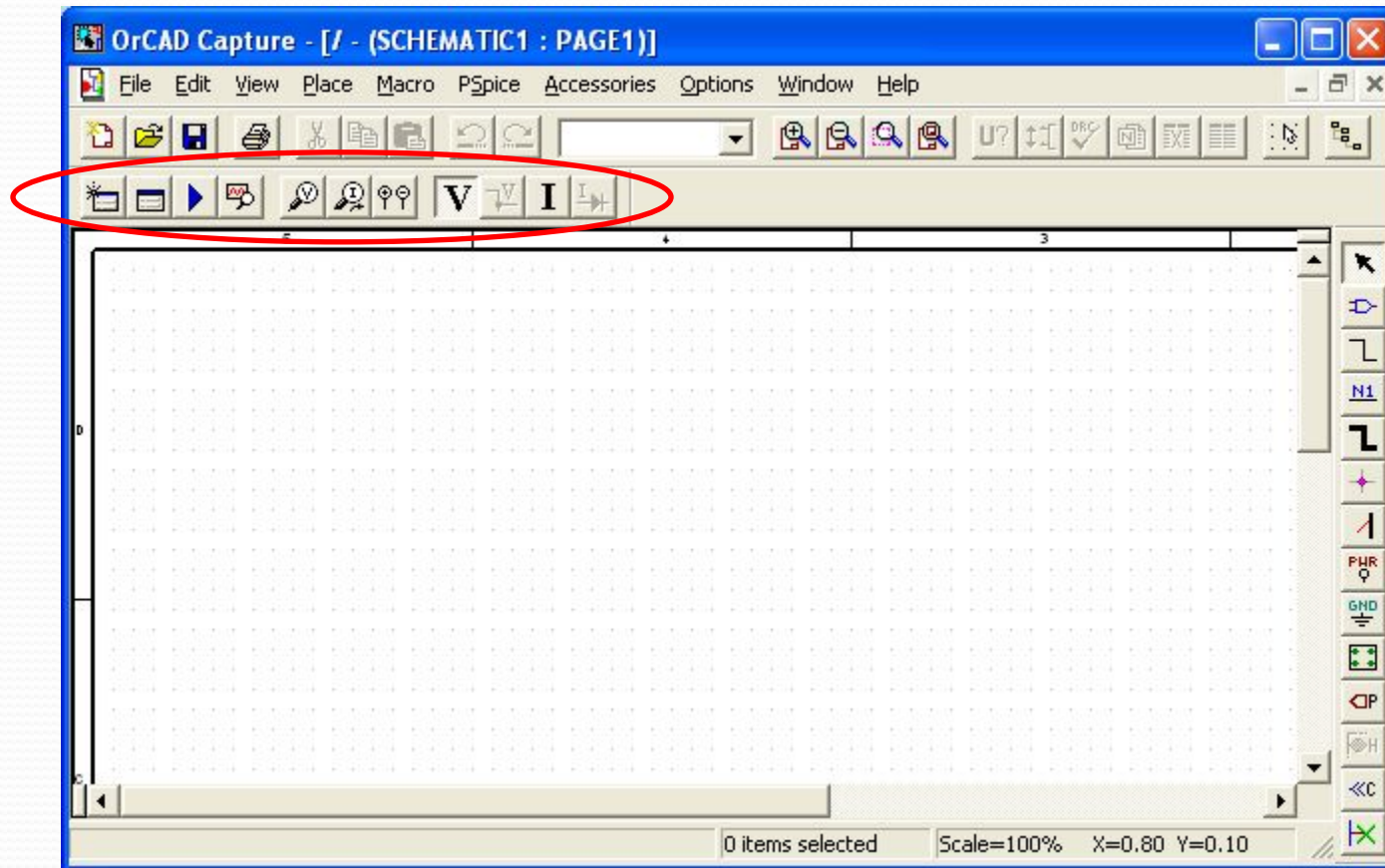
Опции анализа

Шаг расчета

Пока ничего не настраиваем и просто нажимаем ОК



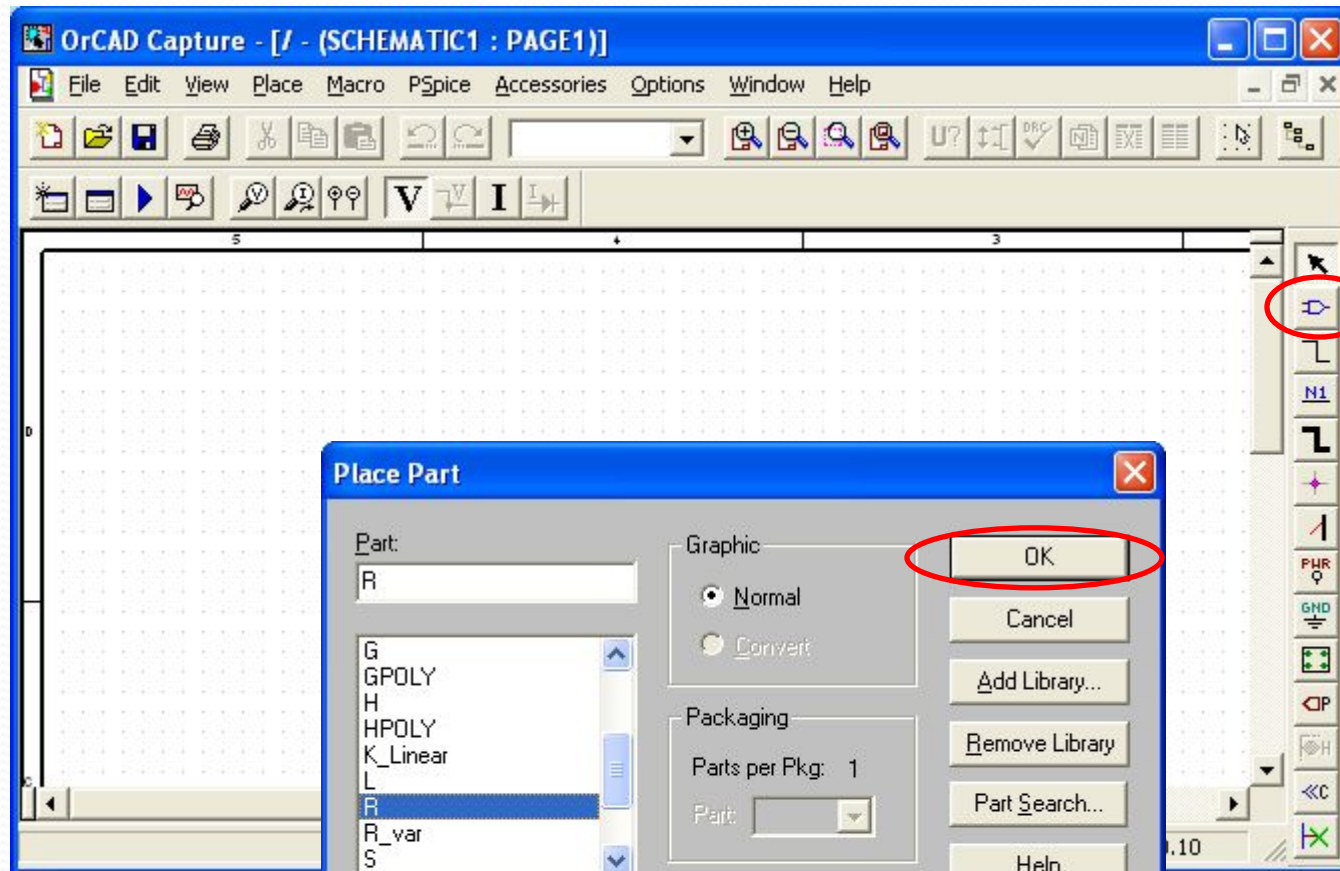
# Создание нового проекта



Обведенная панелька стала активной

Можно начинать рисовать схему

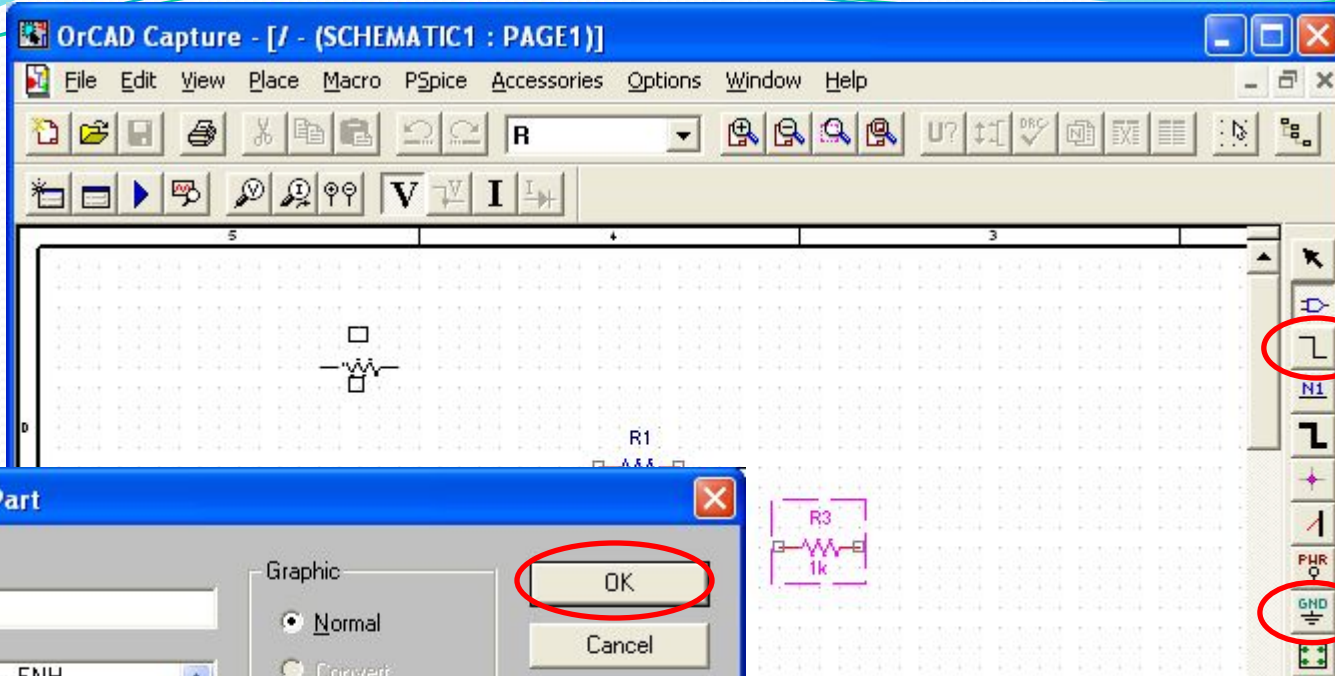
# Создание нового проекта



нажимаем

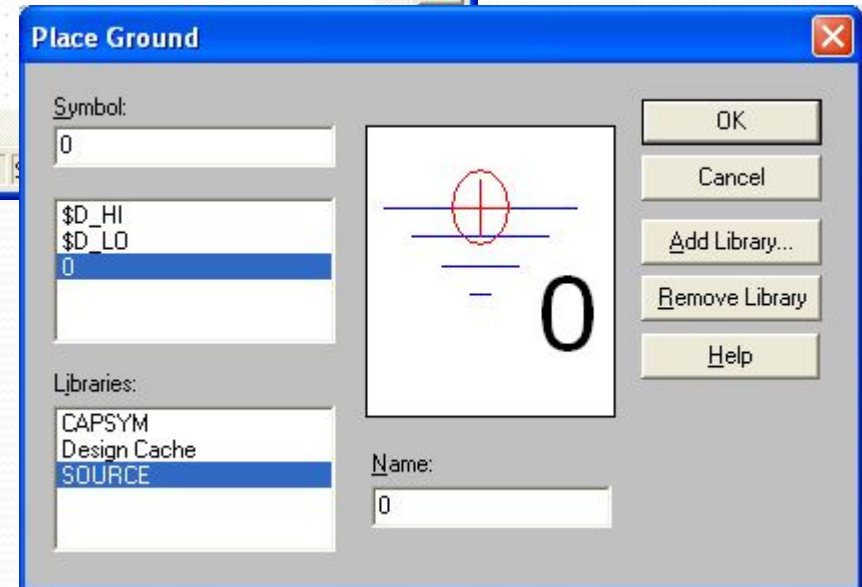
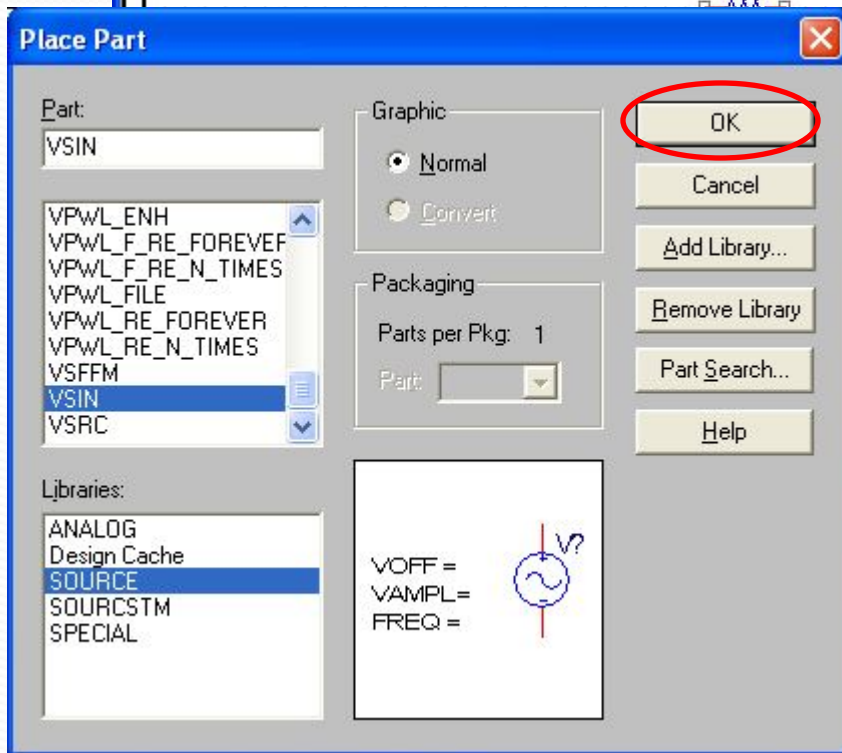
Выбираем  
элементы из  
библиотек

# Создание нового проекта

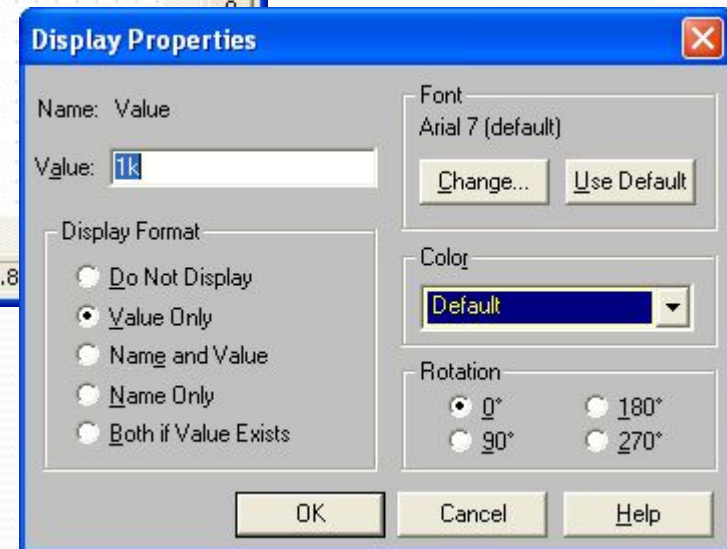
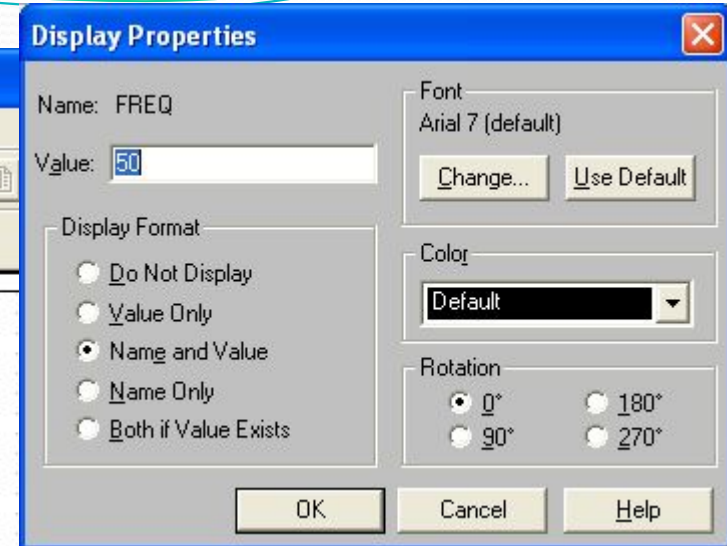
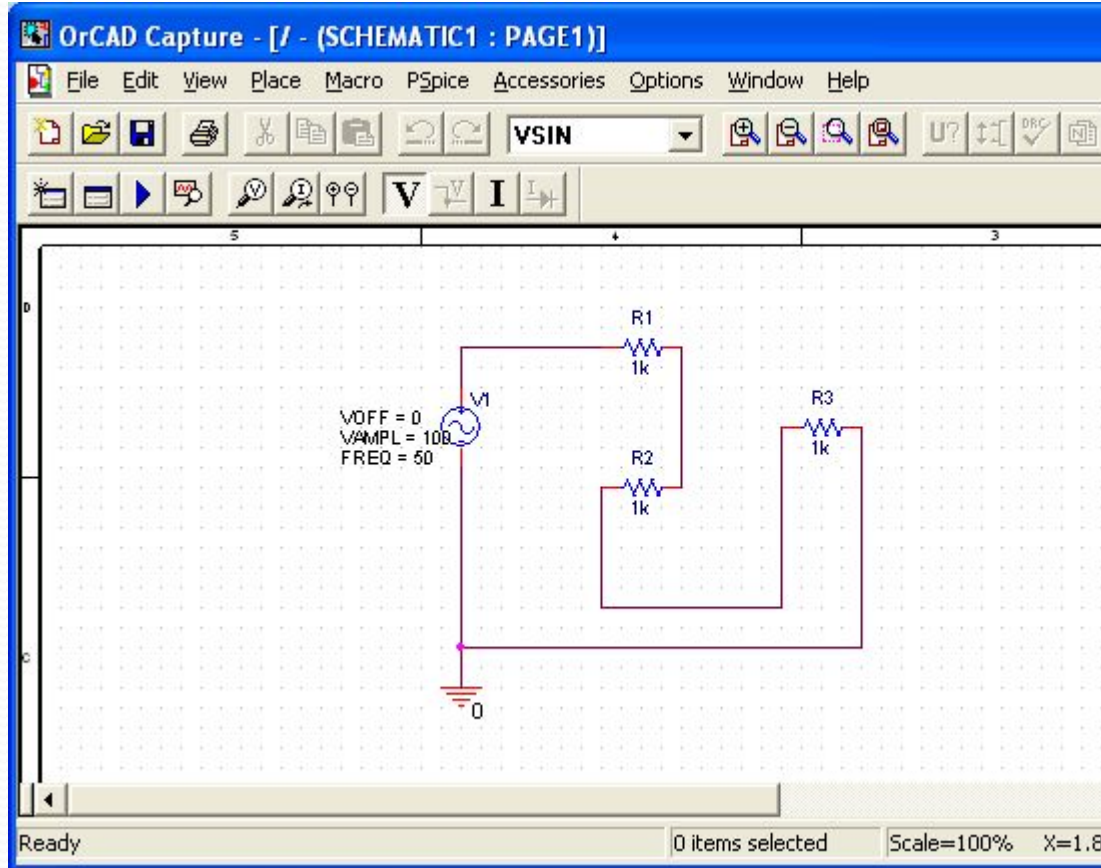


Соединение

Заземление



# Создание нового проекта



Изменение номинала резистора и настройки источника меняются путем двукратного нажатия левой клавиши мыши

# Создание нового проекта

## Настройки моделирования во временной области

Жмем

The image shows the OrCAD Capture interface with a schematic diagram and a 'Simulation Settings - Proekt' dialog box. The schematic includes a sine wave source labeled 'V1' with parameters:  $V_{OFF} = 0$ ,  $V_{AMPL} = 100$ , and  $FREQ = 50$ . The dialog box is open to the 'Analysis' tab, showing the following settings:

- Analysis type: Time Domain (Transient)
- Run to time: 100m seconds (TSTOP)
- Start saving data after: 0 seconds
- Options:  General Settings,  Monte Carlo/Worst Case,  Parametric Sweep,  Temperature (Sweep),  Save Bias Point,  Load Bias Point
- Transient options:  Skip the initial transient bias point calculation (SKIPBP)
- Maximum step size: 0.1m seconds
- Output File Options... button

At the bottom of the dialog box, the 'OK' and 'Применить' (Apply) buttons are circled in red. The 'Отмена' (Cancel) and 'Справка' (Help) buttons are also visible.

# Создание нового проекта

## Моделирование и вывод результата

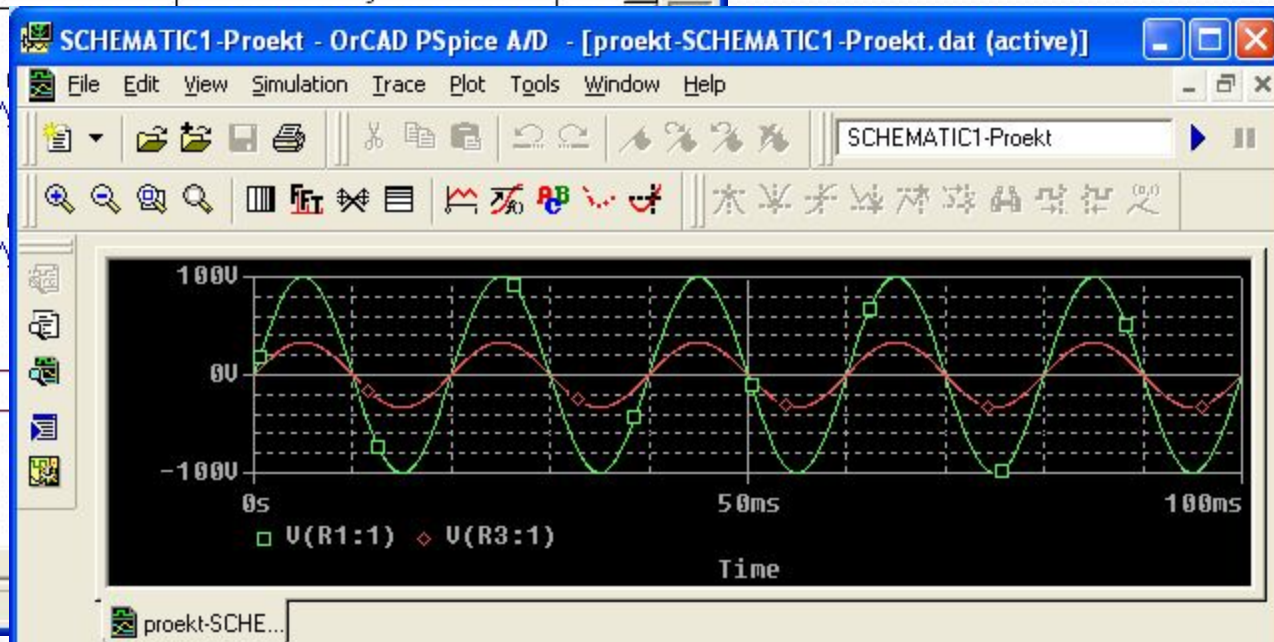
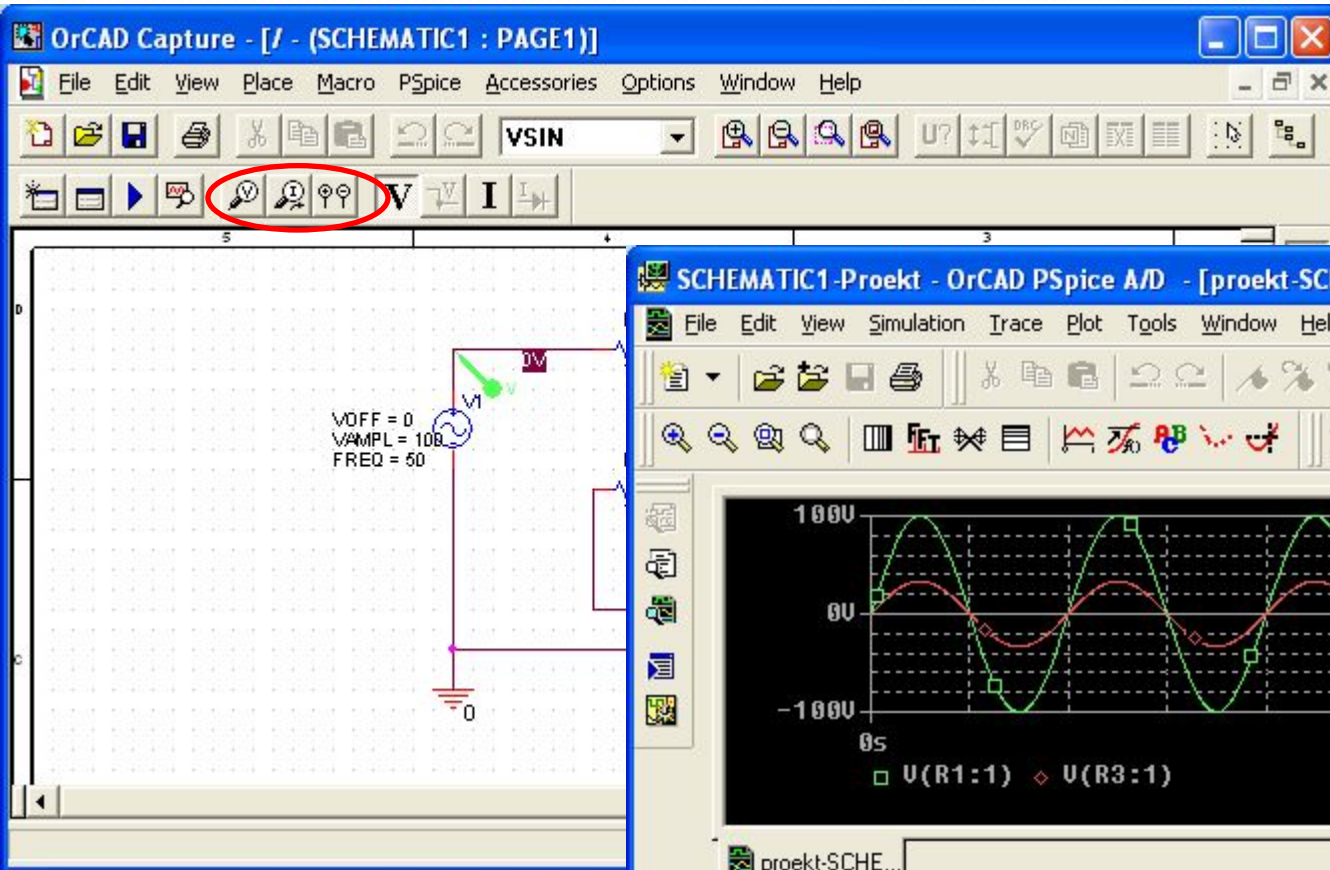
The image displays two overlapping windows from the OrCAD software suite. The top window, titled "OrCAD Capture - [ / - (SCHEMATIC1 : PAGE1)]", shows a schematic diagram of a circuit. A voltage source is connected to a ground symbol. The voltage source parameters are:  $V_{OFF} = 0$ ,  $V_{AMPL} = 100$ , and  $FREQ = 50$ . A red circle highlights the simulation button (a play icon) in the toolbar. The bottom window, titled "SCHEMATIC1-Proekt - OrCAD PSpice A/D - [proekt-SCHEMATIC1-Proekt.dat (active)]", shows the simulation results. The plot area is empty, with a time axis ranging from 0s to 100ms. The status bar at the bottom of the PSpice window displays the following text:

```
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point for Transient Analy
Bias point calculated
Transient Analysis
Transient Analysis finished
Simulation complete
```

Below the plot area, the simulation parameters are shown: "Time step = 72.00E-06" and "Time = .1". The status bar at the bottom of the PSpice window shows "Time = .1" and "100%".

# Создание нового проекта

Моделирование и вывод результата



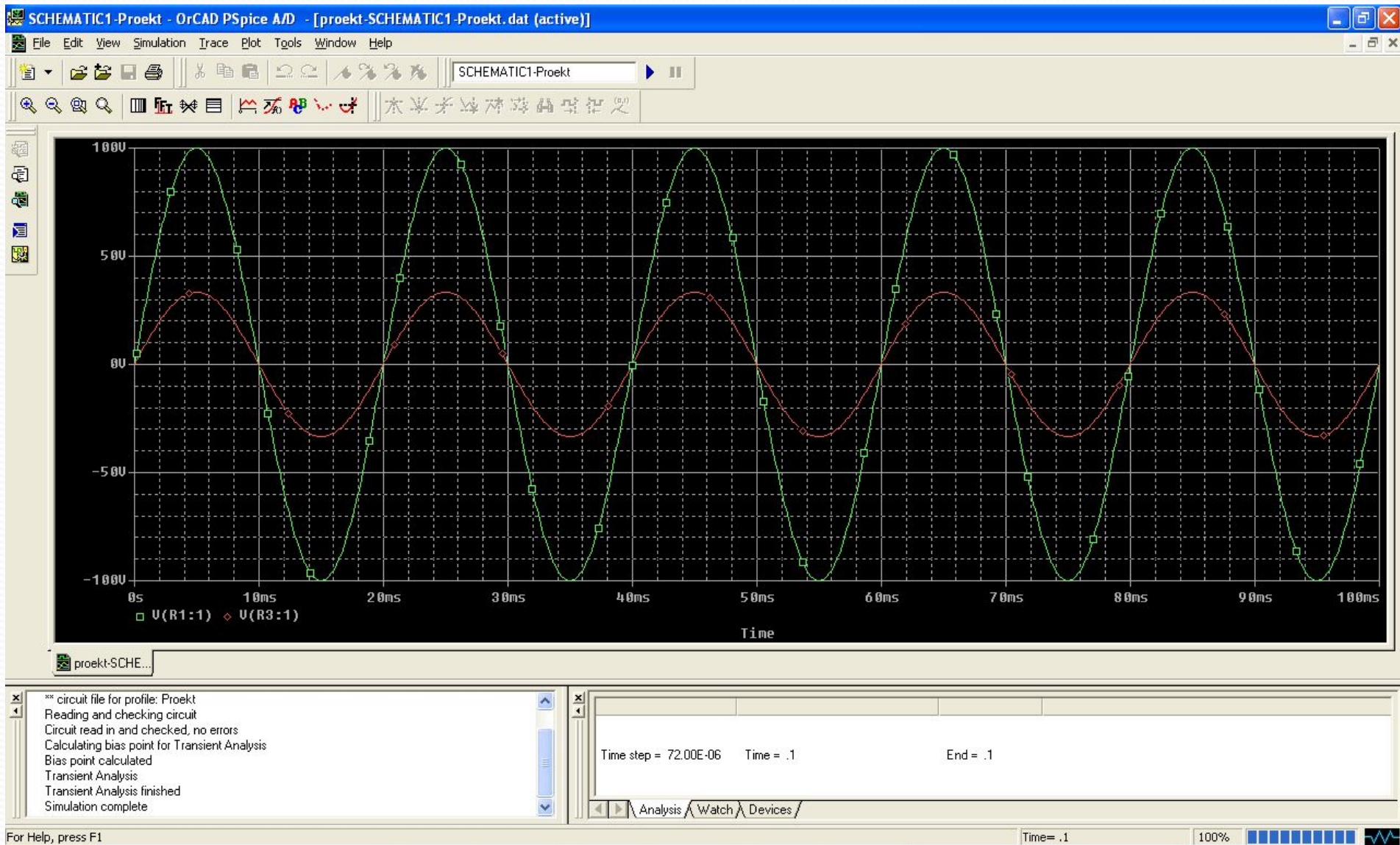
Reading and checking circuit  
Circuit read in and checked, no errors  
Calculating bias point for Transient Analy  
Bias point calculated  
Transient Analysis  
Transient Analysis finished  
Simulation complete

Time step = 72.00E-06 Time = .1 Enc

Analysis Watch Devices

# Создание нового проекта

## Моделирование и вывод результата





# Создание нового проекта

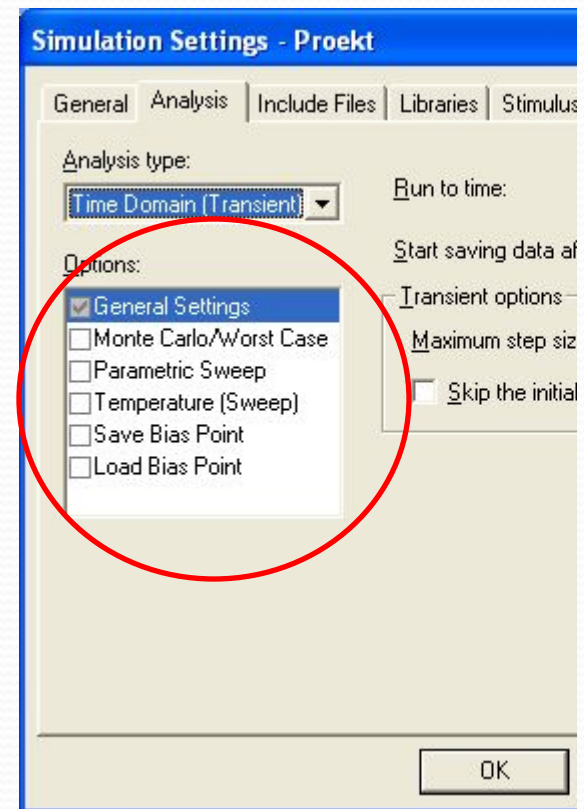
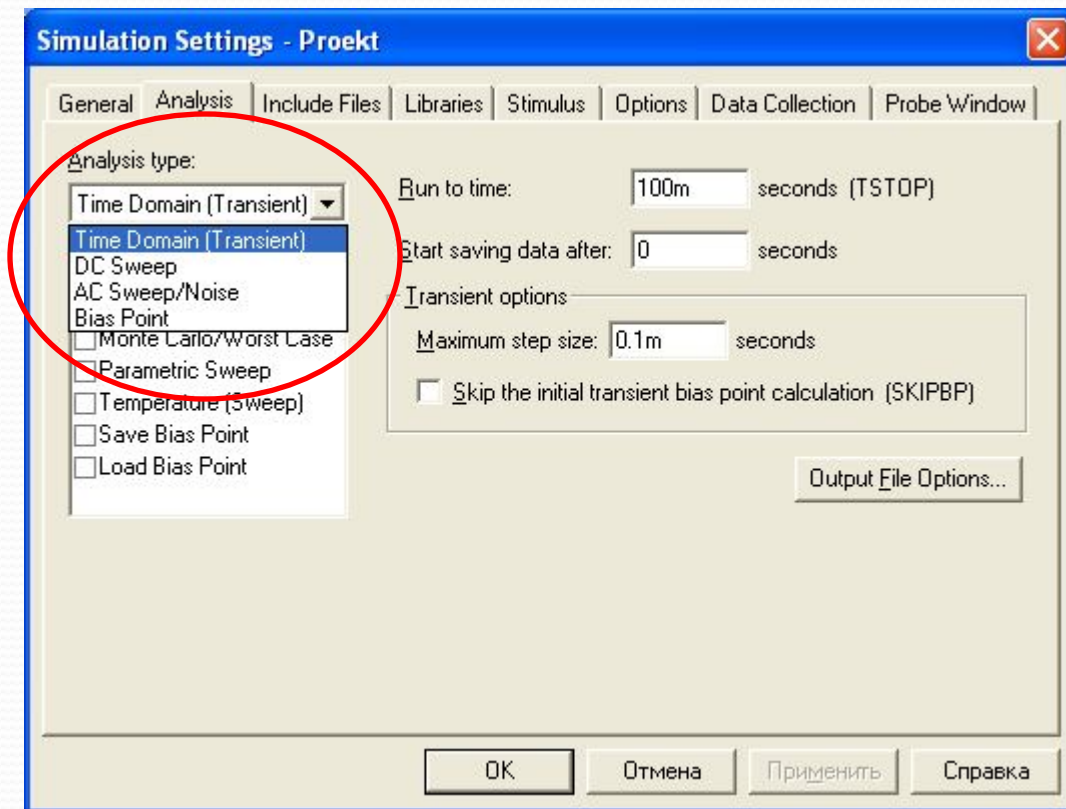
## Моделирование и вывод результата

The screenshot shows the OrCAD PSpice A/D interface. The main window displays a simulation plot with a green sine wave and a red curve. The y-axis ranges from -100U to 100U, and the x-axis is labeled '0s'. A legend at the bottom of the plot shows a green square for 'U(R1:1)' and a red diamond for 'U(R3:1)'. The 'Trace' menu is open, showing options: 'Add Trace...', 'Delete All Traces', 'Undelete Traces', 'Fourier', 'Performance Analysis...', 'Cursor', 'Macros...', 'Goal Functions...', and 'Eval Goal Function...'. The status bar at the bottom indicates 'Add trace[s] to the selected plot' and 'Time= .1'.

The 'Add Traces' dialog box is open, showing a list of simulation output variables. The 'Simulation Output Variables' list includes: \*, I(R1), I(R2), I(R3), I(V1), Time, V(0), V(N00011), V(N00017), V(N00020), V(R1:1), V(R1:2), V(R2:1), V(R2:2), V(R3:1), V(R3:2), V(V1:+), V(V1:-), V1(R1), V1(R2), V1(R3), V1(V1), V2(R1), and V2(R2). The 'Functions or Macros' section is set to 'Analog Operators and Functions' and lists various mathematical functions: #, (), \*, +, -, /, @, ABS(), ARCTAN(), ATAN(), AVG(), AVGX(.), COS(), D(), DB(), ENVMAX(.), ENVMIN(.), EXP(), G(), IMG(), LOG(), LOG10(), M(), and MAX(). The 'Simulation Output Variables' section has checkboxes for 'Analog' (checked), 'Digital' (unchecked), 'Voltages' (checked), 'Currents' (checked), 'Noise (V7/Hz)' (unchecked), 'Alias Names' (checked), and 'Subcircuit Nodes' (unchecked). The text '23 variables listed' is shown at the bottom of the list. The 'Trace Expression:' field is empty. The dialog has 'OK', 'Cancel', and 'Help' buttons.

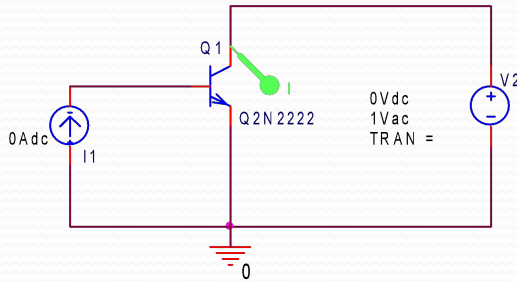
# Виды моделирования

1. Моделирование во временной области (переходные процессы)
2. Моделирование в режиме по постоянному току
3. Моделирование в частотной области
4. Параметрическое моделирование
5. Статистическое моделирование



# Виды моделирования

Расчет по постоянному току.  
Снятие ВАХ полупроводниковых приборов



Simulation Settings - Proekt

General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window

Analysis type:  
DC Sweep

Options:  
 Primary Sweep  
 Secondary Sweep

Sweep variable:  
 Voltage source Name: V2  
 Current source Model type:  
 Global parameter Model name:  
 Model parameter Parameter name:  
 Temperature

Sweep type:  
 Linear Start value: 0 End value: 15 Increment: 0.1  
 Logarithmic Decade  
 Value list

OK Отмена Применить Справка

Simulation Settings - Proekt

General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window

Analysis type:  
DC Sweep

Options:  
 Primary Sweep  
 Secondary Sweep  
 Monte Carlo/Worst Case  
 Parametric Sweep  
 Temperature (Sweep)  
 Save Bias Point  
 Load Bias Point

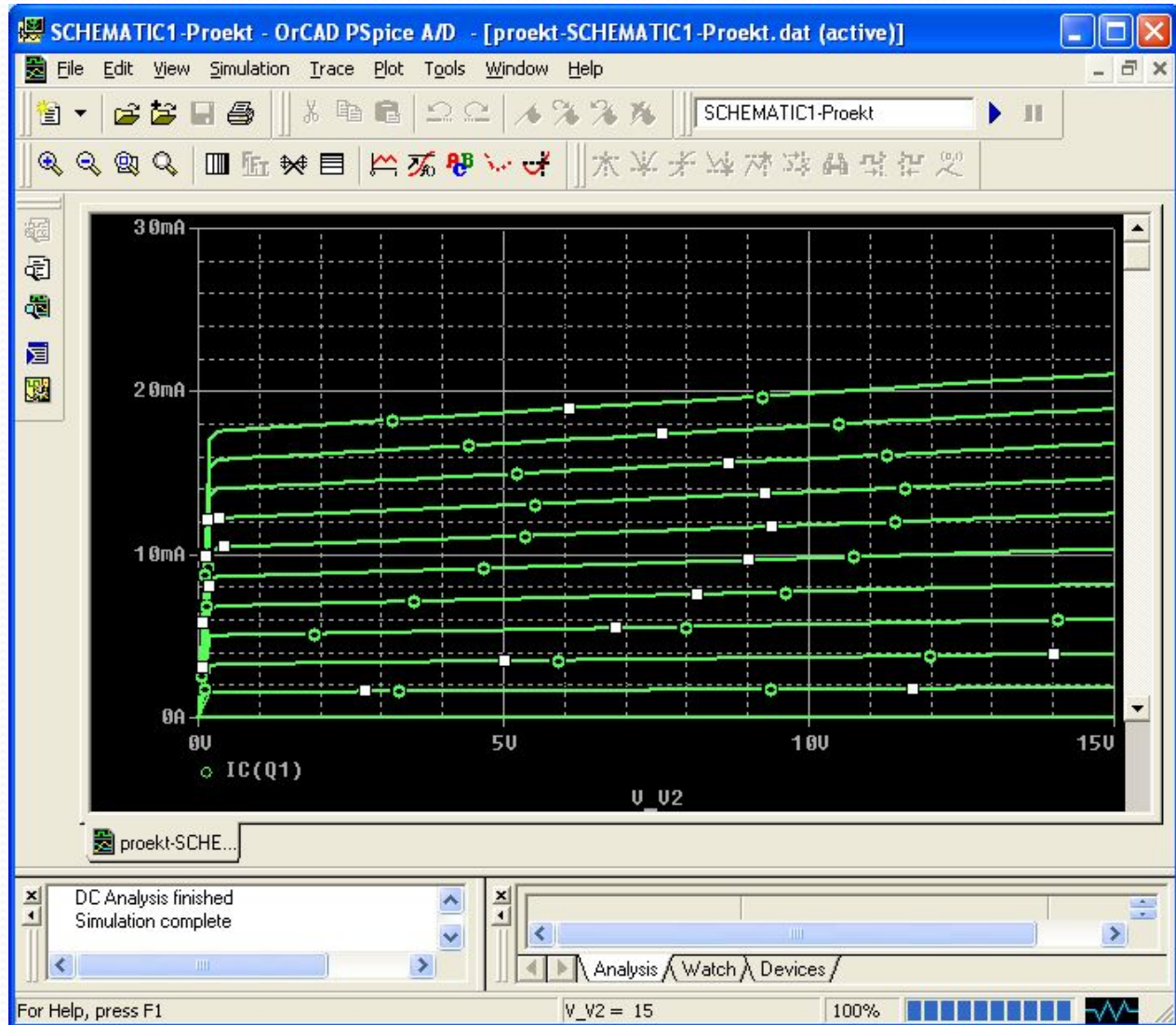
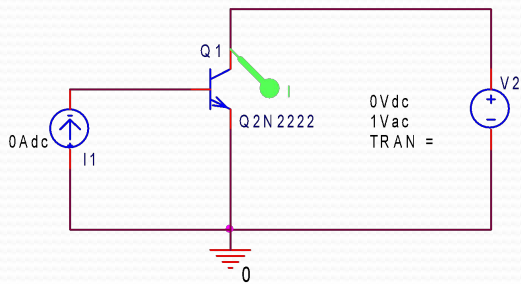
Sweep variable:  
 Voltage source Name: I1  
 Current source Model type:  
 Global parameter Model name:  
 Model parameter Parameter name:  
 Temperature

Sweep type:  
 Linear Start value: 0 End value: 0.1m Increment: 0.01m  
 Logarithmic Decade  
 Value list

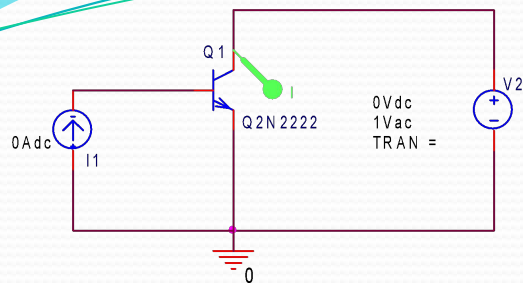
OK Отмена Применить Справка

# Виды моделирования

Расчет по постоянному току.  
Снятие ВАХ полупроводниковых приборов



# Виды моделирования



Расчет по постоянному току.  
Снятие ВАХ полупроводниковых приборов.  
Температурный анализ.

**Simulation Settings - vax**

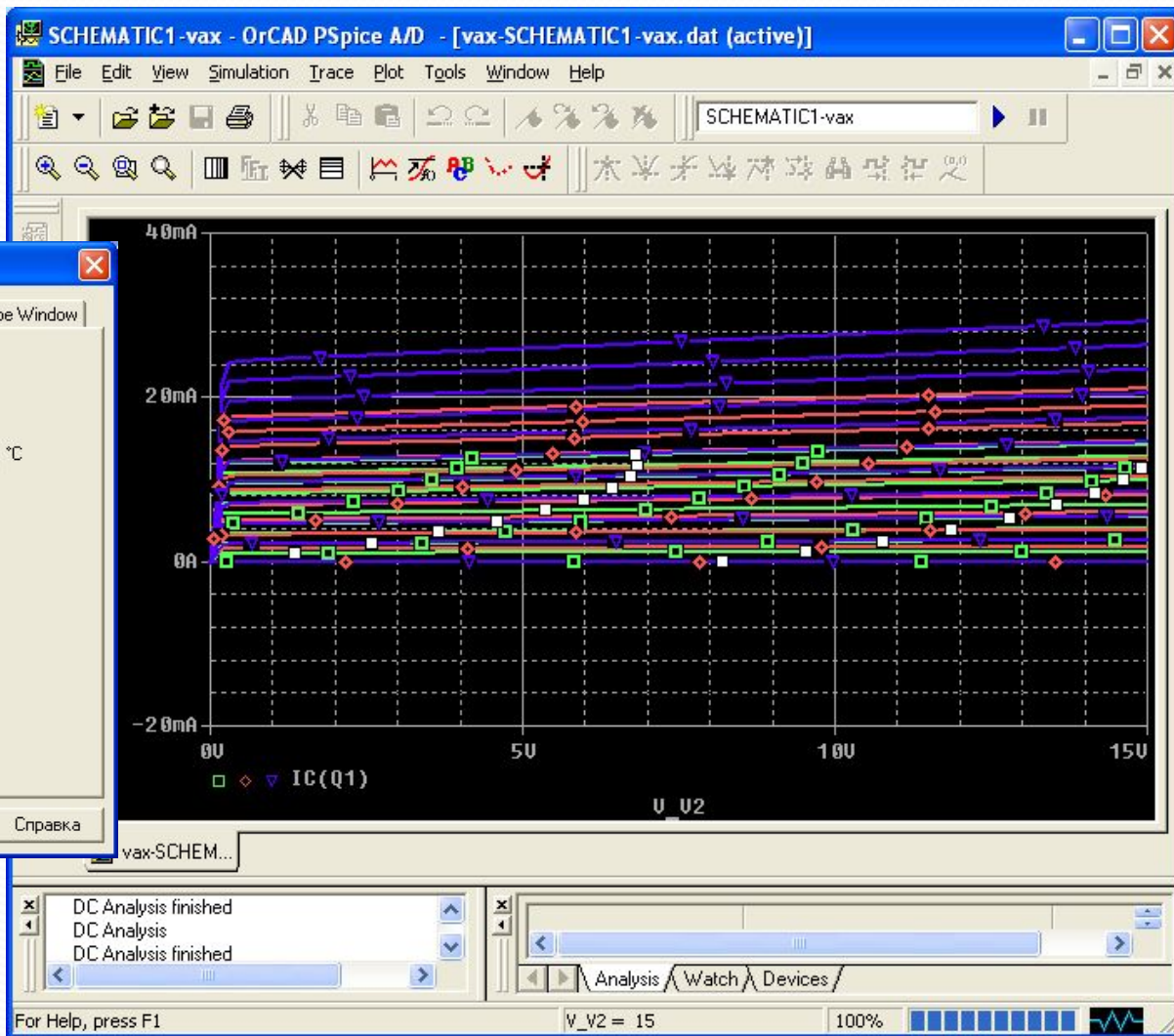
General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window

Analysis type:  
DC Sweep

Options:  
 Primary Sweep  
 Secondary Sweep  
 Monte Carlo/Worst Case  
 Parametric Sweep  
 Temperature (Sweep)  
 Save Bias Point  
 Load Bias Point

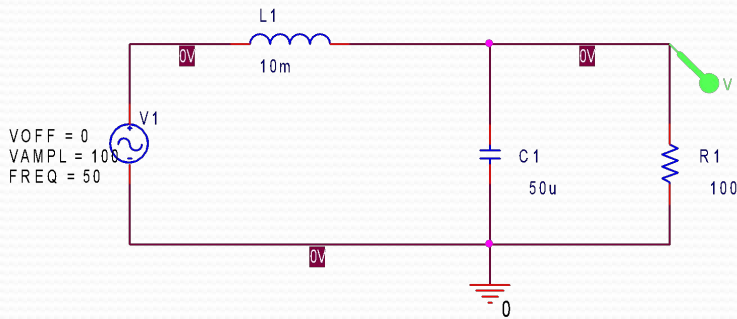
Run the simulation at temperature:  °C  
Repeat the simulation for each of the temperatures:  
 °C  
Enter a list of temperatures, separated by spaces.  
For example, 0 27 125

OK Отмена Применить Справка

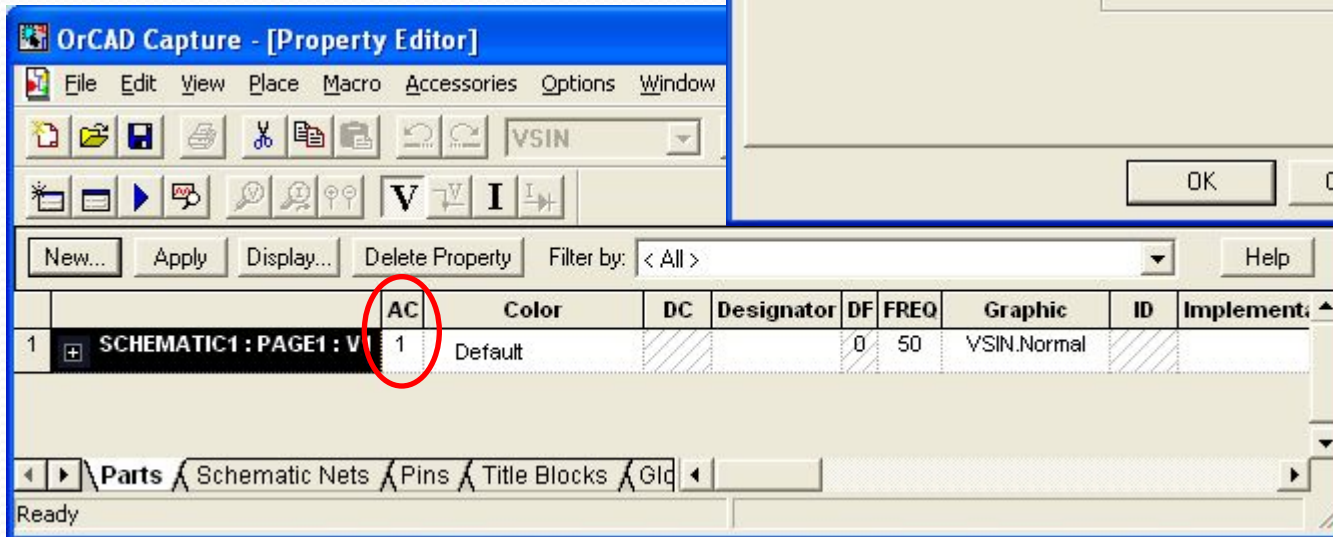
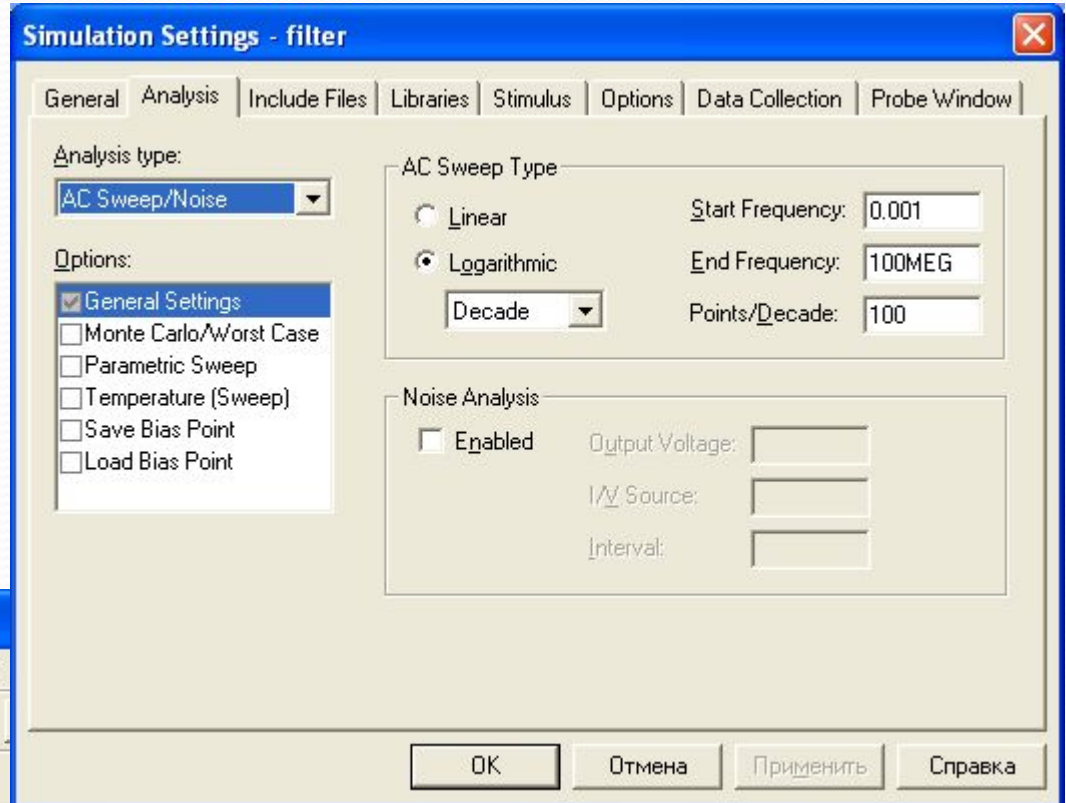


# Виды моделирования

## Моделирование в частотной области

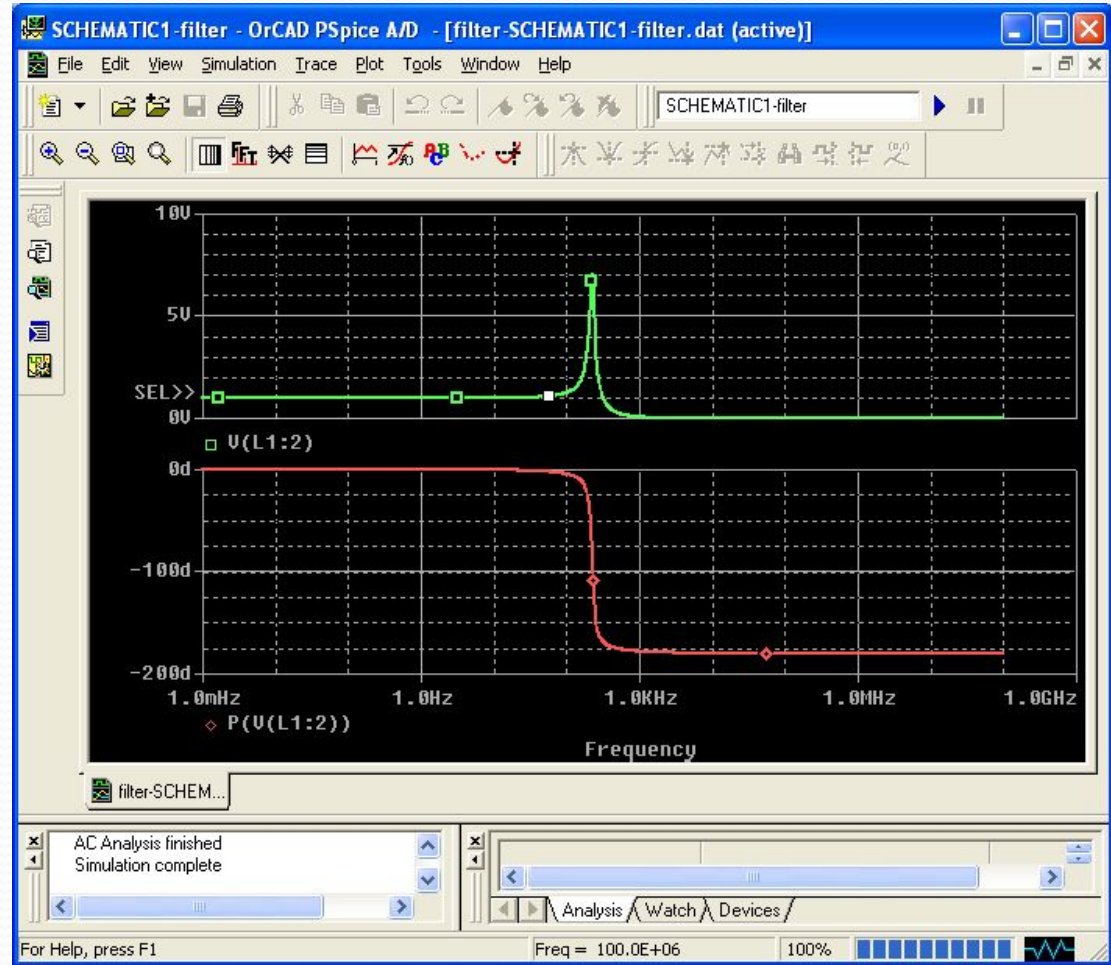
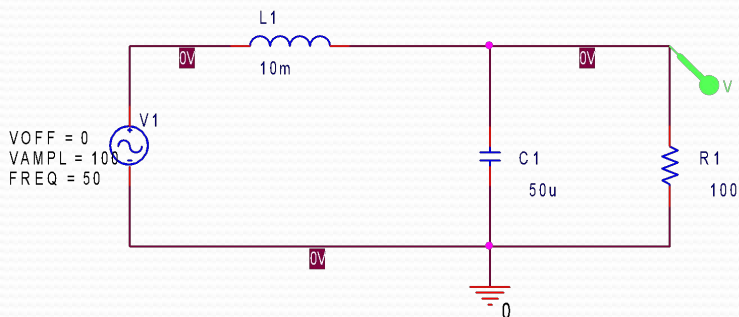


Дополнительные  
настройки источника



# Виды моделирования

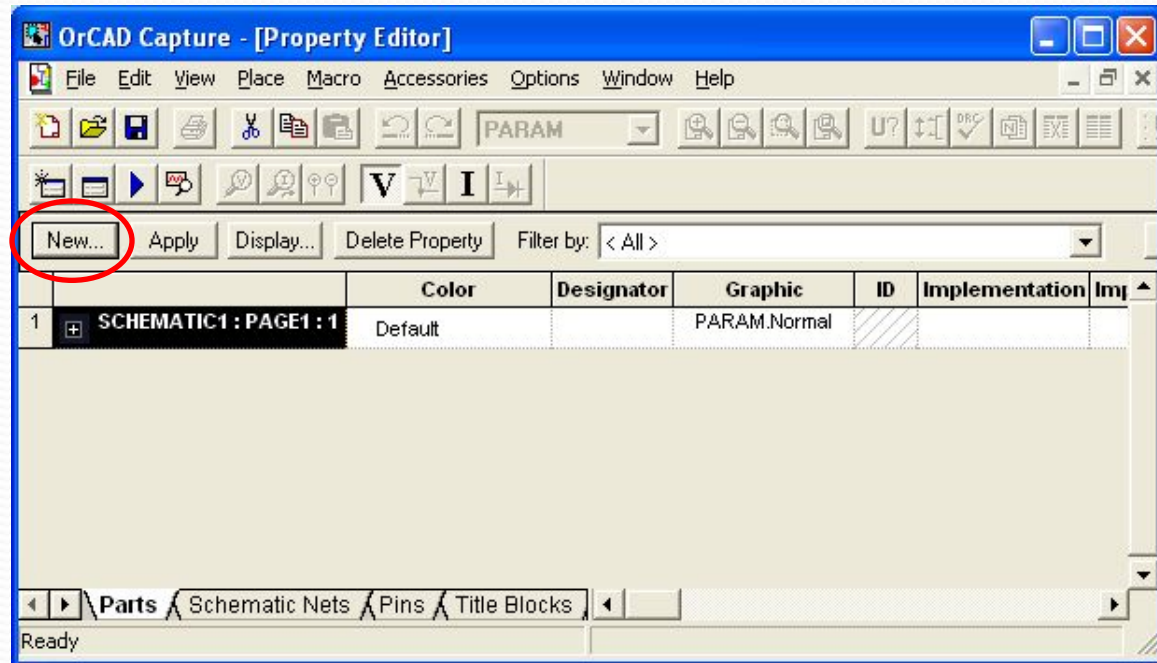
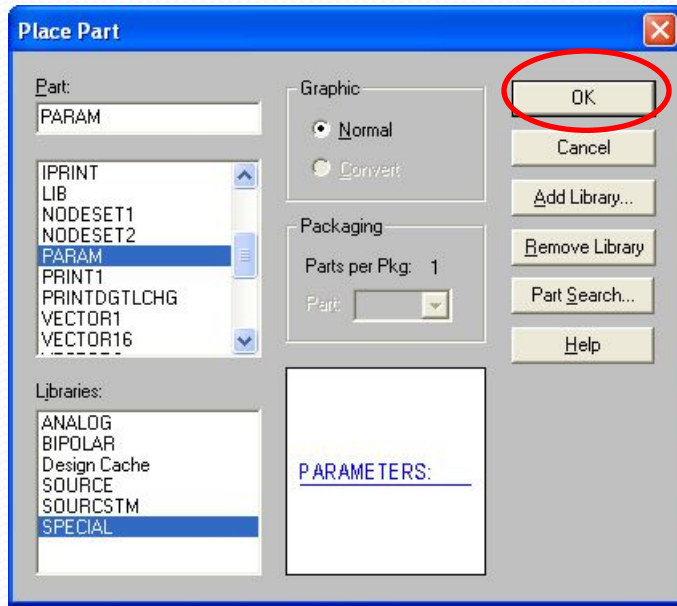
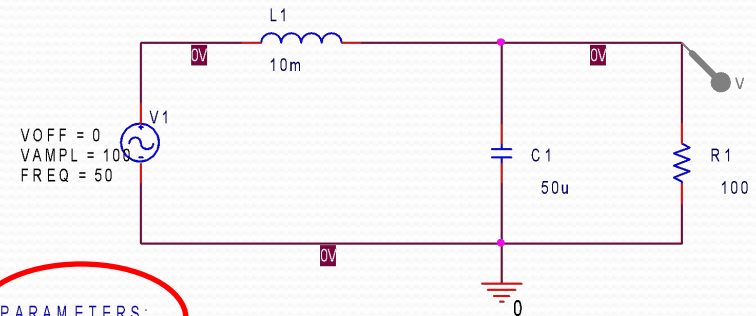
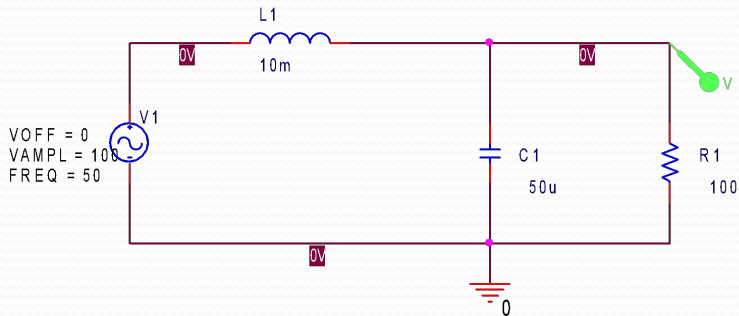
## Моделирование в частотной области



АЧХ и ФЧХ Фильтра

# Виды моделирования

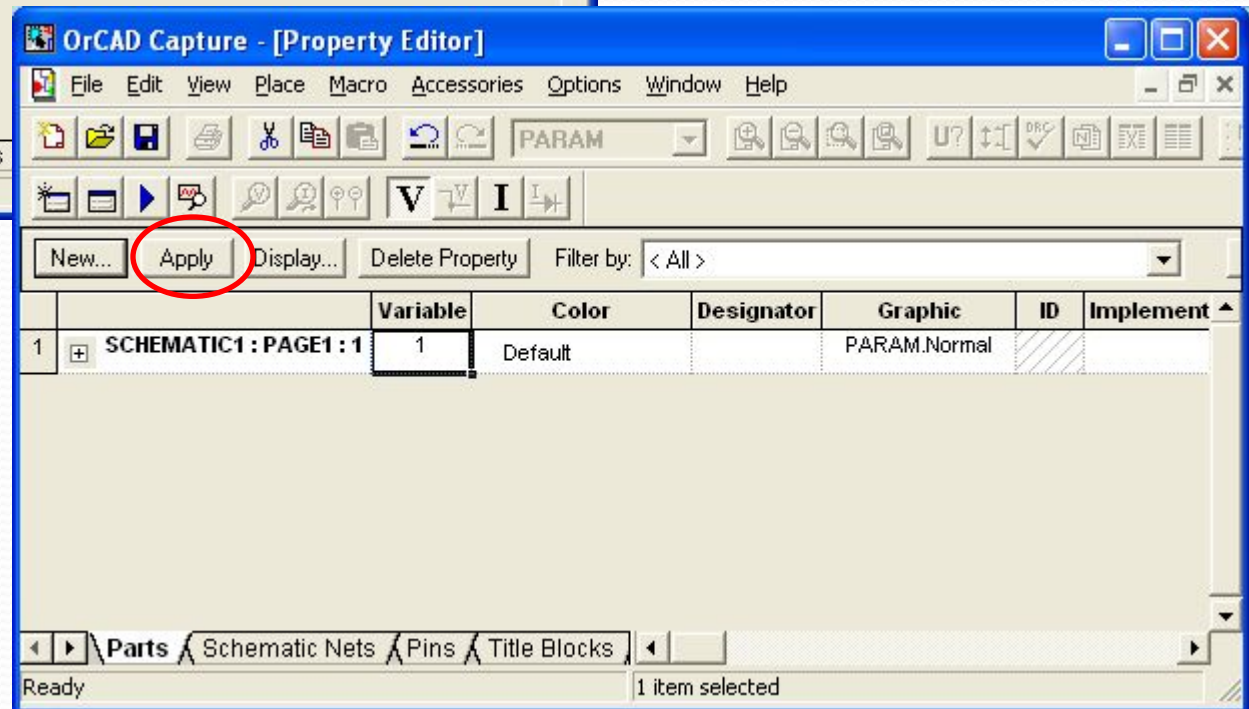
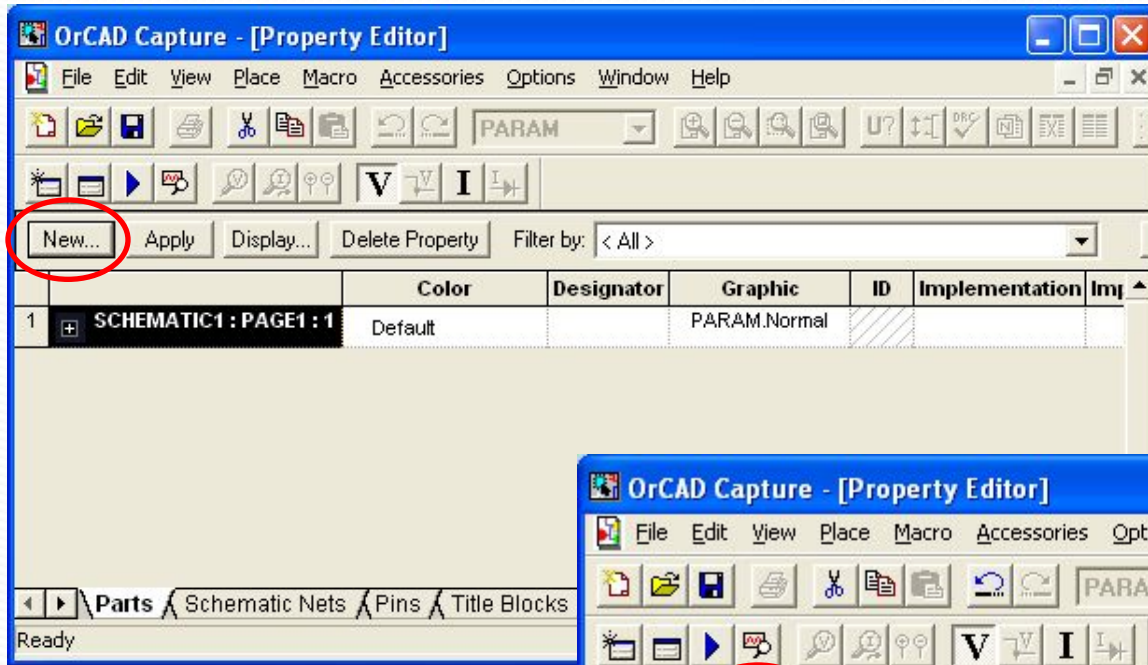
## Параметрическое моделирование





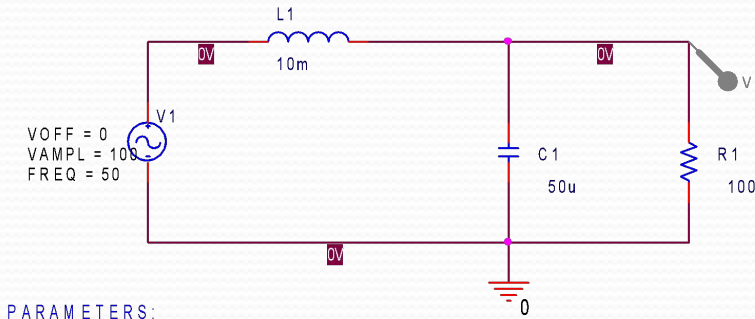
# Виды моделирования

## Параметрическое моделирование

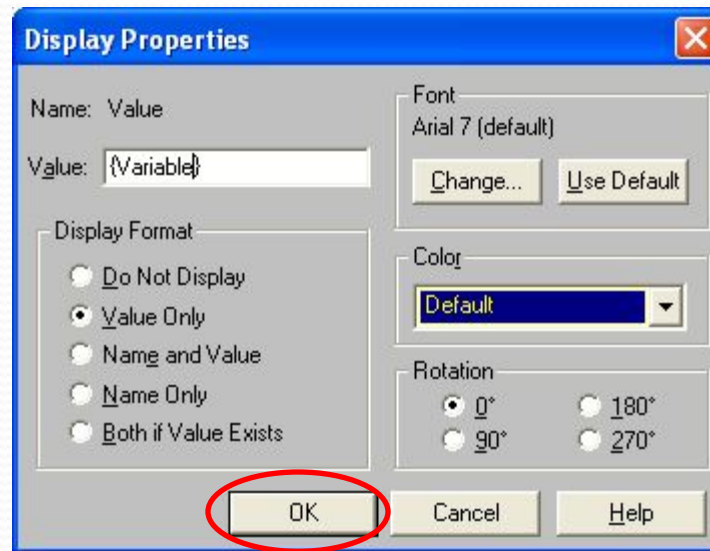


# Виды моделирования

## Параметрическое моделирование



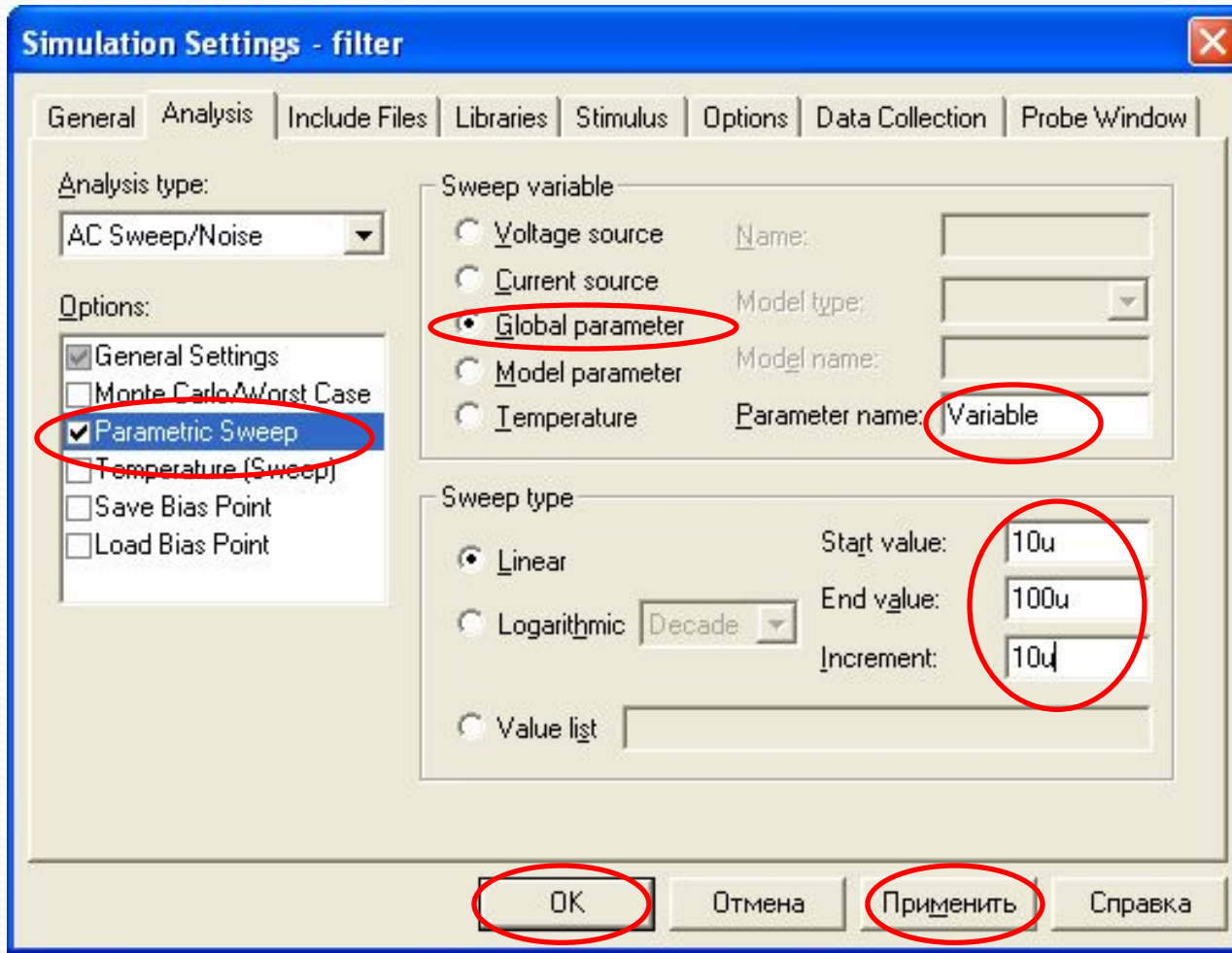
Далее щелкаем на номинале интересующего элемента (например конденсатора) и в фигурных скобках пишем имя переменной которую мы создали выше



Далее идем в настройки параметров моделирования и настраиваем

# Виды моделирования

## Параметрическое моделирование

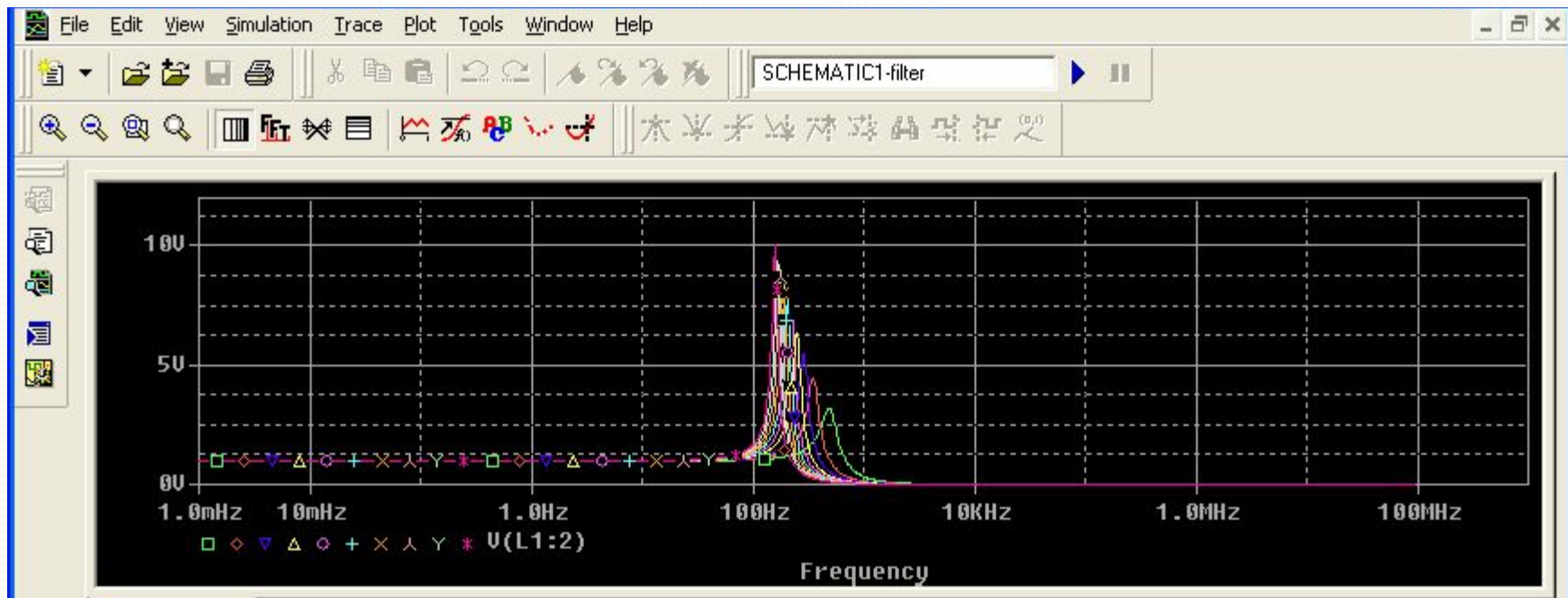
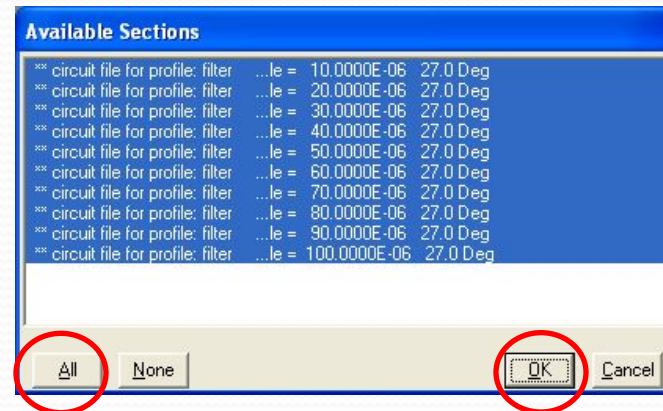


И запускаем схему на расчет

# Виды моделирования

## Параметрическое моделирование

Видим информацию о проделанной работе и выбираем интересующий диапазон (в большинстве случаев весь диапазон) и смотрим результат

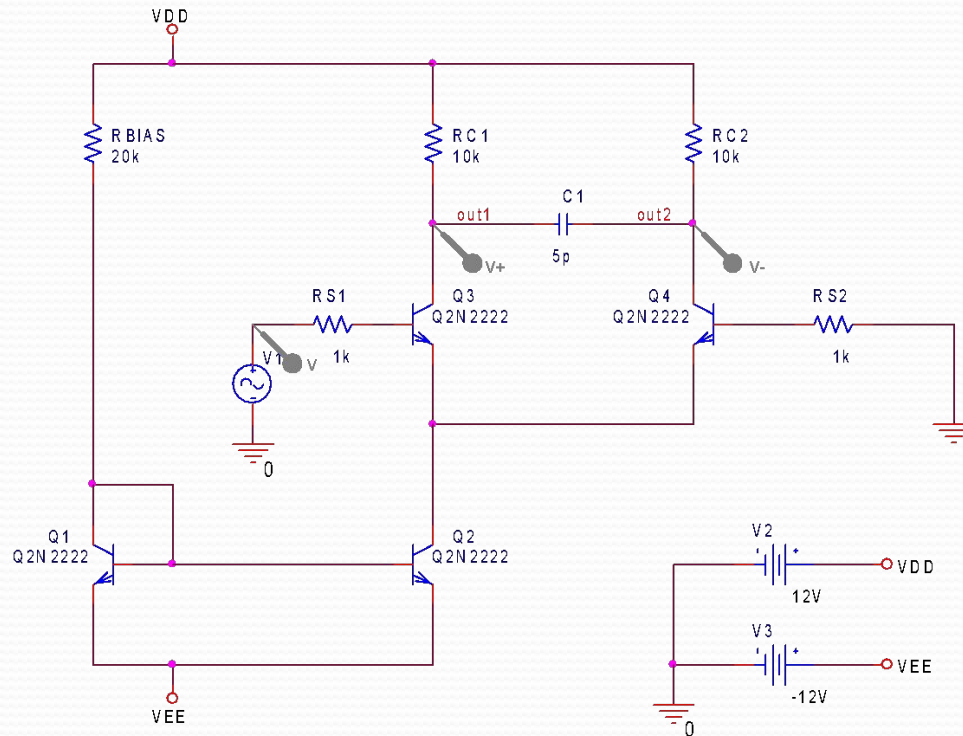


# Основы работы в OrCad

1. Статистический анализ
2. Создание и редактирование моделей компонентов ЭС
3. Оптимизация

# Виды моделирования

## Статистическое моделирование методом Монте-Карло (дифференциальный усилитель)



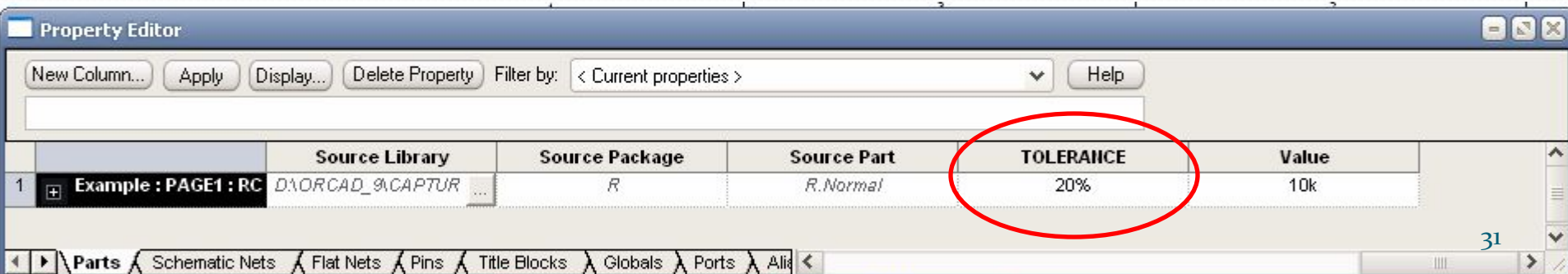
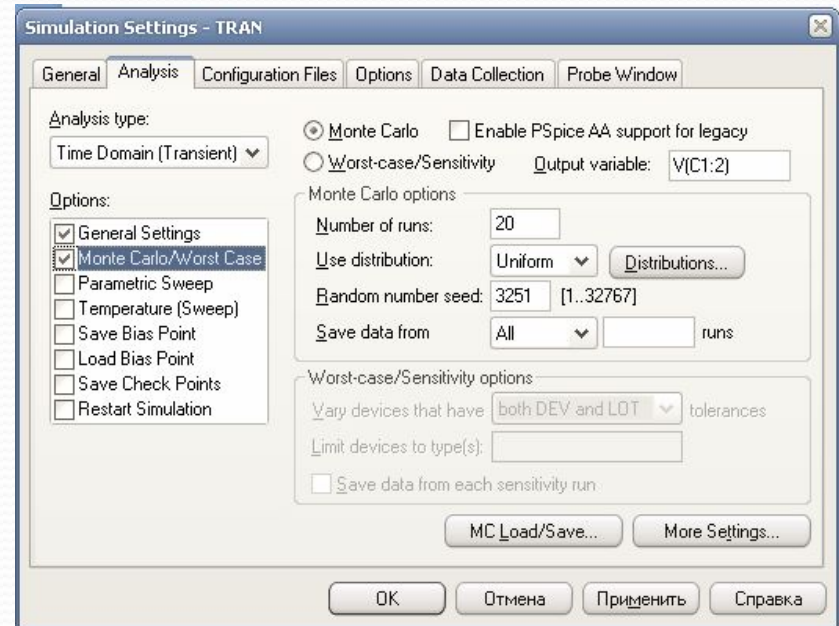
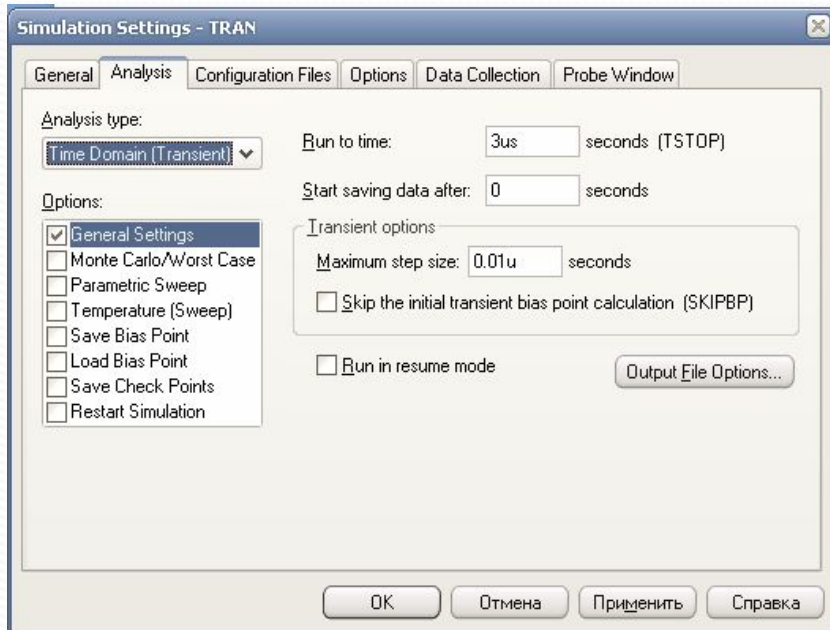
# Виды моделирования

Статистическое моделирование методом Монте-Карло  
(дифференциальный усилитель)

Настройки расчетов

во временной области

метода Монте-Карло

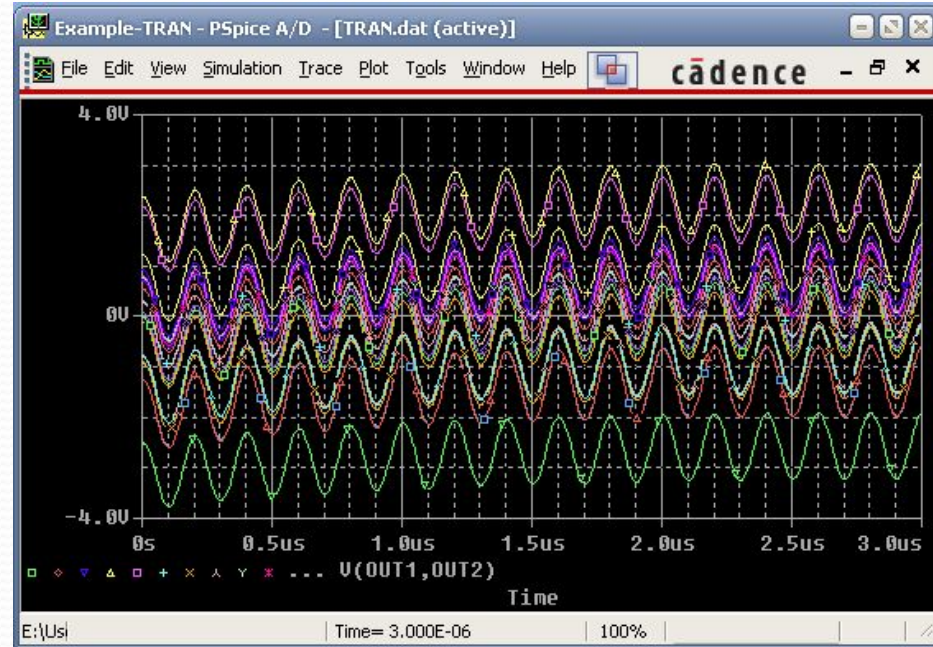
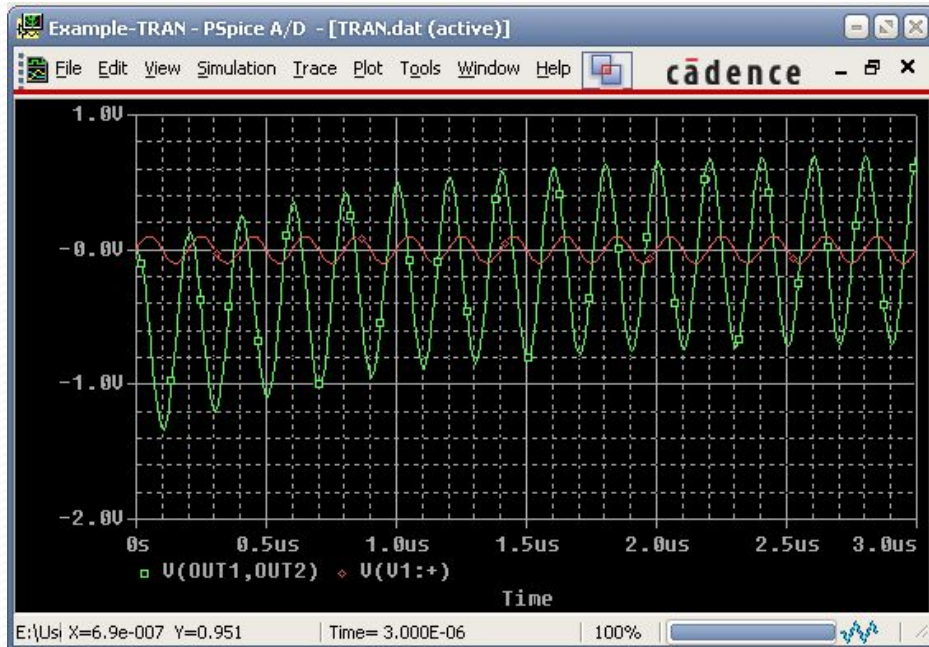


# Виды моделирования

Статистическое моделирование методом Монте-Карло  
(дифференциальный усилитель)

во временной области

Результаты расчетов  
метода Монте-Карло



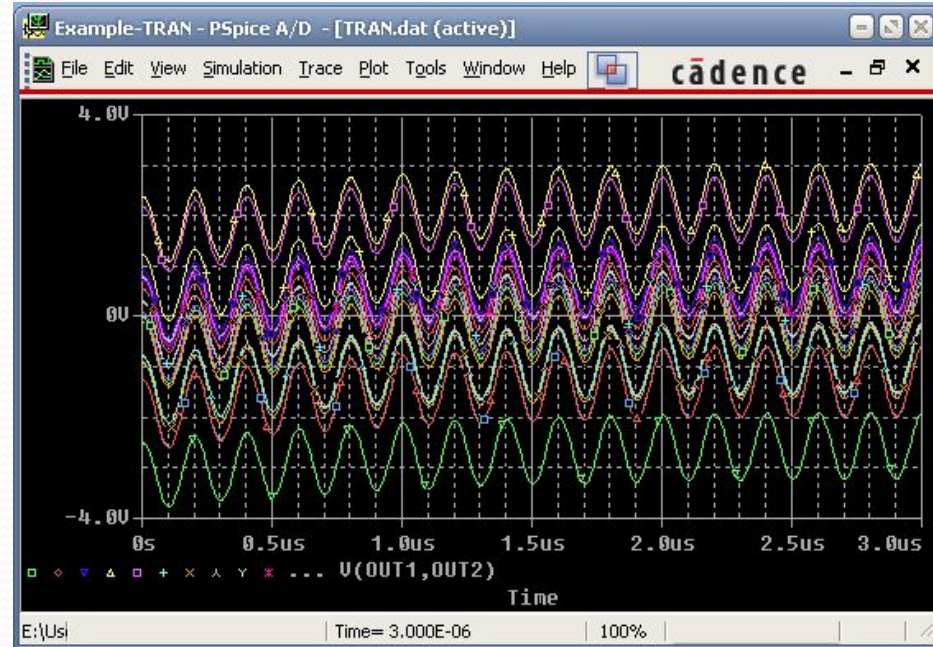
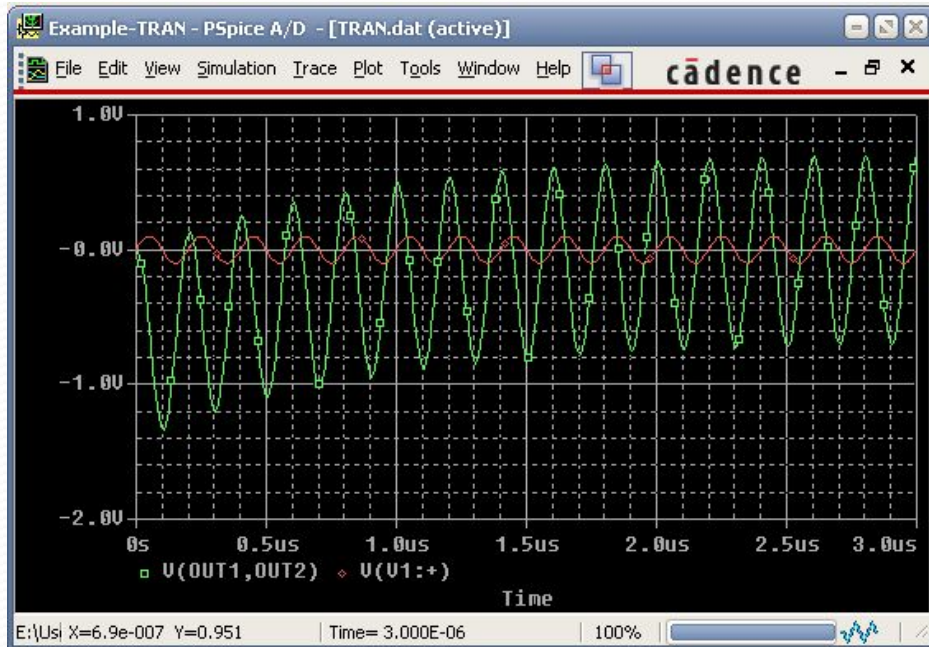


# Виды моделирования

Статистическое моделирование методом Монте-Карло  
(дифференциальный усилитель)

во временной области

Результаты расчетов  
метода Монте-Карло

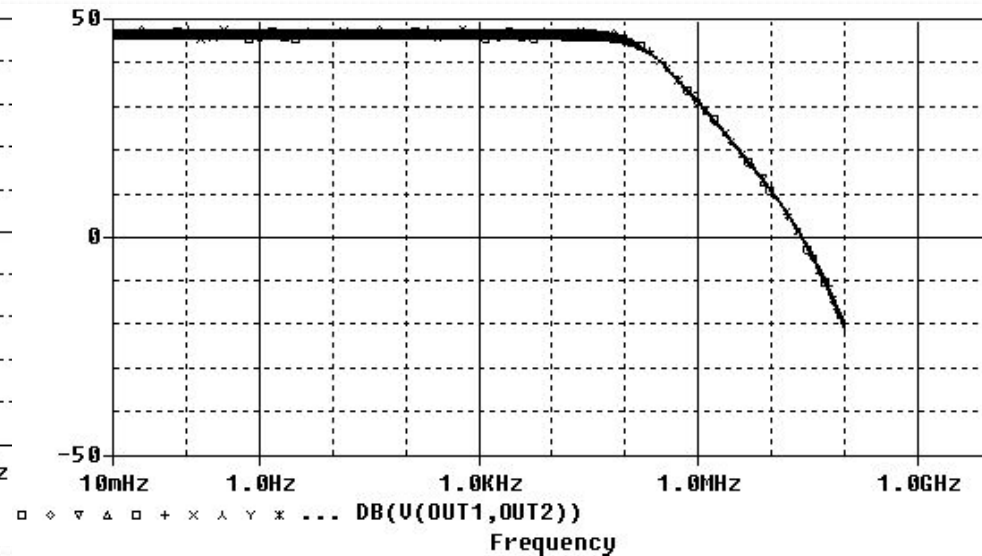
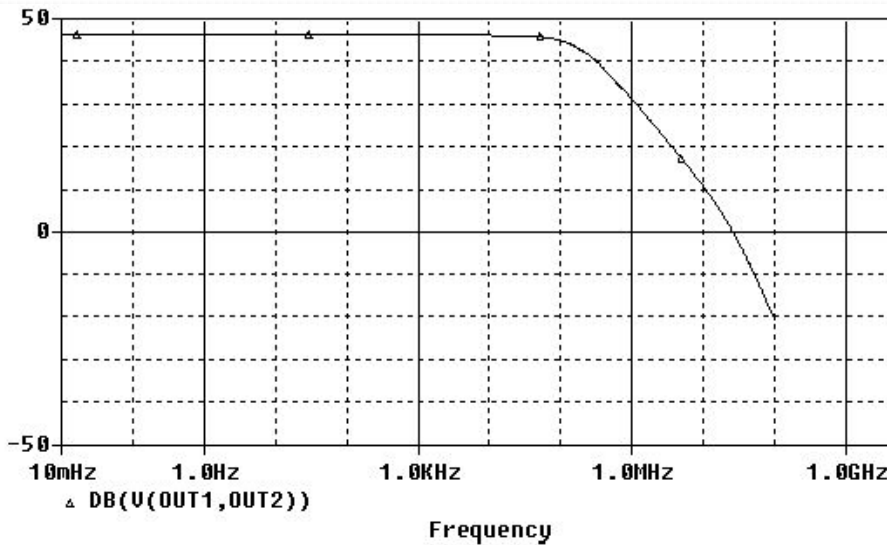


# Виды моделирования

Статистическое моделирование методом Монте-Карло  
(дифференциальный усилитель)

Результаты расчетов  
метода Монте-Карло

В частотной области

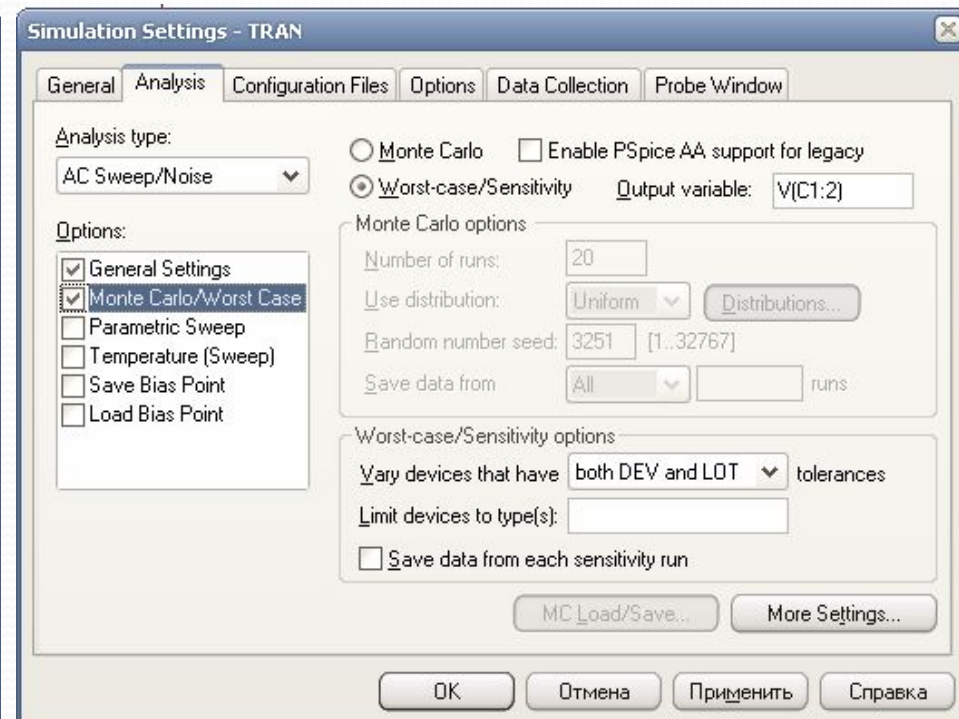
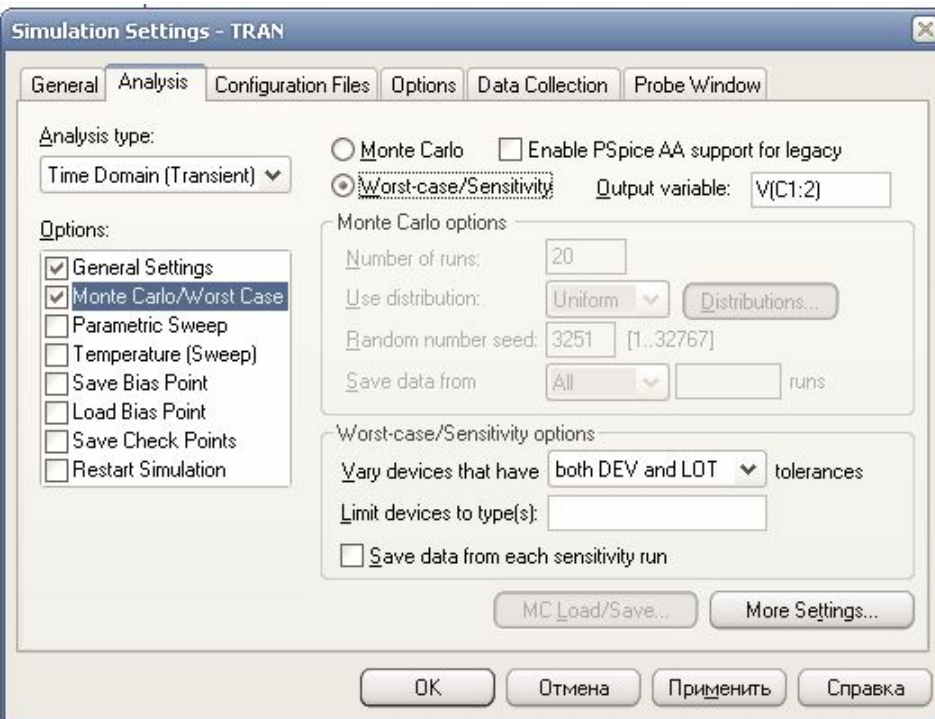


# Виды моделирования

Статистическое моделирование методом. Расчет на наихудший случай  
(дифференциальный усилитель)

Настройки расчетов  
во временной области

в частотной области

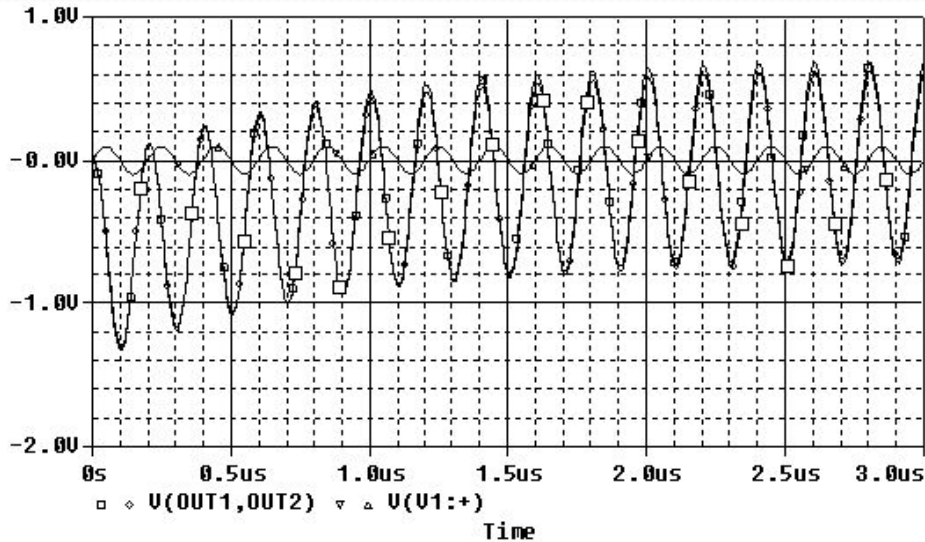


# Виды моделирования

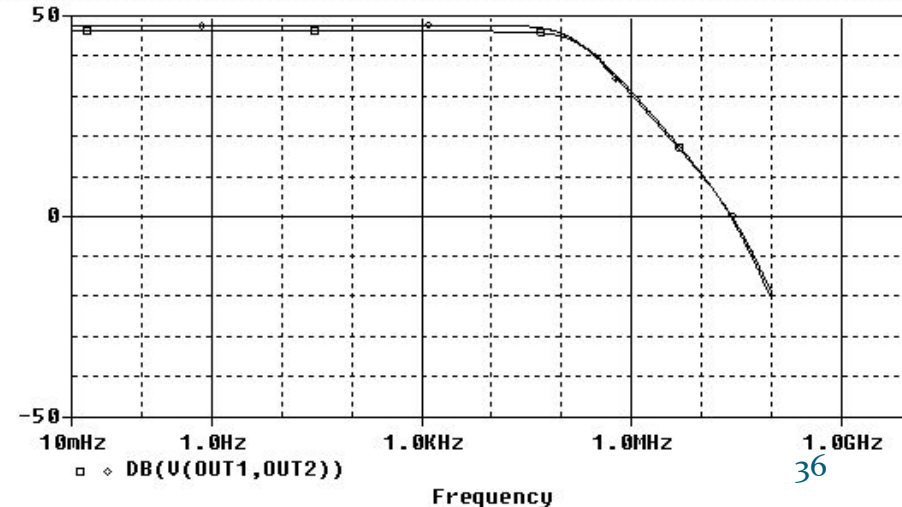
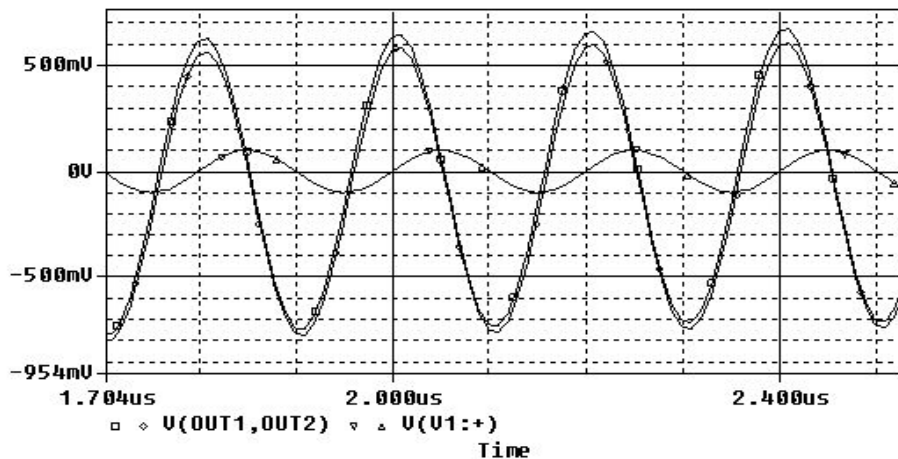
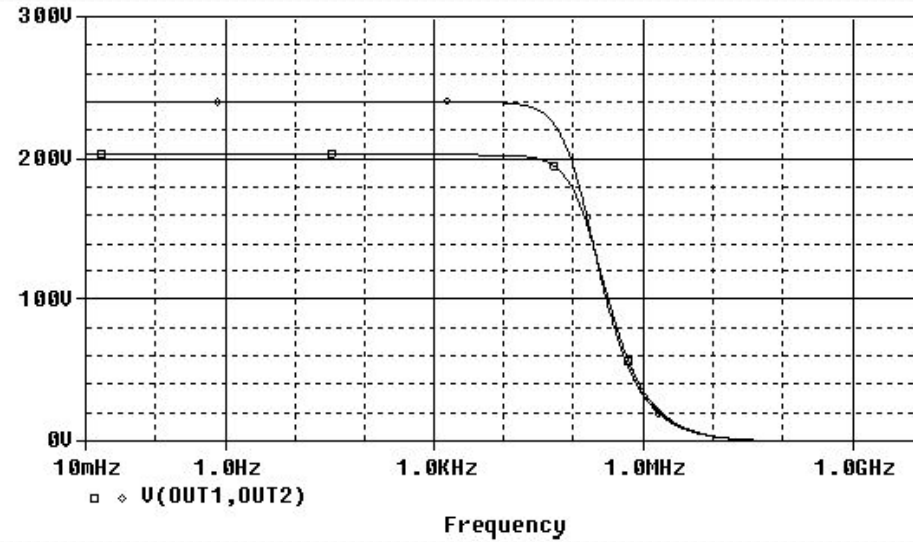
Статистическое моделирование методом. Расчет на наихудший случай  
(дифференциальный усилитель)

Результаты расчетов

во временной области

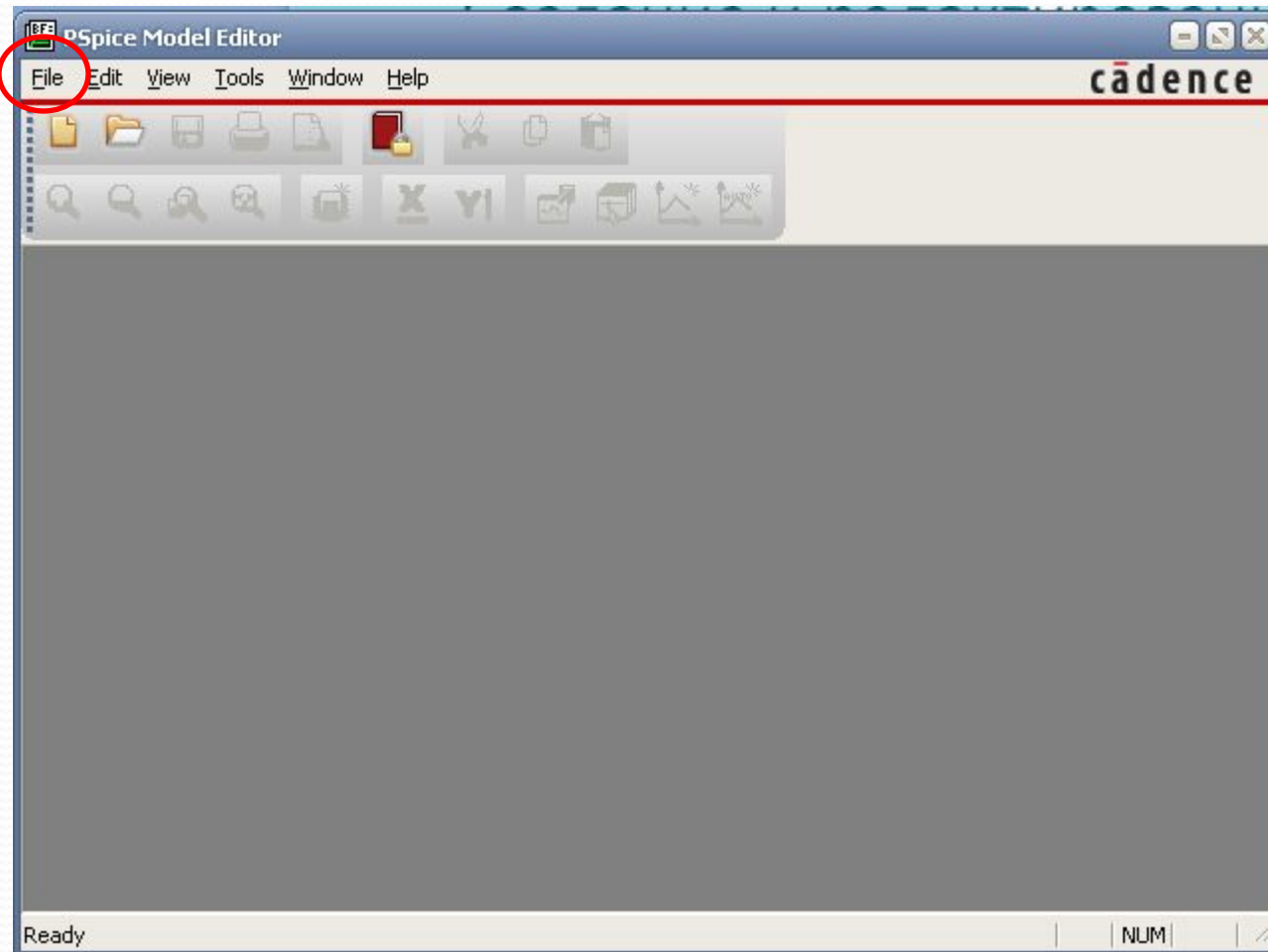


в частотной области (АЧХ, ЛАЧХ)



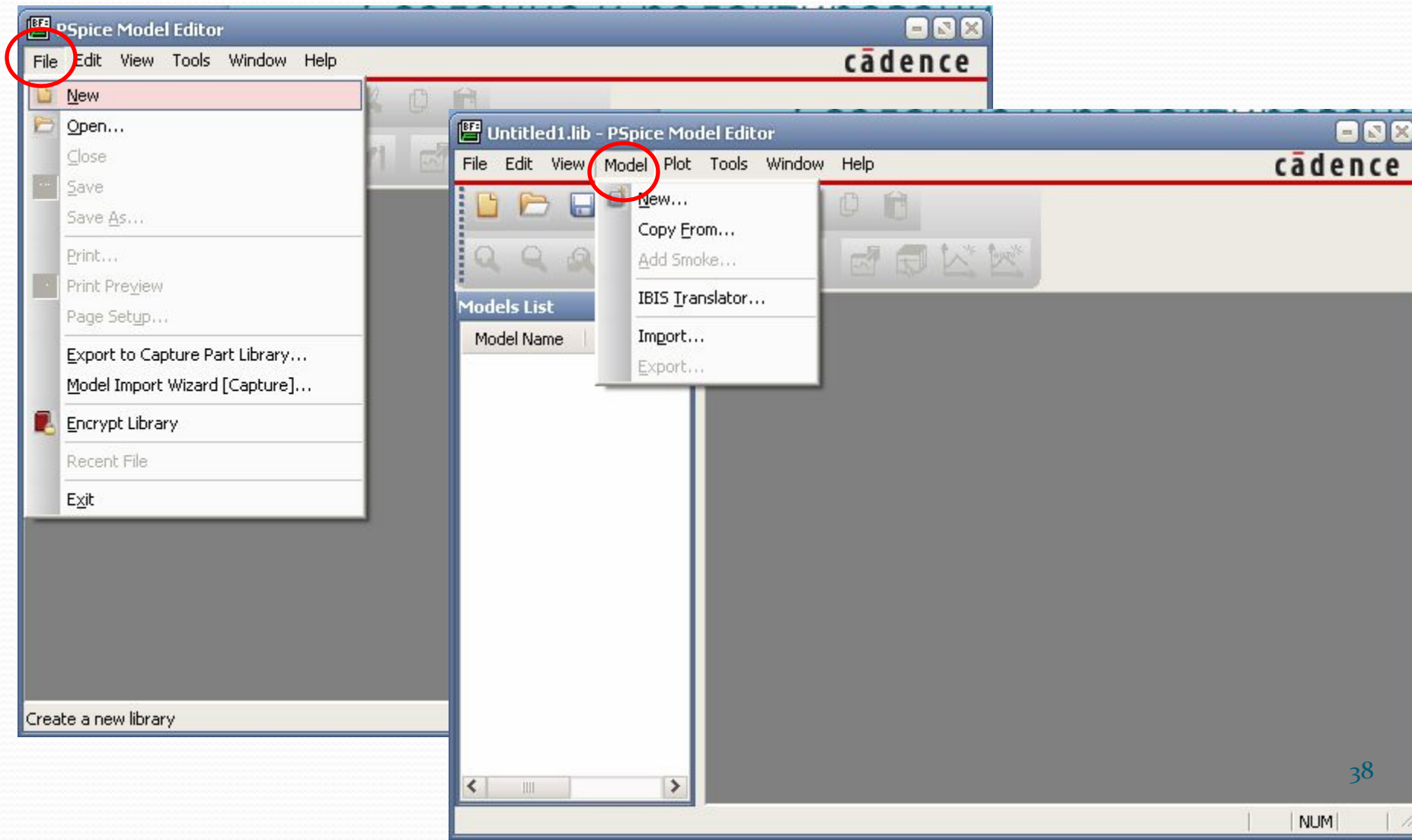
# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

Главное окно приложения

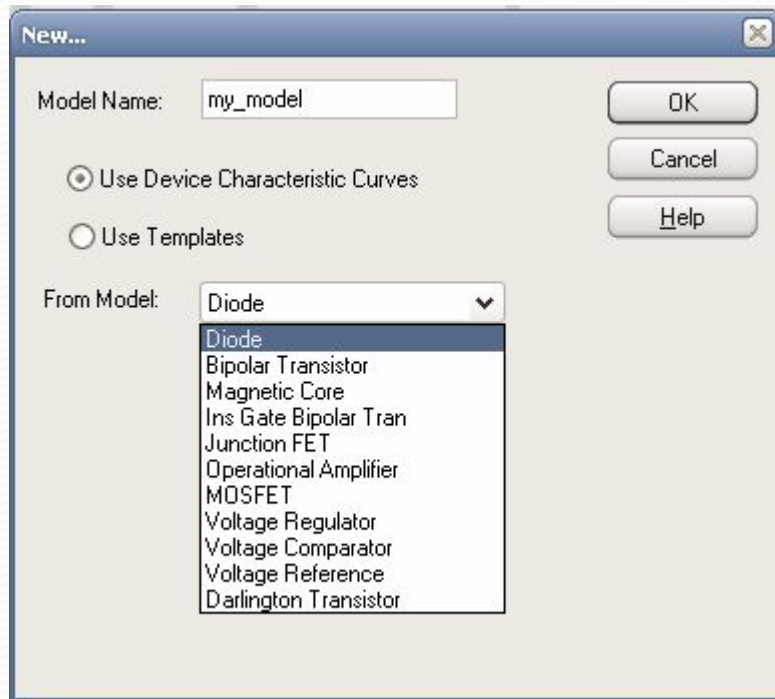


# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

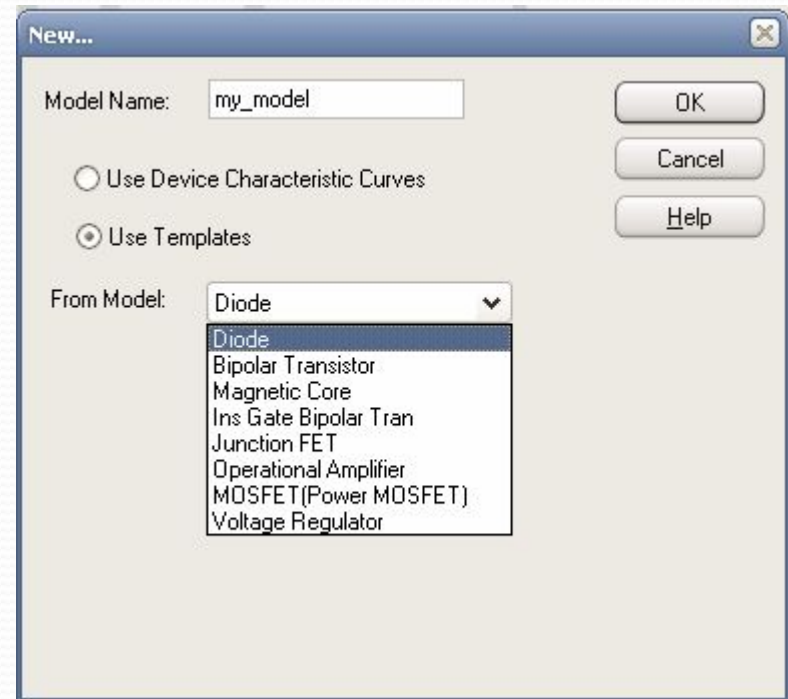
Главное окно приложения



# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor



1



2

# Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor

1

The screenshot displays the PSpice Model Editor interface for a diode model named 'my\_model\*'. The main configuration window is titled 'Forward Current' and contains the following data table:

#	Vfwd	Ifwd
1		
2		
3		
4		
5		
6		
7		
8		

To the right of the table is a graph showing the forward current (Ifwd) at 27°C as a function of forward voltage. The x-axis is labeled 'Forward Voltage' and ranges from 0.4V to 1.2V. The y-axis is labeled 'Ifwd (27°C)' and ranges from 0A to 300A. The graph shows a curve that starts near zero and rises sharply after approximately 0.8V.

At the bottom of the editor, the 'Parameters' table is visible:

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"/>



# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

1

The screenshot displays the PSpice Model Editor window for a diode model named 'my\_model\*'. The interface includes a menu bar (File, Edit, View, Model, Plot, Tools, Window, Help), a toolbar, and a 'Models List' table. The main workspace is titled 'Reverse Recovery' and contains four input fields: Trr (0), Ifwd (0.01), Irev (0.01), and Rf (100). To the right is a plot of current (I) versus time (Time). The plot shows a current pulse from 20mA to -20mA, with a reverse recovery transient. The y-axis ranges from -20mA to 20mA, and the x-axis ranges from -5ns to 20ns. A legend indicates the plot is for 'I fwd'. Below the plot are buttons for 'Forward Cur...', 'Junction Ca...', 'Reverse Le...', 'Reverse Br...', and 'Reverse Re...'. At the bottom is a 'Parameters' table.

Model Name	Type	Modified Date/Time
my_model*	Diode	11/26/10 at 10:36

Parameter Name	Value	Minimum	Maximum	Default	Active	Fixed
IS	1e-014	1e-020	0.1	1e-014	<input type="checkbox"/>	<input type="checkbox"/>
N	1	0.2	5	1	<input type="checkbox"/>	<input type="checkbox"/>
RS	0.001	1e-006	100	0.001	<input type="checkbox"/>	<input type="checkbox"/>
IKF	0	0	1000	0	<input 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"/>

# Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor

1

The screenshot shows the PSpice Model Editor interface for a diode model named 'my\_model\*'. The 'Reverse Breakdown' section is active, displaying a graph of Reverse Current (I<sub>r</sub>) versus Reverse Voltage (V<sub>r</sub>) at 27°C. The graph shows a curve that starts at 0A and 0V, rises sharply to about 0.1A at 100.10V, and then continues to rise more gradually, reaching approximately 1.00A at 100.30V. The x-axis is labeled 'Reverse Current' and ranges from 0A to 1.00A. The y-axis is labeled 'Ur (27°C)' and ranges from 100.00 to 100.30V.

The 'Parameters' table at the bottom of the window lists the following parameters:

Parameter Name	Value	Minimum	Maximum	Default	Active	Fixed
IS	1e-014	1e-020	0.1	1e-014	<input type="checkbox"/>	<input type="checkbox"/>
N	1	0.2	5	1	<input type="checkbox"/>	<input type="checkbox"/>
RS	0.001	1e-006	100	0.001	<input type="checkbox"/>	<input type="checkbox"/>
IKF	0	0	1000	0	<input 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"/>

# Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor

1

Options

Part Creation Setup

- Always Create Part When Saving Model
- Pick symbols manually

Schematic Editor

- Design Entry HDL
- Capture
- Schematics

Save Part To

- Path Same As Model Library
- User-Defined Part Library

(none) Browse...

Base Parts On

Parts In An Existing Part Library

C:\OrCAD\OrCAD\_16.2\tools\Capture\Li Browse...

Misc Settings

Current Library Path: C:\OrCAD\OrCAD\_16.2\tools\PSpice\UserLib

Backup Directory: C:\OrCAD\OrCAD\_16.2\tools\pspice\Backup

- Synchronize Graph Splitter Windows
- Automatically Update Graph
- Use environment variable in pll view

OK Cancel Help

Untitled1.lib:my\_model - PSpice Model Editor - [Reverse Breakdown]

File Edit View Model Plot Tools Window Help

Extract Parameters  
Customize...  
Options...

Models List

Model Name	Type	Created/Last Modified
my_model*	Diode	11/26/10 at 10:36

Parameters

Parameter Name	Value	Minimum	Maximum	Default
IS	1e-014	1e-020	0.1	1e-014
N	1	0.2	5	1
RS	0.001	1e-006	100	0.001
IKF	0	0	1000	0
XTI	3	-100	100	3
EG	1.11	0.1	5.51	1.11
CJO	1e-012	1e-020	0.001	1e-012
M	0.3333	0.1	10	0.3333
VJ	0.75	0.3905	10	0.75

Set up symbol creation options

ence

.75A 1.00A

NUM

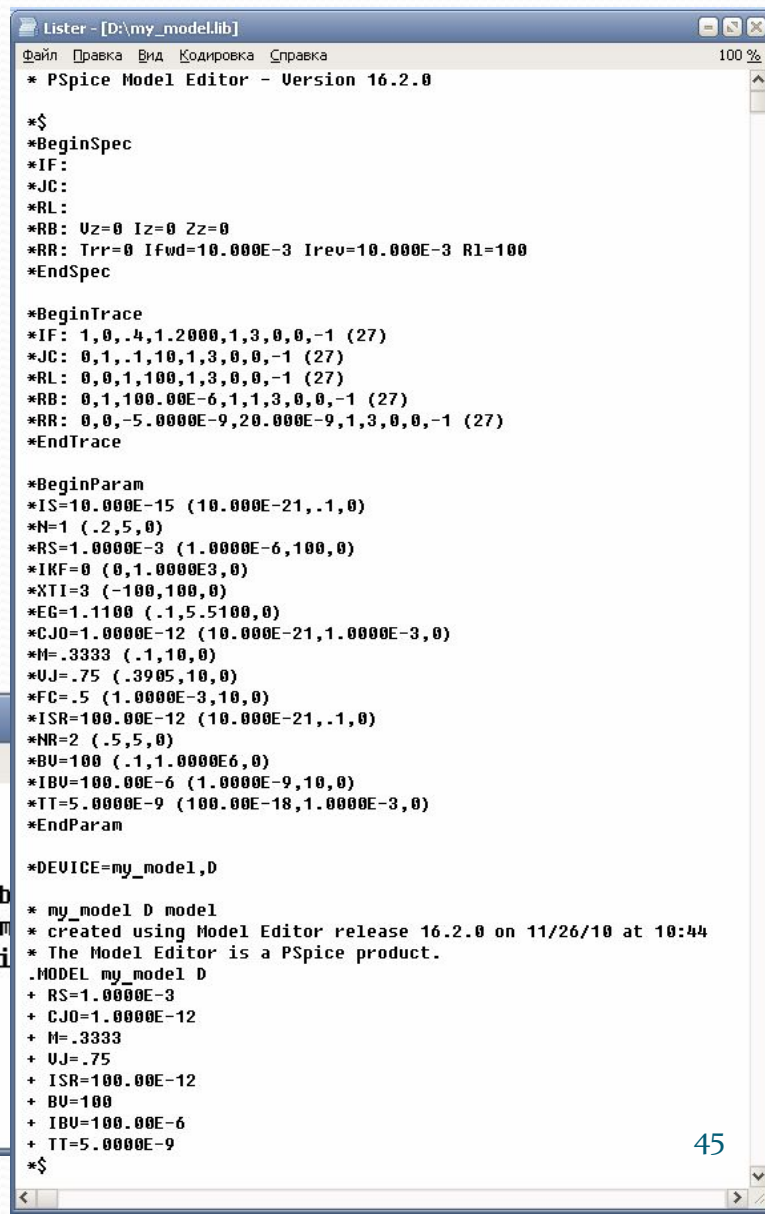
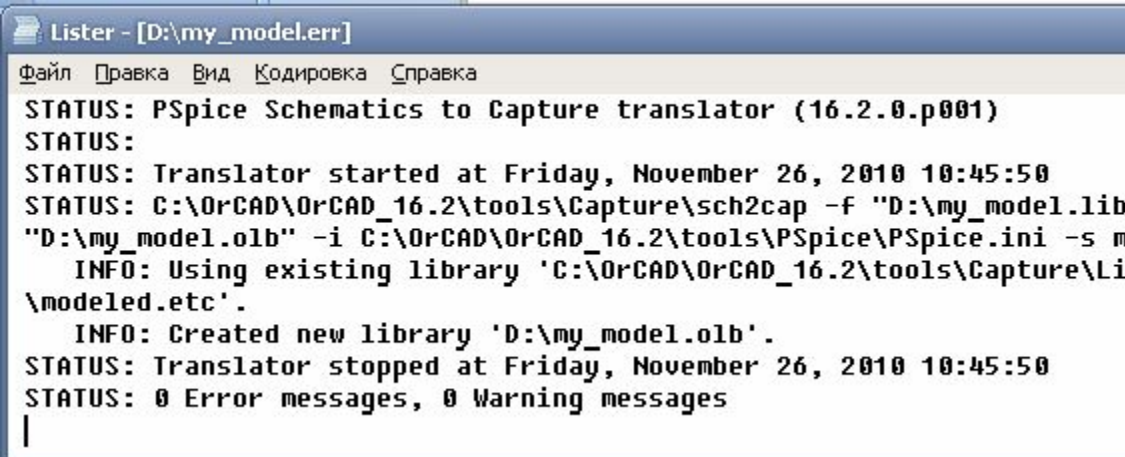
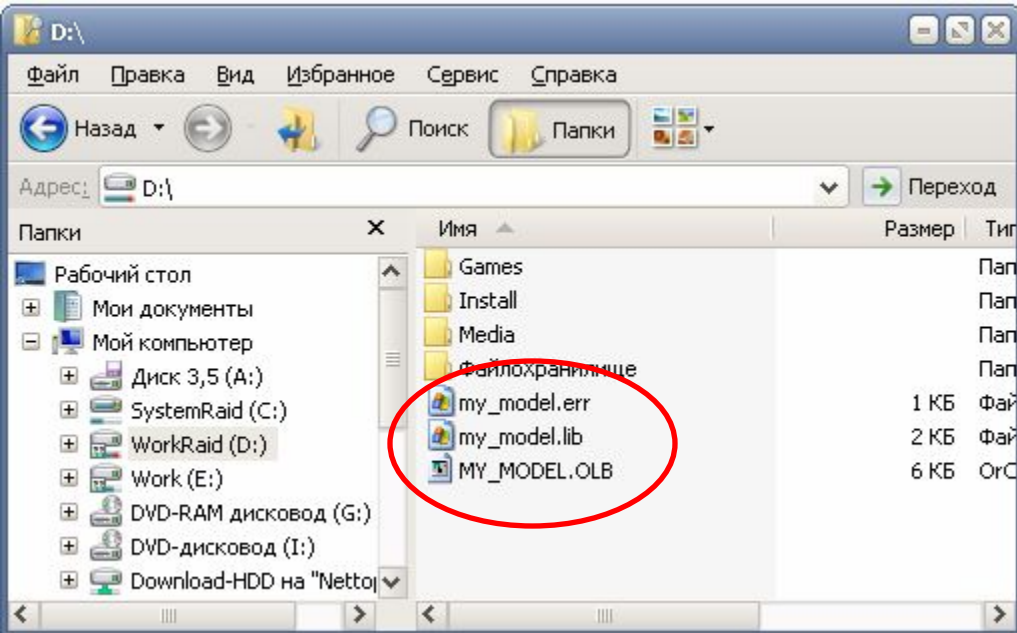
# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

The screenshot displays the PSpice Model Editor interface. The main window shows a 'Reverse Breakdown' plot with a green curve on a black grid. The y-axis ranges from 100.20 to 100.30. A 'Сохранить как' (Save As) dialog box is open, showing the file 'my\_model.lib' selected in the 'WorkRaid (D:)' drive. The dialog also shows a list of folders and files, including 'Имя файла: my\_model.lib' and 'Тип файла: Model Library Files (\*.lib)'. A 'Parameters' table is visible at the bottom left of the editor window.

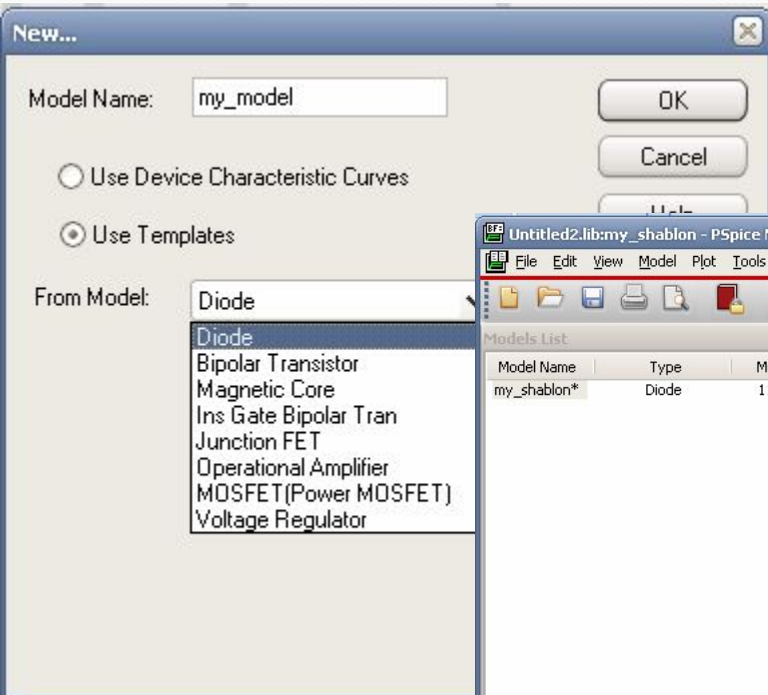
Parameter Name	Value	Minimum	Maximum	Default
IS	1e-014	1e-020	0.1	1e-014
N	1	0.2	5	1
RS	0.001	1e-006	100	0.001
IKF	0	0	1000	0
XTI	3	-100	100	3
EG	1.11	0.1	5.51	1.11
CJO	1e-012	1e-020	0.001	1e-012
M	0.3333	0.1	10	0.3333
VJ	0.75	0.3905	10	0.75

Save the active library with a new name

# 1 Создание и редактирование моделей компонентов ЭС в PSpice Model Editor



# Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor



Simulation Parameters

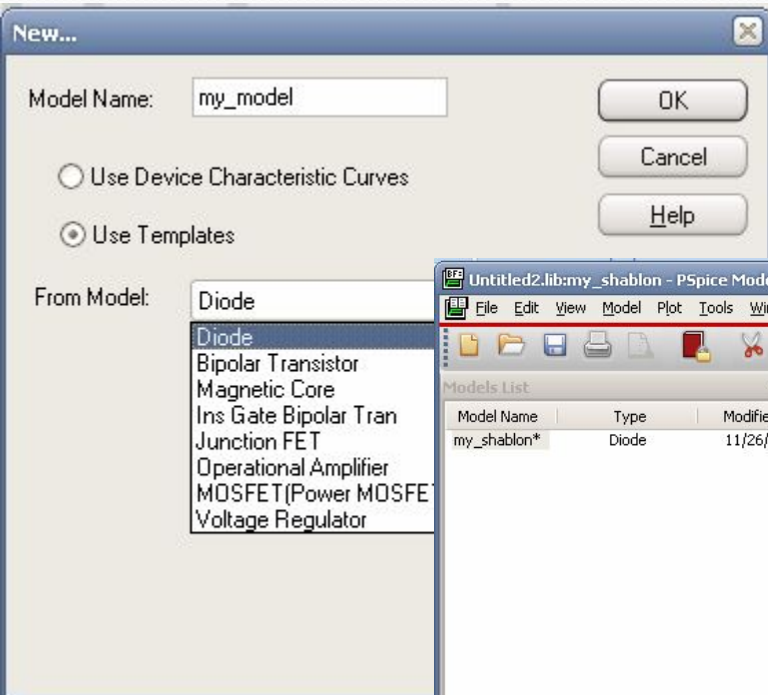
Property Name	Description	Value	Default	Unit	Distribution	Postol	Illegal	Editable
IS	Saturation current	10f	10f	A				<input type="checkbox"/>
RS	Ohmic resistance	0	0	Ohm				<input type="checkbox"/>
N	Emission coefficient	1	1					<input type="checkbox"/>
TT	Transit time	0	0	sec				<input type="checkbox"/>
CJO	Junction capacitance	0	0	F				<input type="checkbox"/>
VJ	Junction potential	1	1	V				<input type="checkbox"/>
M	Grading coefficient	0.5	0.5					<input type="checkbox"/>
EG	Activation energy	1.11	1.11	eV				<input type="checkbox"/>
XTI	Isat temperature exp	3	3					<input type="checkbox"/>
KF	Flicker noise coef.	0	0					<input type="checkbox"/>
AF	Flicker noise exp.	1	1					<input type="checkbox"/>
FC	Depletion cap. coef.	0.5	0.5					<input type="checkbox"/>
BV	Rev breakdown volt	100	100	V				<input type="checkbox"/>
IBV	I at V-breakdown	.001	.001	A				<input type="checkbox"/>

Model Text (Read Only)

```
* created using Model Editor release 16.2.0 on 11/26/10 at 10:57
*DEVICE=my_shablon,D
.subckt my_shablon AN CAT
+ params:
+ AREA=1.0
D_my_shablon AN CAT model122 {area}
.model model122 d
.ends my_shablon
```

2

# Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor



Models List

Model Name	Type	Modified (Date)
my_shablon*	Diode	11/26/10

Test Node Mapping

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

Node	Port
TERM_AN	AN
NODE_AN	AN
NODE_CAT	CAT

Smoke Parameters

These are Device Maximum Operating condition parameters required for Smoke Analysis

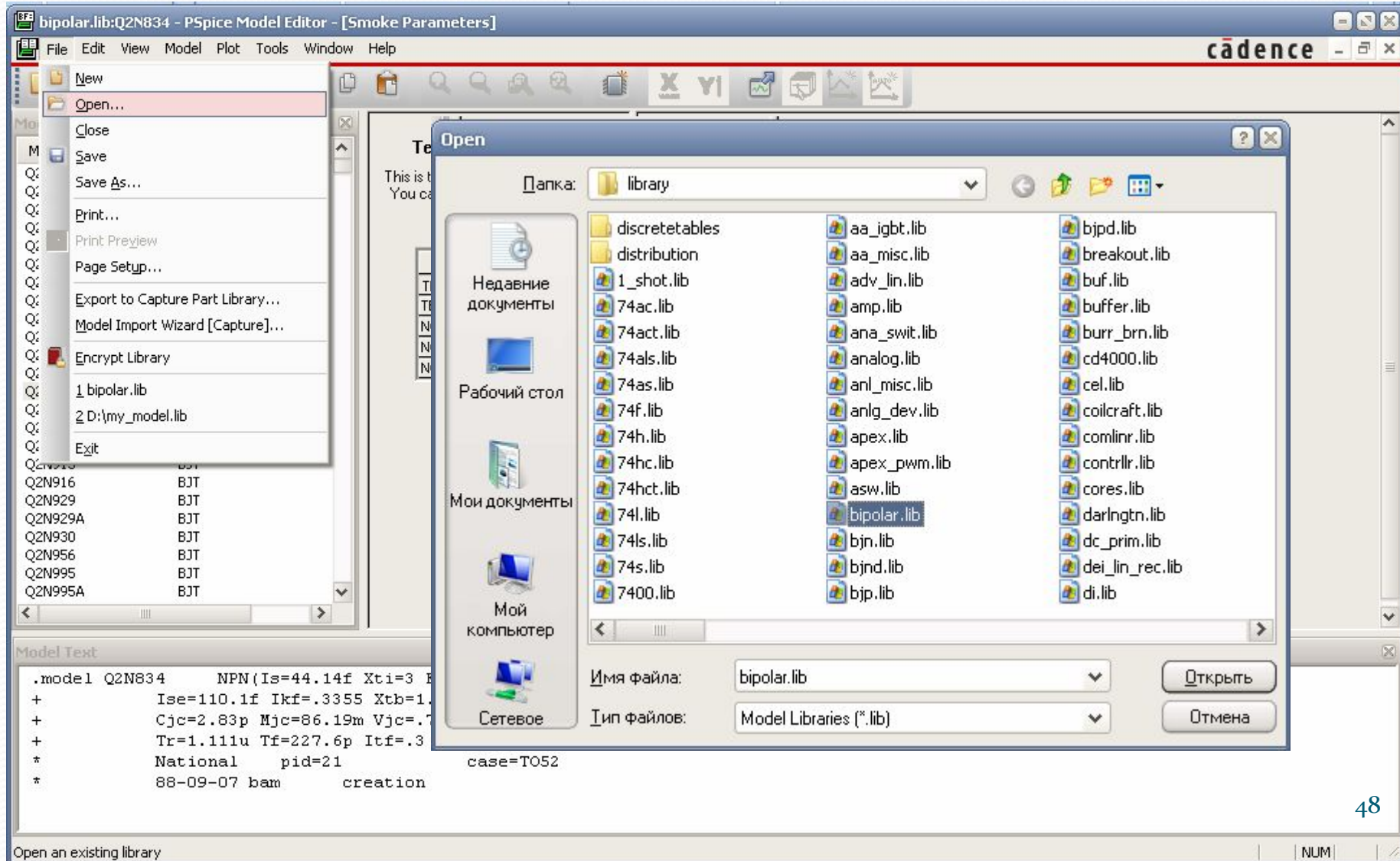
Device Max Ops	Description	Value	Unit
IF	Max forward current		A
VR	Peak reverse voltage		V
PDM	Max pwr dissipation		W
TJ	Max junction temp		C
RJC	J-C thermal resist		CAW
RCA	C-A thermal resist		CAW

Model Text (Read Only)

```
* created using Model Editor release 16.2.0 on 11/26/10 at 10:57
*DEVICE=my_shablon,D
.subckt my_shablon AN CAT
+ params:
+ AREA=1.0
D_my_shablon AN CAT model122 (area)
.model model122 d
.ends my_shablon
```

2

# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor





# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

The screenshot displays the PSpice Model Editor window for a BJT model. The interface includes a menu bar (File, Edit, View, Model, Plot, Tools, Window, Help), a toolbar, and a main workspace divided into three sections:

- Models List:** A table listing various BJT models. The model **Q2N2222** is selected.
- Test Node Mapping:** A section for defining node and port mappings. It contains a table with the following data:

Node	Port
TERM_IC	C
TERM_IB	B
NODE_VC	C
NODE_VB	B
NODE_VE	E
- Smoke Parameters:** A section for defining maximum operating condition parameters. It contains a table with the following data:

Device Max Ops	Description	Value	Unit
IB	Max base current		A
IC	Max collector current	800m	A
VCB	Max C-B voltage	60	V
VCE	Max C-E voltage	30	V
VEB	Max E-B voltage	5	V
PDM	Max pwr dissipation	1.2	W
TJ	Max junction temp	200	C
RJC	J-C thermal resist	146	CAW
RCA	C-A thermal resist	292	CAW
SBSLP	Second brkdown slope		
SBINT	Sec brkdown intercept		A
SBSLP	SB temp derate slope		%/C
SBMIN	SB temp derate at TJ		%

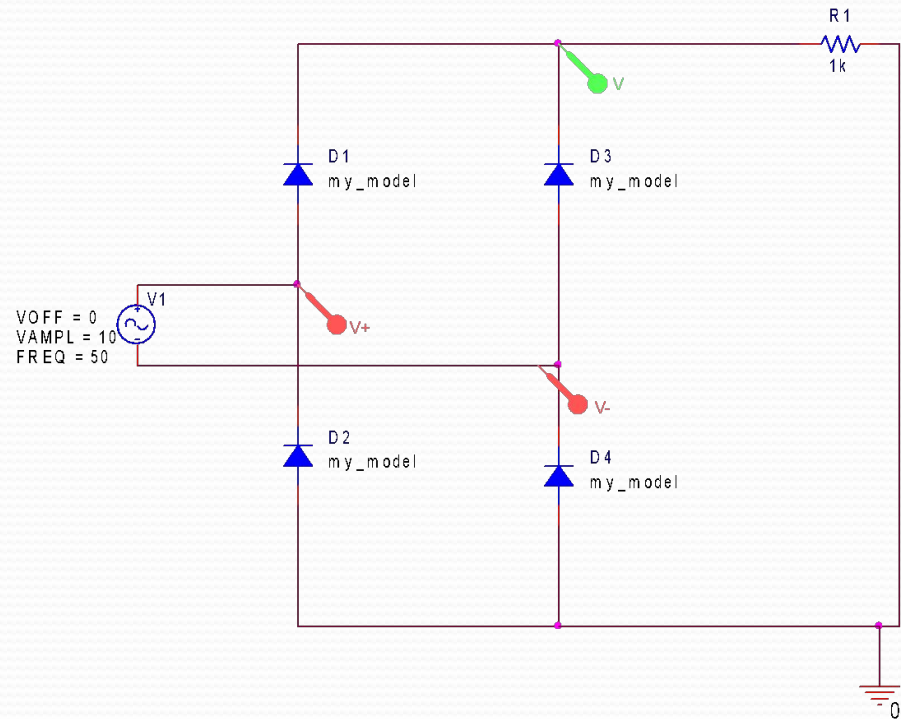
At the bottom, the **Model Text** window shows the following code:

```
.model Q2N2222 NPN(Is=14.34f Xti=3 Eg=1.11 Vaf=74.03 Bf=255.9 Ne=1.307
+ Ise=14.34f Ikf=.2847 Xtb=1.5 Br=6.092 Nc=2 Isc=0 Ikr=0 Rc=1
+ Cjc=7.306p Mjc=.3416 Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 Vje=.75
+ Tr=46.91n Tf=411.1p Itf=.6 Vtf=1.7 Xtf=3 Rb=10)
* National pid=19 case=TO18
* 88-09-07 bam creation
```

# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

## Проверка работоспособности созданной модели

1. Открываем библиотеку \*.olb в OrCad (библиотека символа)
2. Подключаем библиотеку \*.lib (библиотека математической модели)
3. Собираем схему и моделируем.



# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

## Проверка работоспособности созданной модели

The screenshot displays the OrCAD Capture interface for a PSpice simulation. The main workspace shows a circuit diagram with a voltage source V1 (VOFF = 0, VAMPL = 10, FREQ = 50) connected to a bridge-like structure of four diodes (D1, D2, D3, D4) labeled 'my\_model'. A resistor R1 (1k) is connected to the output of the bridge. The status bar at the bottom indicates that the PSpice netlist generation is complete.

Property	Value
Title	<Title>
Size	Document Number
A	<Doc>
Date:	Friday, November 26, 2010
Sheet	1 of 1
Rev	<Rev Code>

PSpice netlist generation complete  
Creating PSpice Netlist  
Writing PSpice Flat Netlist D:\temp\my\_model\PSpiceFiles\SCHEMATIC1\SCHEMATIC1.net  
PSpice netlist generation complete

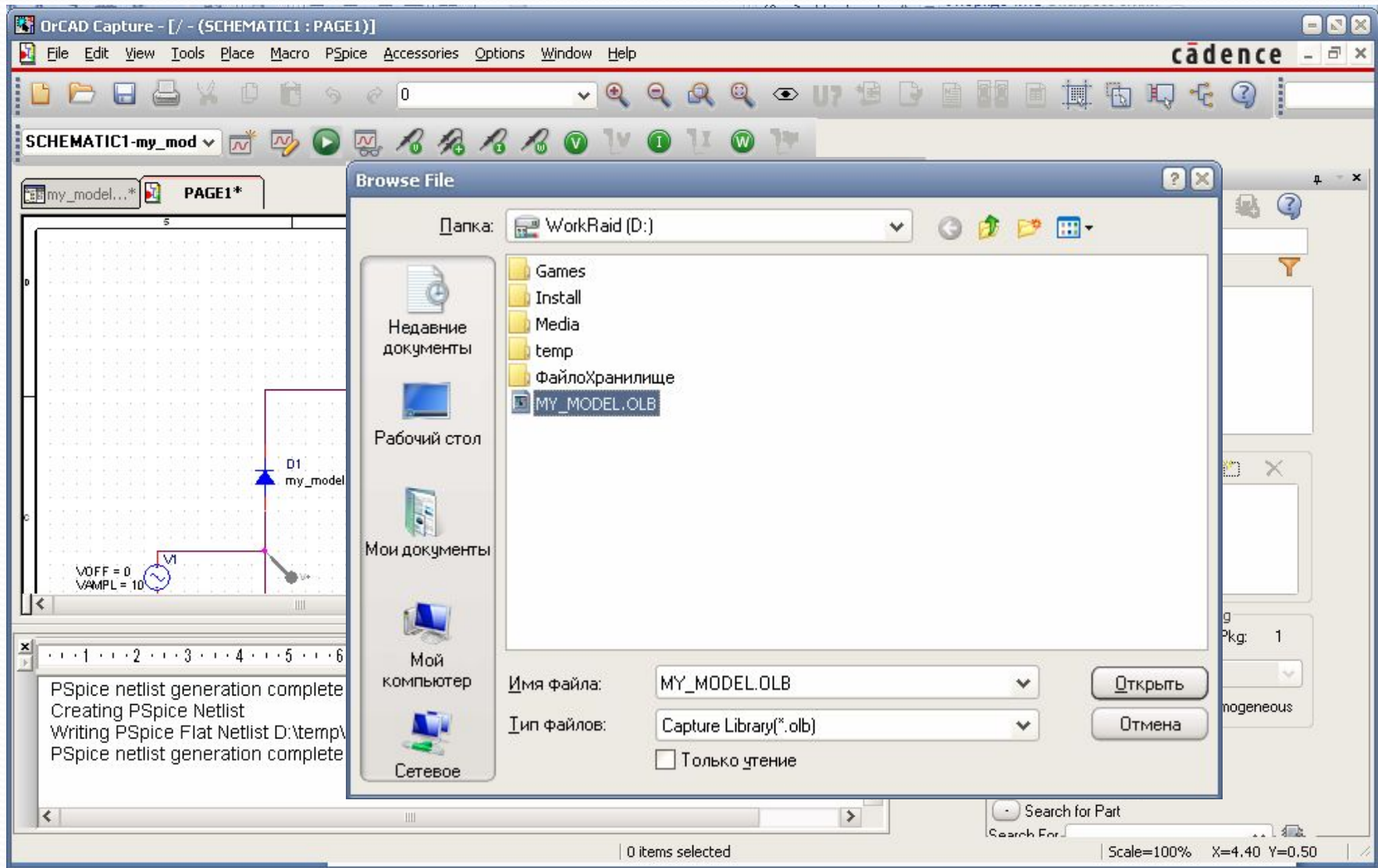
# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

## Проверка работоспособности созданной модели

The screenshot displays the OrCAD Capture interface. The main workspace shows a circuit diagram with a voltage source V1 (V<sub>OFF</sub> = 0, V<sub>AMPL</sub> = 10), two diodes D1 and D3 (both labeled 'my\_model'), and a resistor R1 (1k). The 'Place Part' dialog box is open on the right, showing a search for a part. The 'Part List' field contains the text 'Добавить библиотеку' (Add library). The 'Libraries' list includes 'ANALOG', 'Design Cache', 'DIG\_PRIM', and 'SOURCE'. A red circle highlights a small icon in the 'Libraries' list. The 'Packaging' section shows 'Parts per Pkg: 1' and 'Type: Homogeneous'. The 'Search for Part' section is visible at the bottom right. The status bar at the bottom indicates '0 items selected' and 'Scale=100% X=4.40 Y=0.50'. A console window at the bottom left shows the following text:

```
PSpice netlist generation complete
Creating PSpice Netlist
Writing PSpice Flat Netlist D:\temp\my_model-PSpiceFiles\SCHEMATIC1\SCHEMATIC1.net
PSpice netlist generation complete
```

# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor



# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

The screenshot displays the OrCAD Capture interface for a PSpice simulation. The main workspace shows a circuit schematic on a grid. The circuit includes a voltage source labeled  $V1$  with parameters  $V_{OFF} = 0$  and  $V_{AMPL} = 10$ . Two diodes,  $D1$  and  $D3$ , are placed, both labeled with the model name `my_model`. A resistor  $R1$  with a value of  $1k$  is also present. The schematic is titled `SCHEMATIC1-my_mod` and `PAGE1*`.

The `Place Part` dialog box is open on the right side of the screen. It shows the `Part` field set to `my_model`. The `Part List` field also contains `my_model`. The `Libraries` section lists several libraries, with `MY_MODEL` selected. The `Packaging` section shows `Parts per Pkg:` set to `1`. The `Part:` dropdown is empty, and the `Type:` is set to `Homogeneous`. A preview of the diode symbol is shown with the label `my_model` below it. The `Normal` radio button is selected, and the `Convert` radio button is unselected. The `Search for Part` field is empty.

The status bar at the bottom of the window indicates `0 items selected`. The bottom right corner shows `Scale=100%` and `X=0 Y=0`.

The command window at the bottom left displays the following text:

```
PSpice netlist generation complete
Creating PSpice Netlist
Writing PSpice Flat Netlist D:\temp\my_model-PSpiceFiles\SCHEMATIC1\SCHEMATIC1.net
PSpice netlist generation complete
```

# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

The image shows a screenshot of the OrCAD Capture software interface. The main window displays a circuit diagram with four diode components labeled D1, D2, D3, and D4, all of type 'my\_model'. A voltage source V1 is connected to the circuit. The simulation parameters are: VOFF = 0, VAMPL = 10, and FREQ = 50. The text 'Жмем кнопку настроек параметров моделирования' (We click the simulation parameter settings button) is overlaid on the diagram. A red circle highlights the 'Simulation Settings' button in the toolbar. The 'Simulation Settings - my\_model' dialog box is open, showing the 'Configuration Files' tab. The 'Library' category is selected. The 'Browse...' button is circled in red. The 'Configured Files' list contains 'nom.lib'. The 'Library Path' is set to 'C:\OrCAD\OrCAD\_16.2\tools\PSpice\UserLib'. The status bar at the bottom shows 'Ready'.

DrCAD Capture - [ / - (SCHEMATIC1 : PAGE1) ]

File Edit View Tools Place Macro PSpice Accessories Options Window Help

cadence

SCHEMATIC1-my\_mod

my\_model...\* PAGE1\*

Жмем кнопку настроек параметров моделирования

Simulation Settings - my\_model

General Analysis Configuration Files Options Data Collection Probe Window

Category:

- Stimulus
- Library
- Include

Details

Filename:

Browse...

Configured Files

- nom.lib

Add as Global

Add to Design

Add to Profile

Edit

Change

Library Path

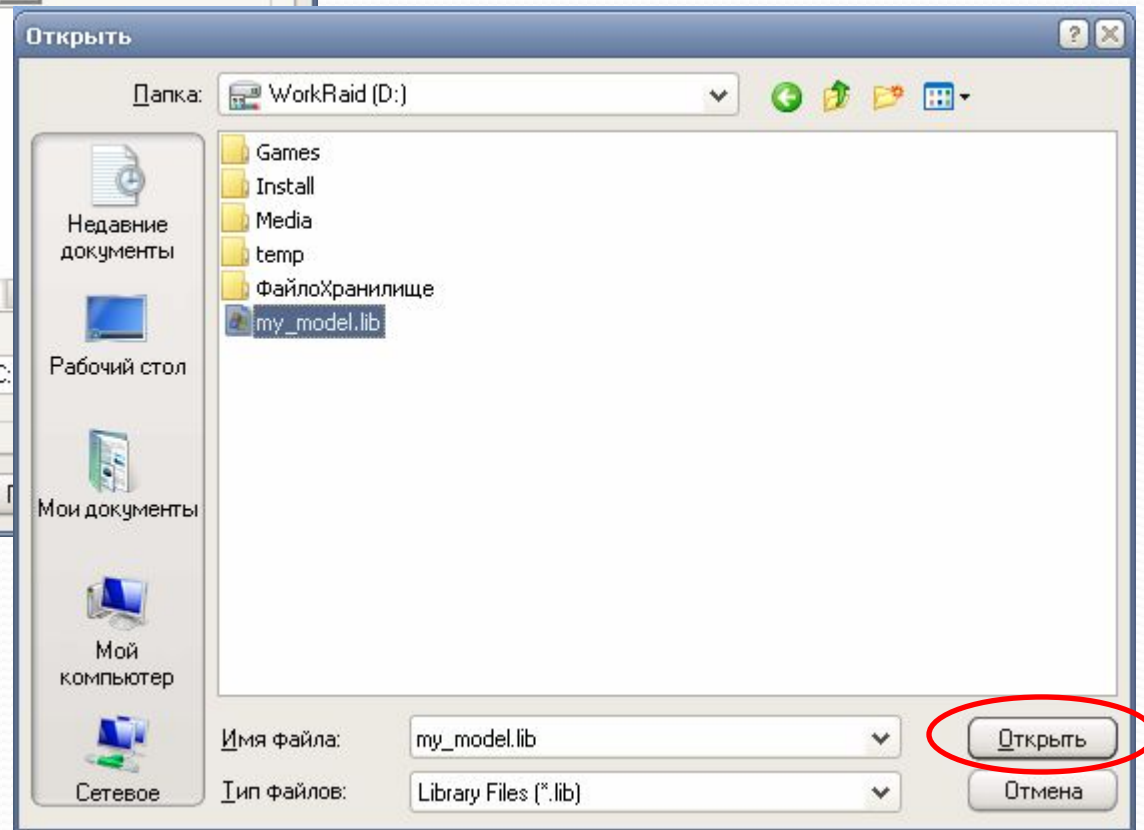
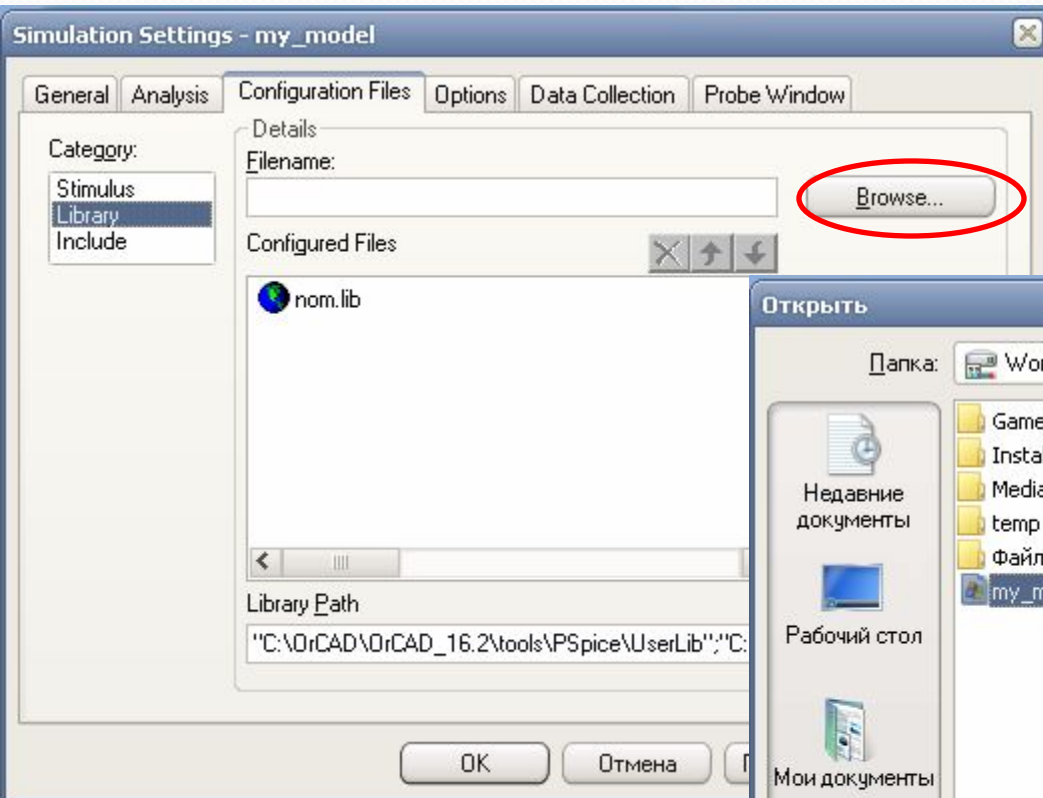
"C:\OrCAD\OrCAD\_16.2\tools\PSpice\UserLib"; "C:\

Browse...

OK Отмена Применить Справка

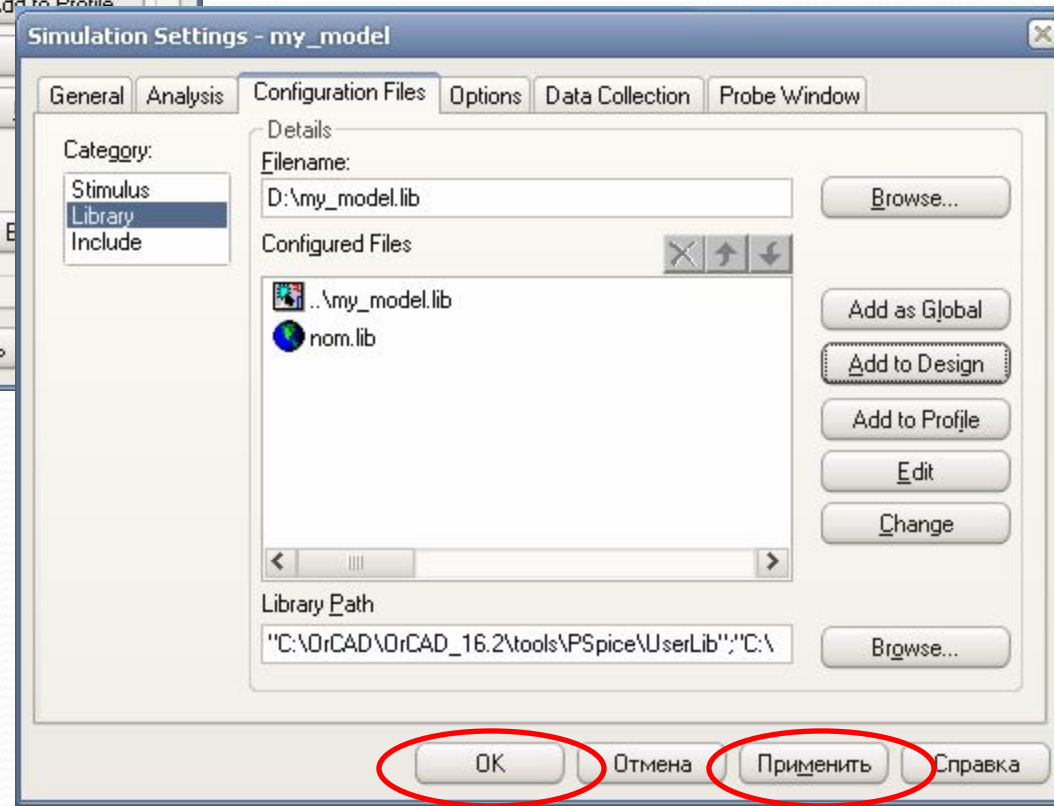
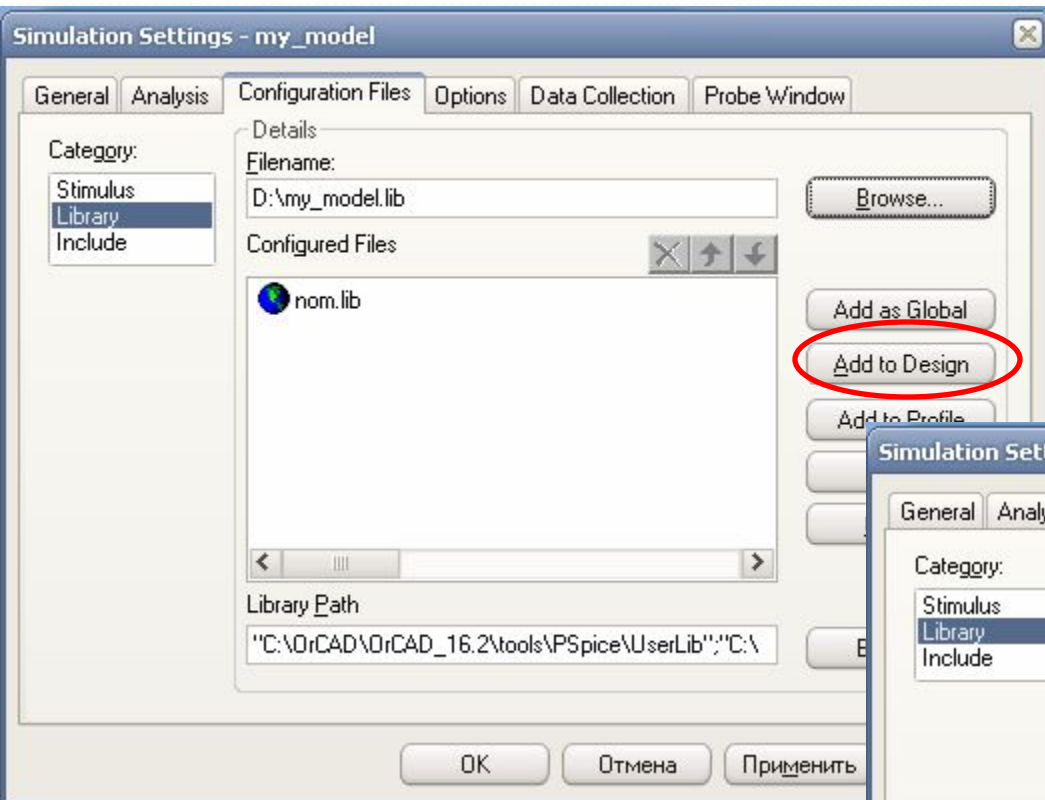
Ready

# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor



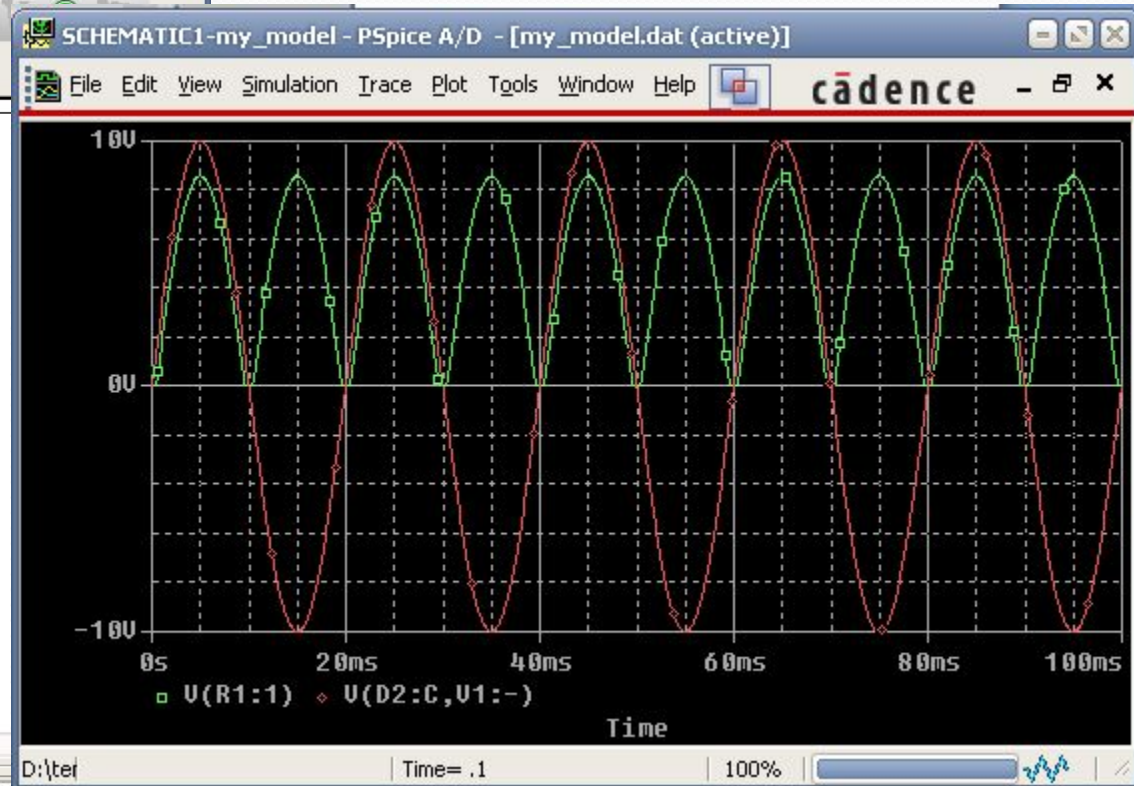
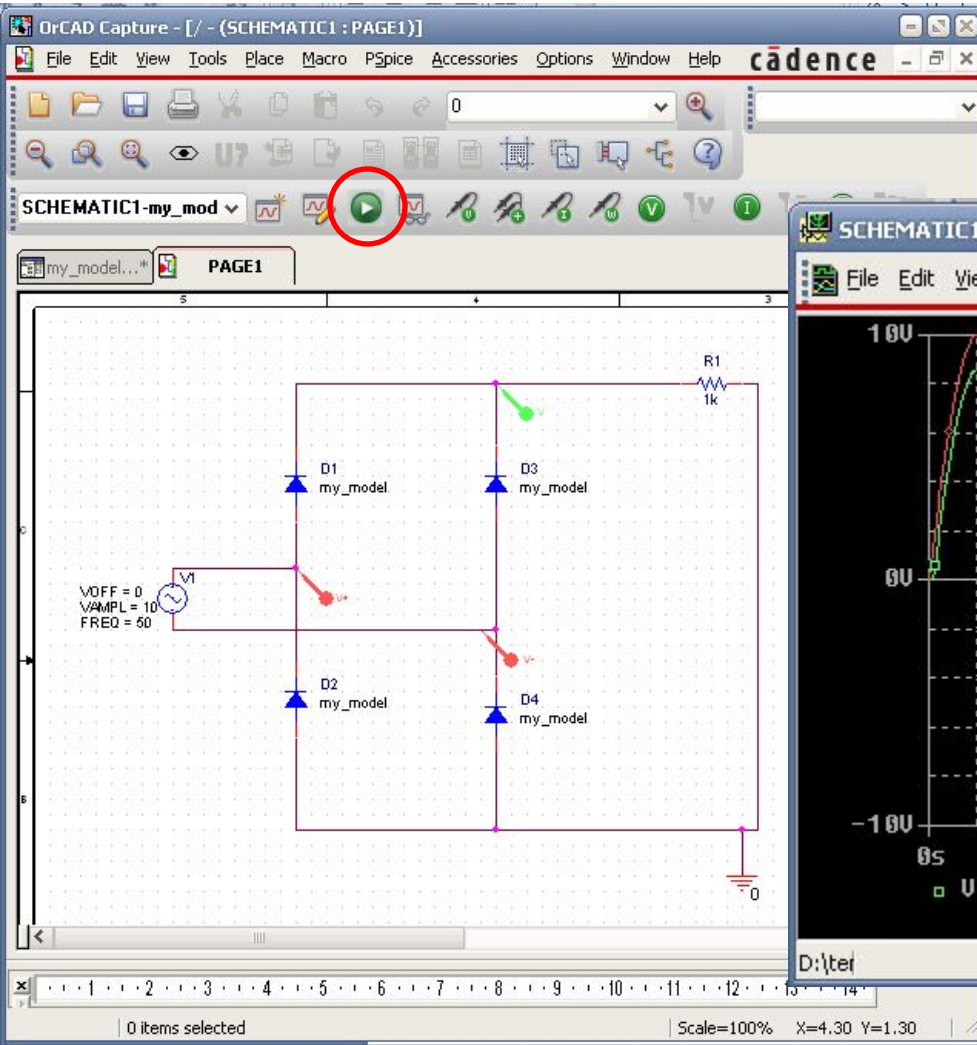



# Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor



# Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

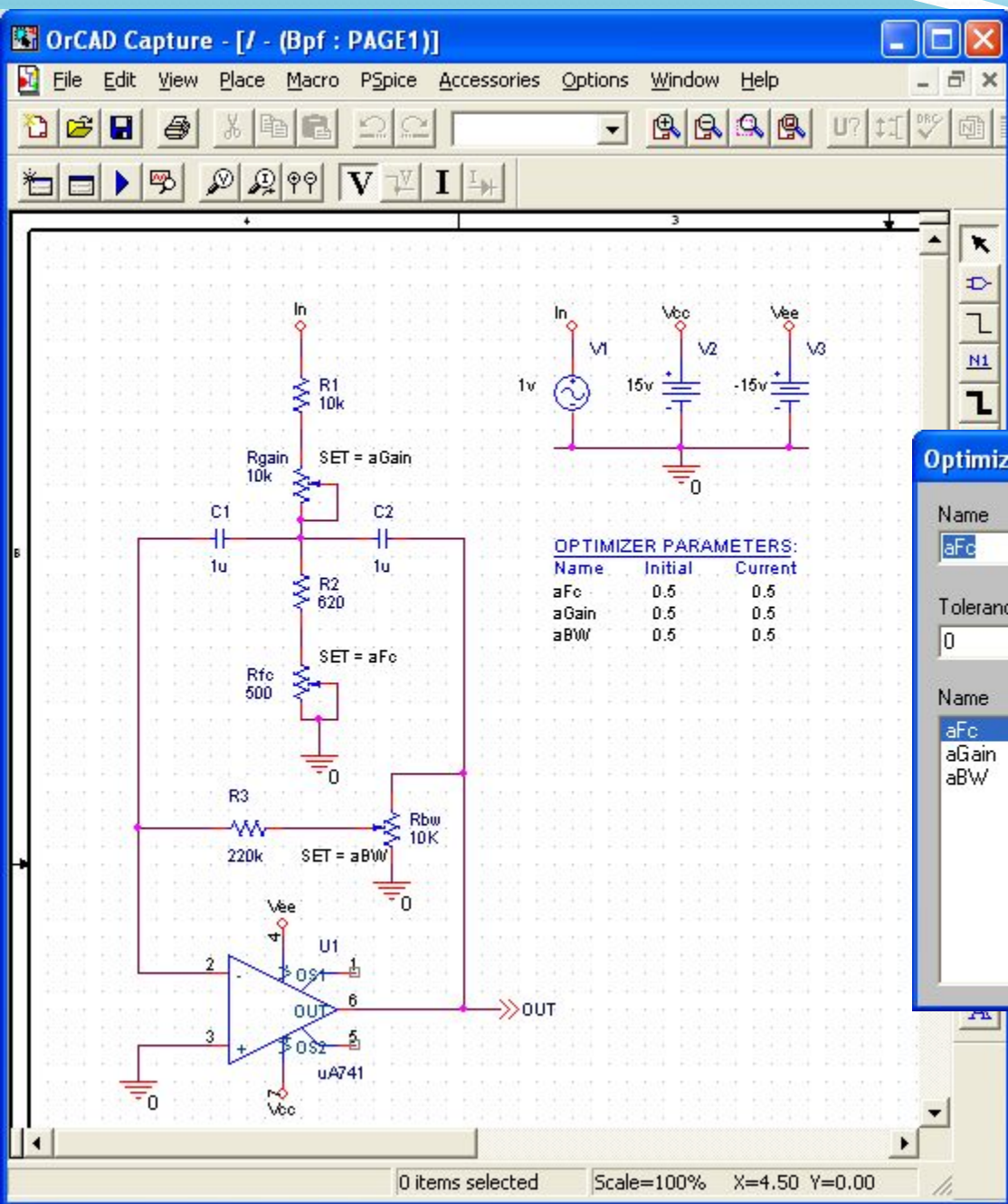
Проверка работоспособности созданной модели





# Программа параметрической оптимизации PSpice Optimizer

Критерий оптимизации – обеспечение заданного значения целевой функции при выполнении ряда линейных и нелинейных ограничений.



### Optimizer Parameters

Name	Initial Value	Current Value
aFc	0.5	0.5
aGain	0.5	0.5
aBW	0.5	0.5

Simulation With

Initial Value

Current Value

Tolerance Lower Limit Upper Limit

0 0.01 1

Add

Change

Delete

OK

Cancel

OrCAD Capture - [ / - (Bpf : PAGE1)]

File Edit View Place Macro PSpice Accessories Options Window Help

- New Simulation Profile
- Edit Simulation Profile
- Run
- View Simulation Results
- View Output File
- Create Netlist
- View Netlist
- Place Optimizer Parameters
- Run Optimizer
- Markers
- Bias Points

OPTIMIZER PARA

Name	Initial
aFc	0.5
aGain	0.5
aBW	0.5

Run PSpice Optimizer. 0 items selected Scale=100% X=2.20 Y=0.10

OrCAD Optimizer - C:\... \SAMPLES\OPTIMIZE\BPF\BPF-Bpf.opt

File Edit Tune Options Help

**Specifications**

- Fc
- BW
- Gain

**Parameters**

- aFc
- aGain
- aBW

RMS Error:

Iteration:

Simulations:

**Edit Specification** ✖

Name:   Enabled

Reference:  Internal  External Weight:

**Internal**

Target:

Range:

Constraint

Type:  ▾

**External**

File:  ...

X Column Name:

Y Column Name:

Tolerance:

**Analysis**

Simulation Profile or Circuit File  AC  DC  Tran

...

Probe File Containing Goal Functions:

...

Evaluate:

OK Cancel

**Edit Specification** ✖

Name:   Enabled

Reference:  Internal  External Weight:

**Internal**

Target:

Range:

Constraint

Type:  ▾

**External**

File:  ...

X Column Name:

Y Column Name:

Tolerance:

**Analysis**

Simulation Profile or Circuit File  AC  DC  Tran

...

Probe File Containing Goal Functions:

...

Evaluate:

OK Cancel

**Edit Specification** ✖

Name:   Enabled

Reference:  Internal  External Weight:

**Internal**

Target:

Range:

Constraint

Type:  ▾

**External**

File:  ...

X Column Name:

Y Column Name:

Tolerance:

**Analysis**

Simulation Profile or Circuit File  AC  DC  Tran

...

Probe File Containing Goal Functions:

...

Evaluate:

OK Cancel

# Результаты расчета параметров при оптимизации

OrCAD Optimizer - C:\... \SAMPLES\OPTIMIZE\BPF\BPF-Bpf.opt\* [Derivs. Avail]

File Edit Tune Options Help

**Specifications**

<input checked="" type="checkbox"/> Fc	<input checked="" type="checkbox"/> BW	<input checked="" type="checkbox"/> Gain
9.98953	1.00777	10.3499
8.3222	0.712187	

RMS Error: 2.156e-001

<input checked="" type="checkbox"/> aFc	<input checked="" type="checkbox"/> aGain
0.457928	0.476062
0.5	0.5

Iteration: 3

Simulations: 9

Optimization complete. S

OrCAD Optimizer - C:\... \SAMPLES\OPTIMIZE\BPF\BPF-Bpf.opt [Derivs. Avail]

File Edit Tune Options Help

- Parameters...
- Specifications...
- Store Values
- Reset Values
- Round Nearest
- Round Calculated
- Update Schematic

**Specifications**

<input checked="" type="checkbox"/> BW	<input checked="" type="checkbox"/> Gain
1.00777	10.3499
0.712187	14.8106

RMS Error: 2.156e-001

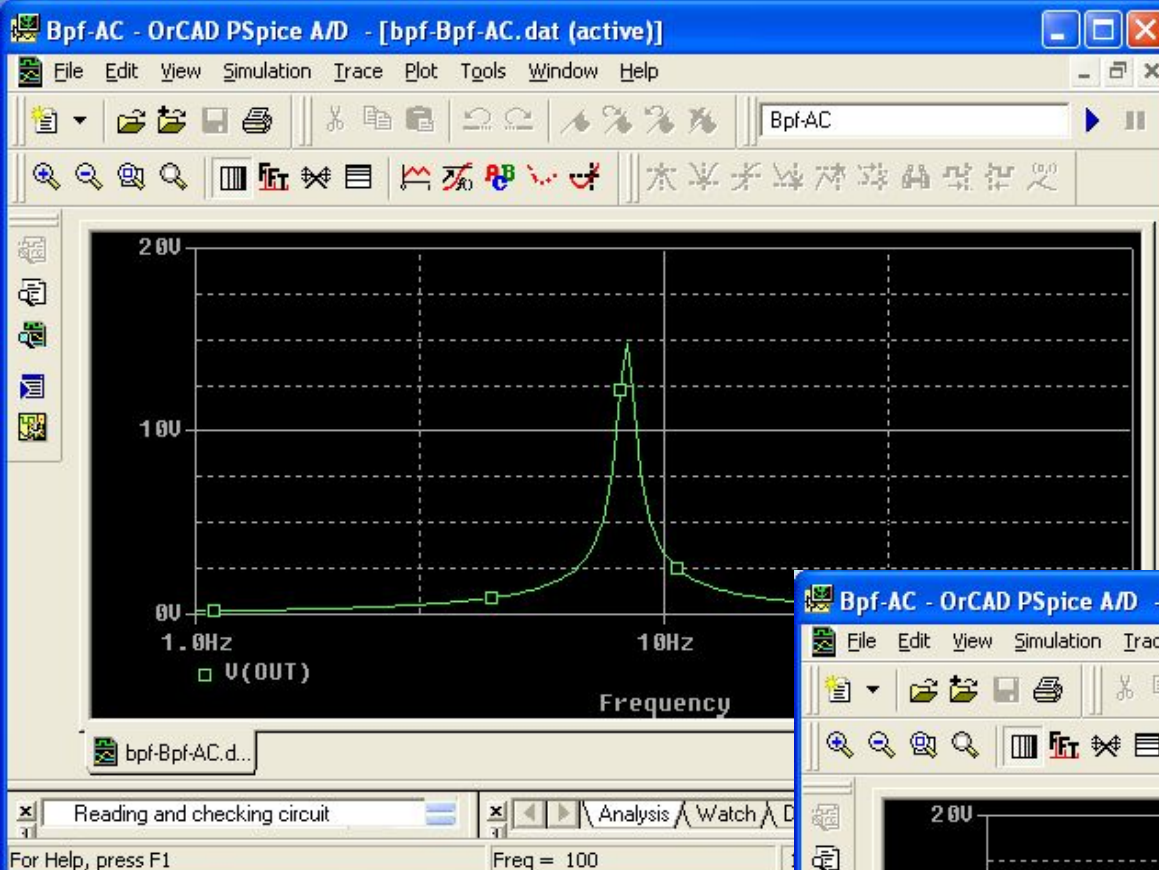
Iteration: 3

Simulations: 9

**Parameters**

<input checked="" type="checkbox"/> aFc	<input checked="" type="checkbox"/> aGain	<input checked="" type="checkbox"/> aBW
0.457928	0.476062	0.702911
0.5	0.5	0.5

Schematic updated.



До оптимизации

После  
оптимизации

