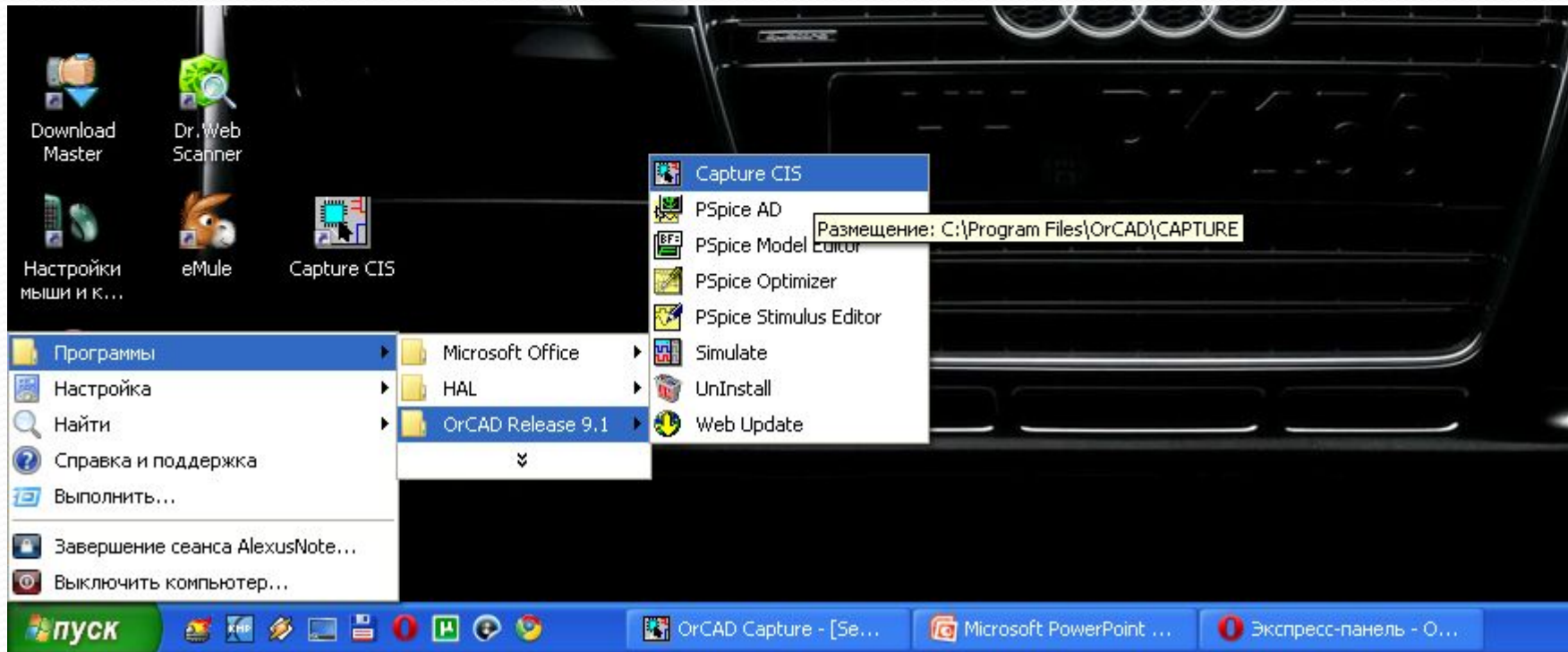


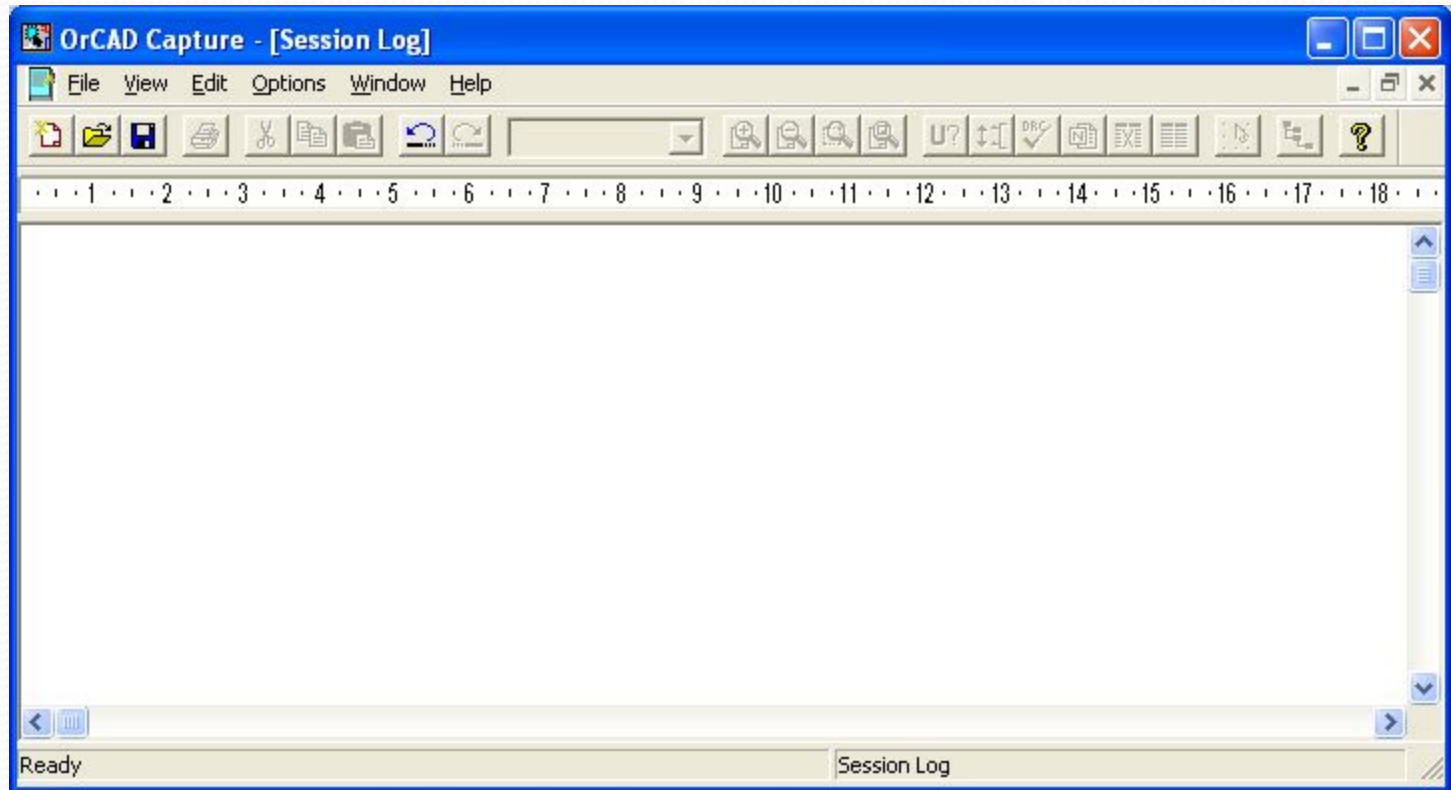
Основы работы в OrCad

Запуск программы и первое знакомство с интерфейсом

Запуск осуществляется при помощи ярлыка из меню ПУСК или Рабочего стола

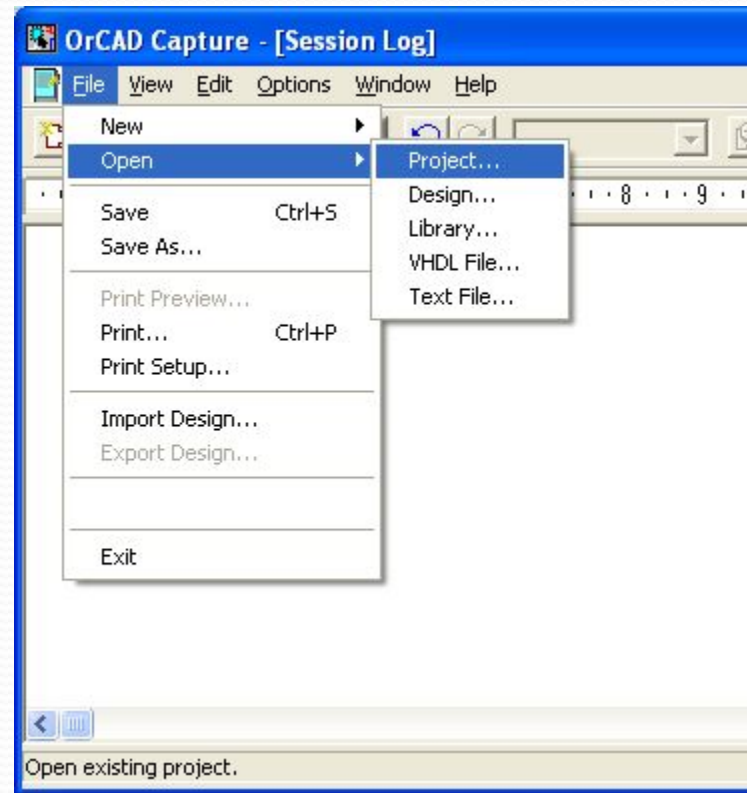
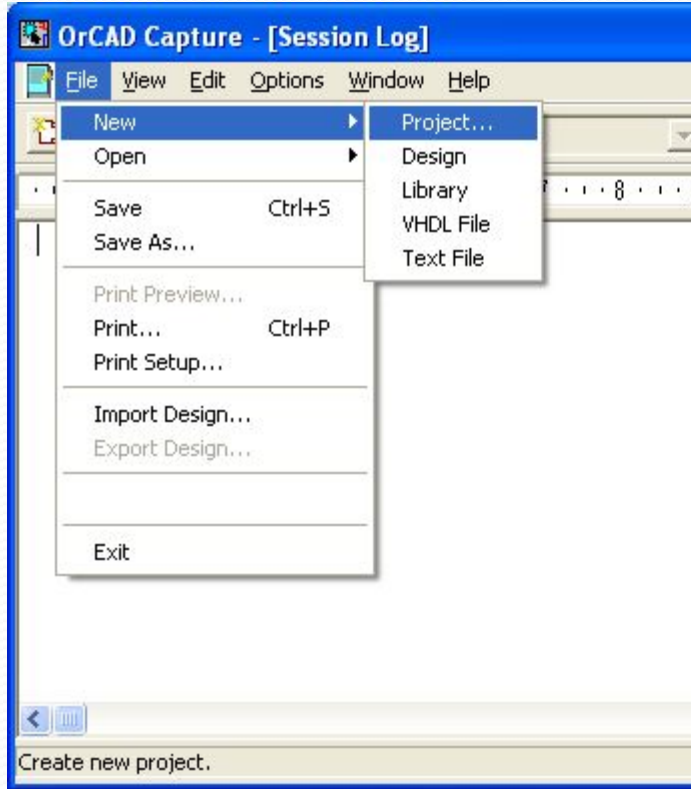


Главное окно программы

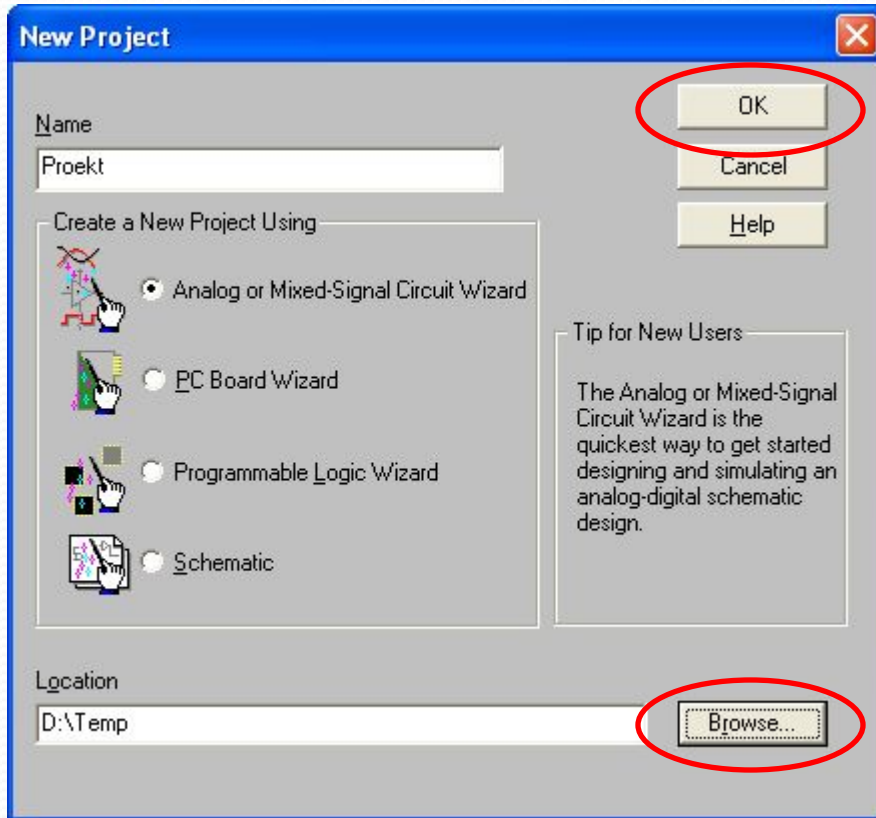


Работа с Меню

Создание (открытие) нового проекта



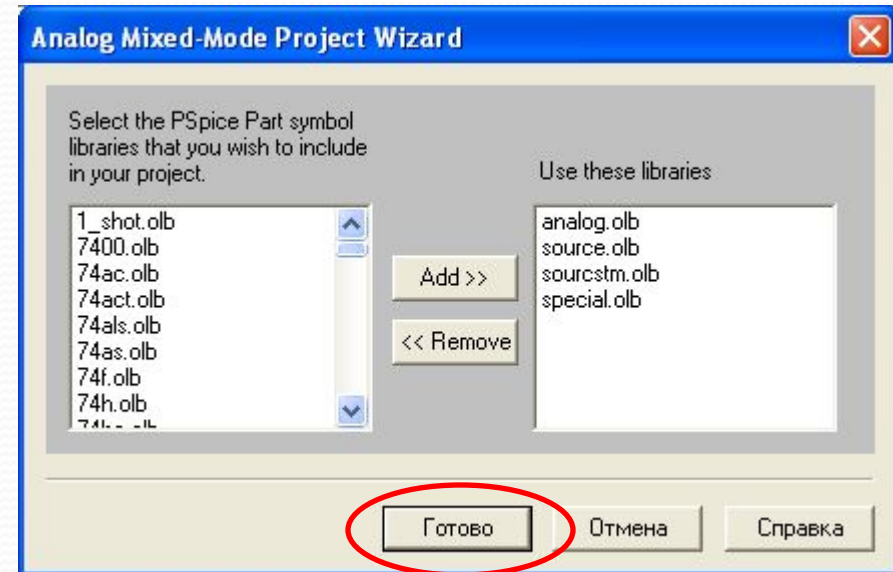
Создание нового проекта



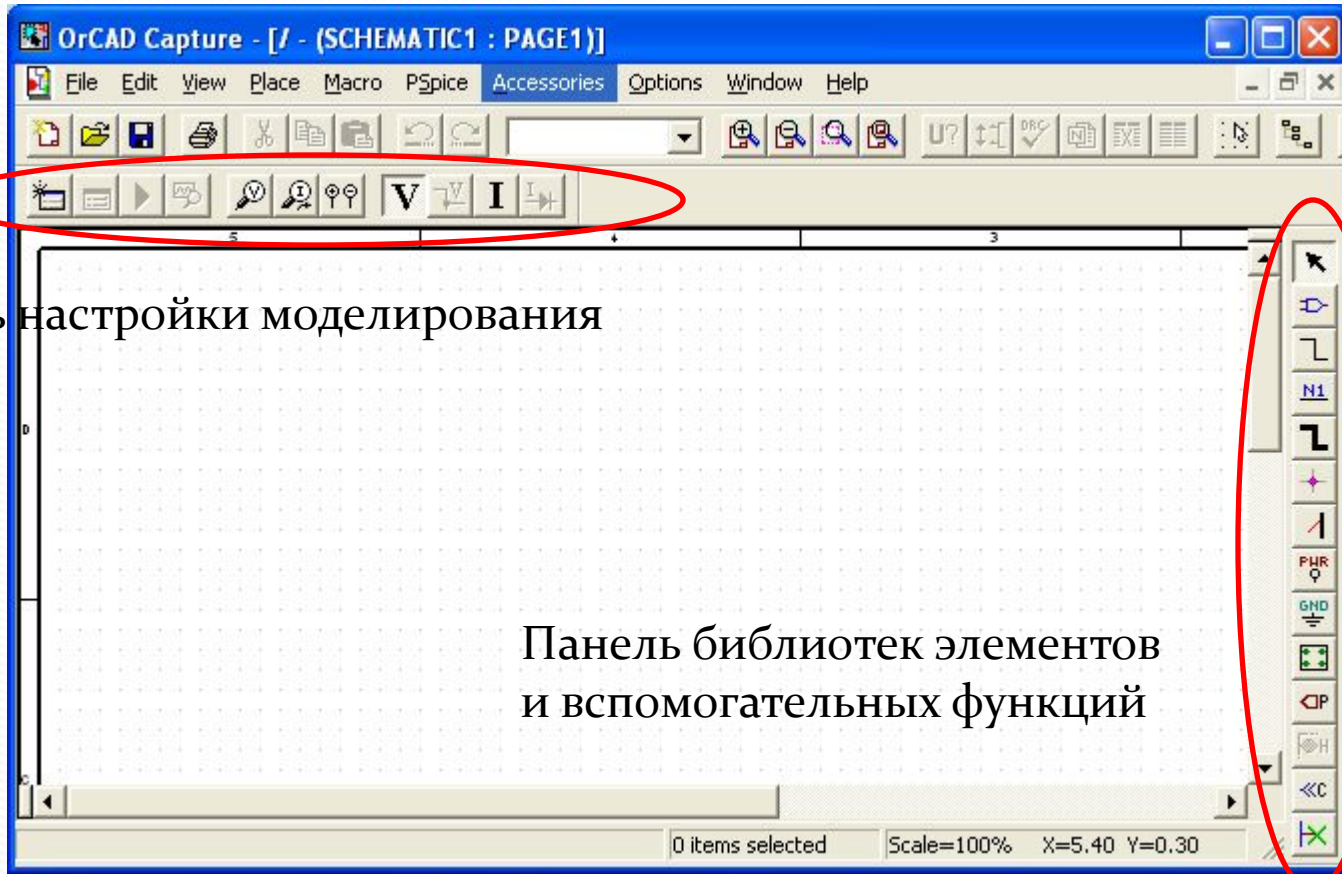
Выполняем процедуру представленную на предыдущем слайде, выбираем «Аналоговая или смешанная схема», указываем путь, нажимаем **ОК** (имя и путь **только латинскими** буквами)



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



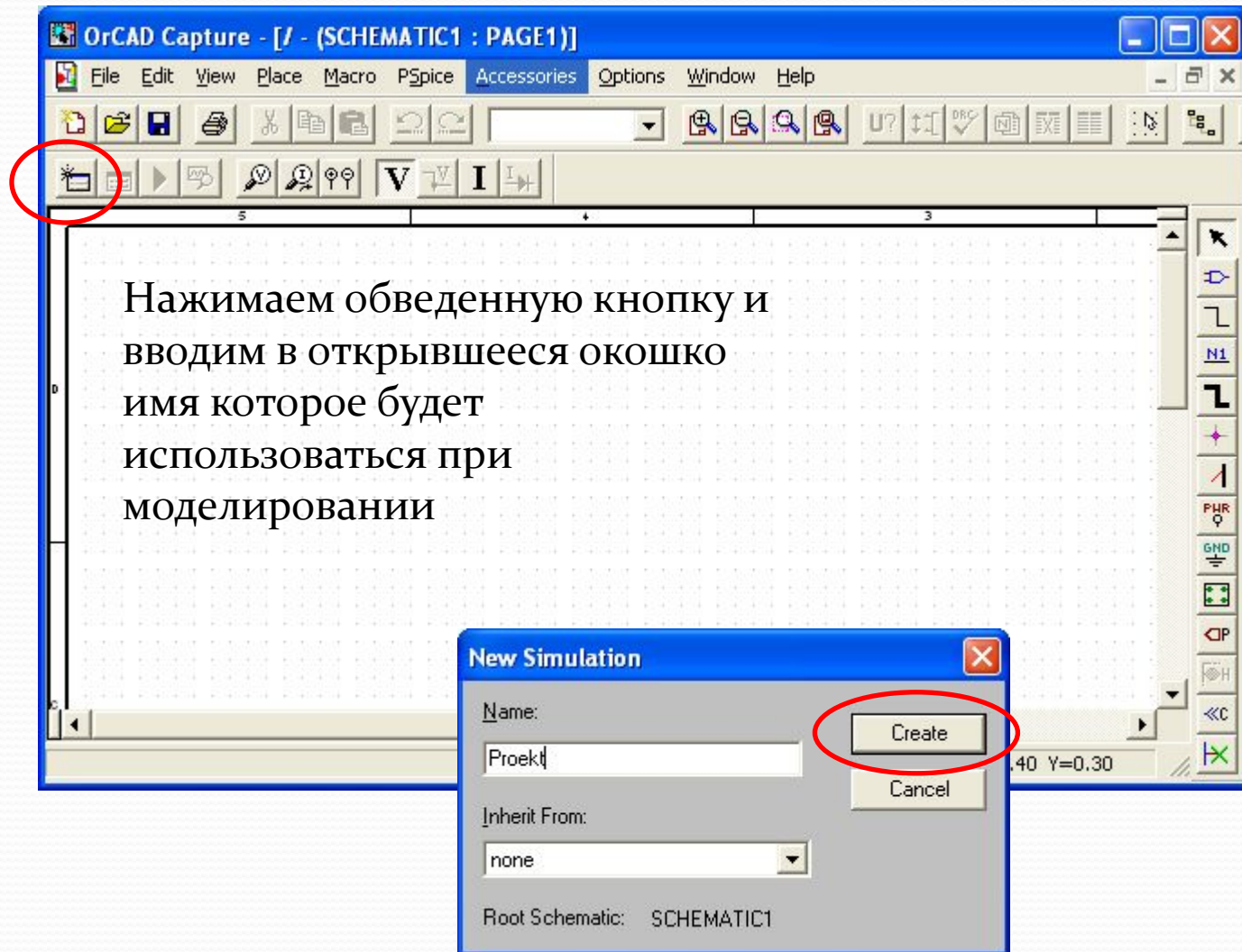
Создание нового проекта



Панель настройки моделирования

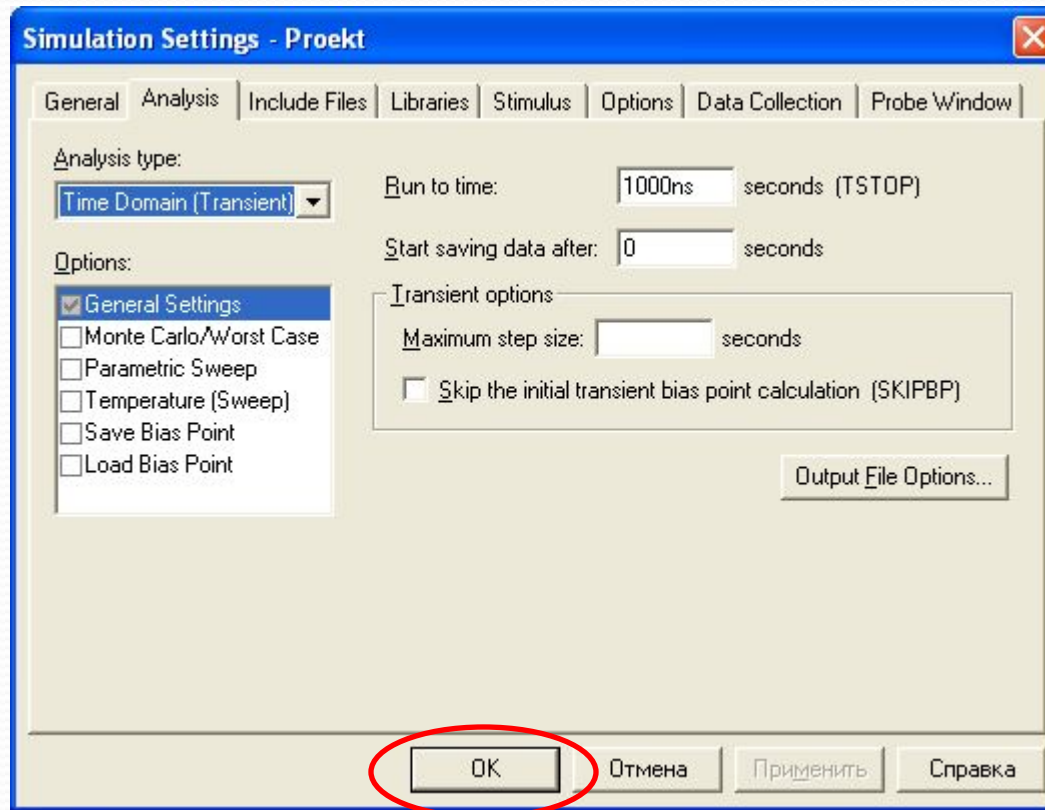
Панель библиотек элементов
и вспомогательных функций

Создание нового проекта



Создание нового проекта

После задания имени профиля моделирования открывается окошко настройки параметров моделирования



Тип анализа

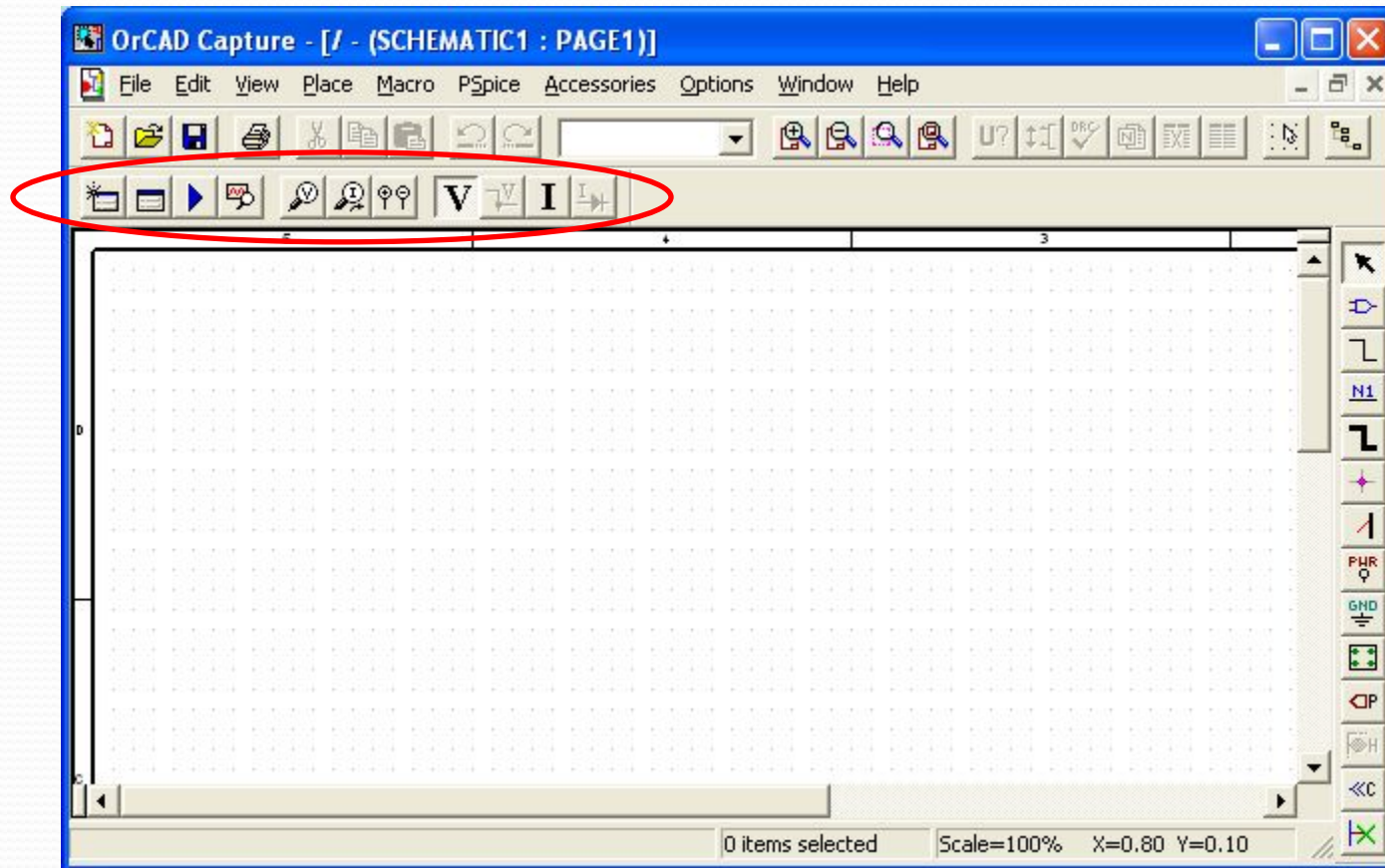
Время расчета

Опции анализа

Шаг расчета

Пока ничего не настраиваем и просто нажимаем ОК

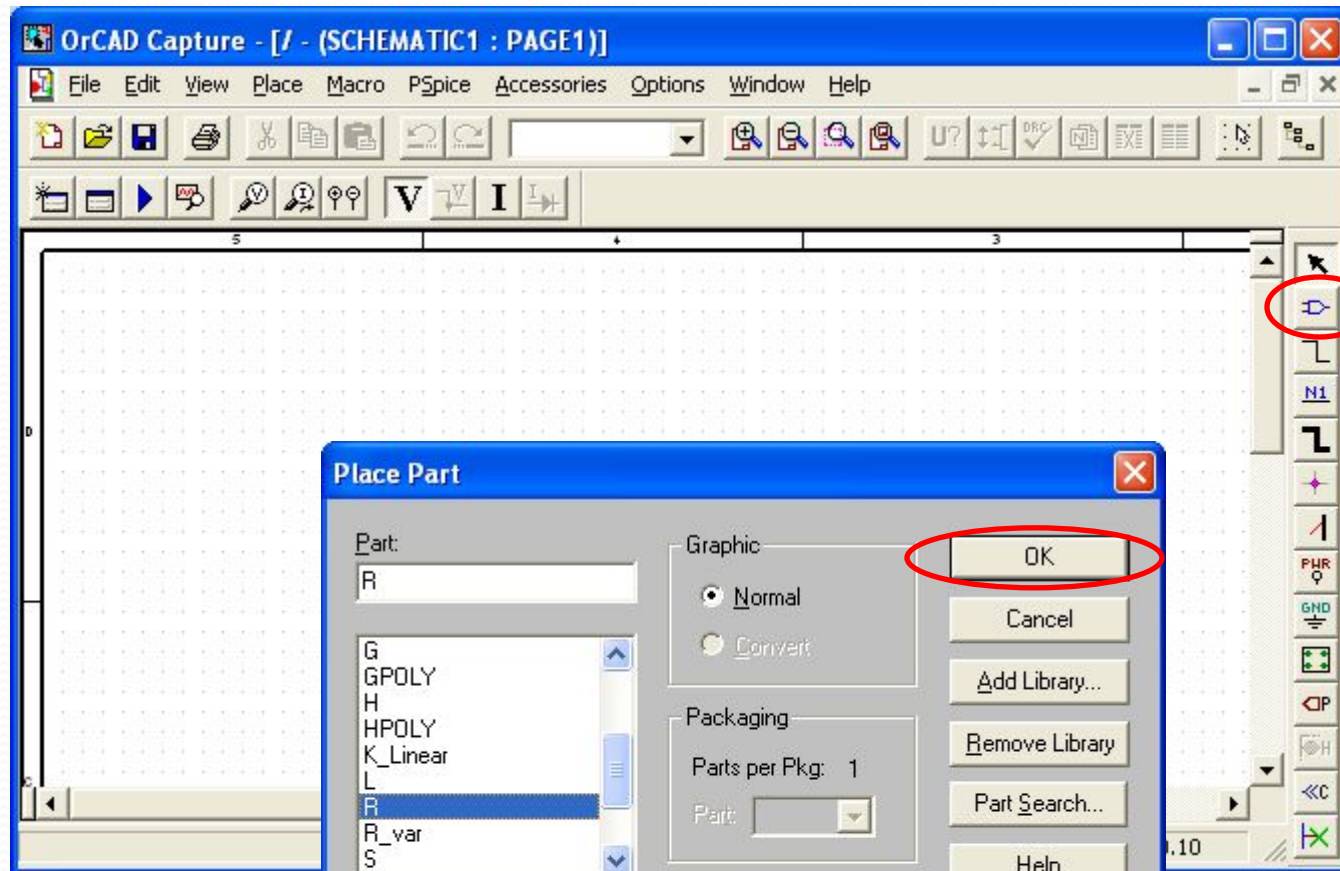
Создание нового проекта



Обведенная панелька стала активной

Можно начинать рисовать схему

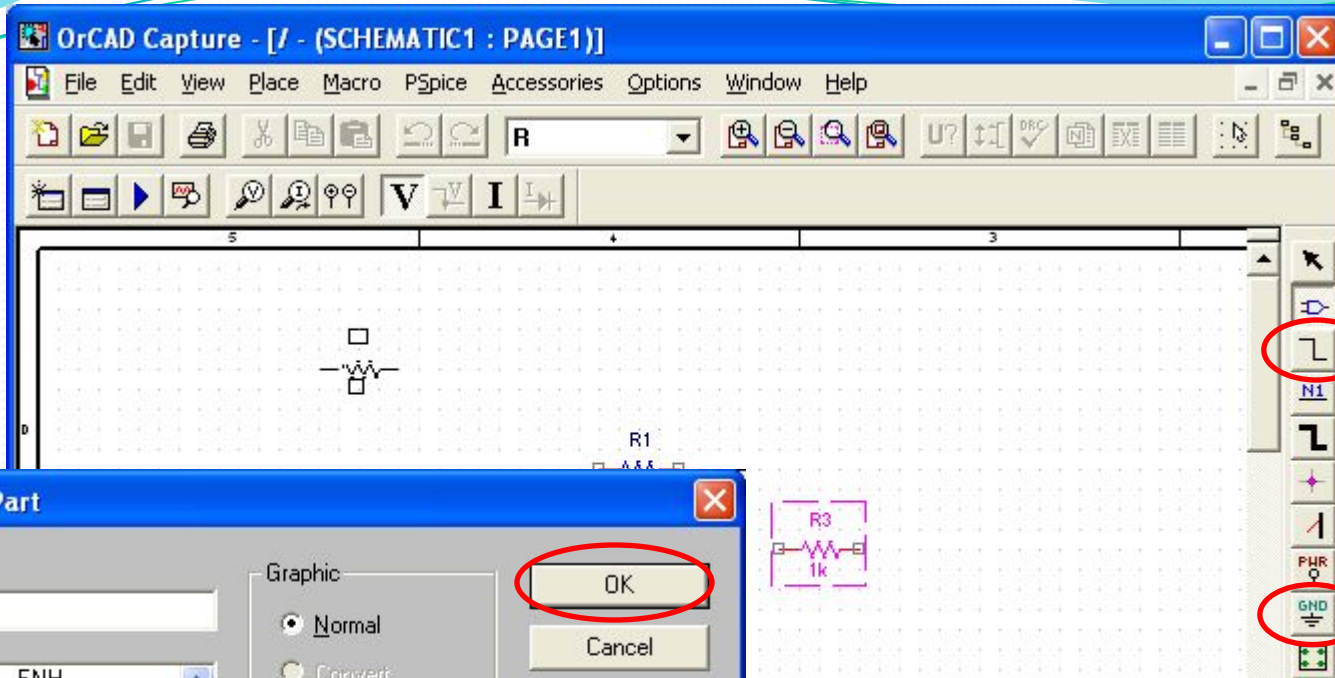
Создание нового проекта



нажимаем

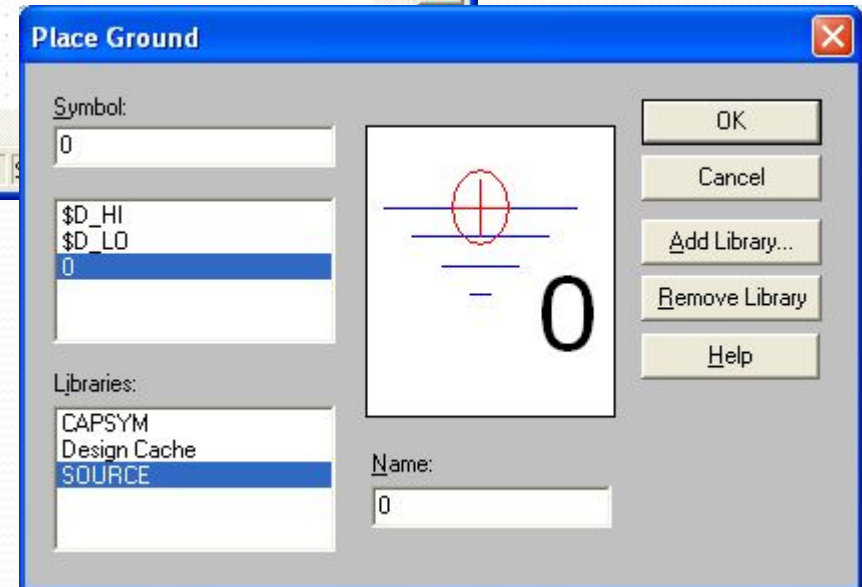
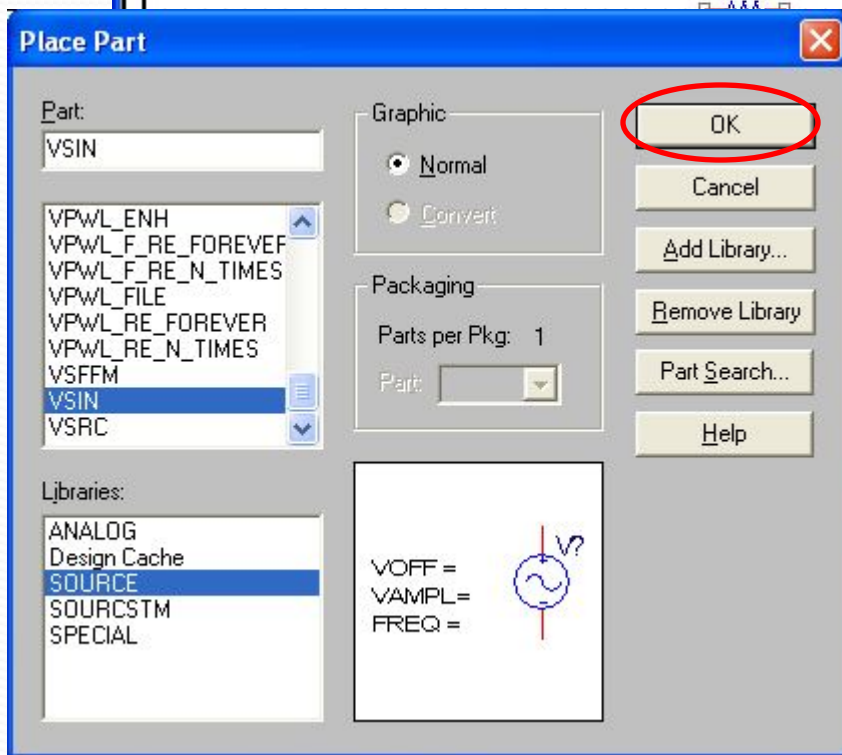
Выбираем
элементы из
библиотек

Создание нового проекта

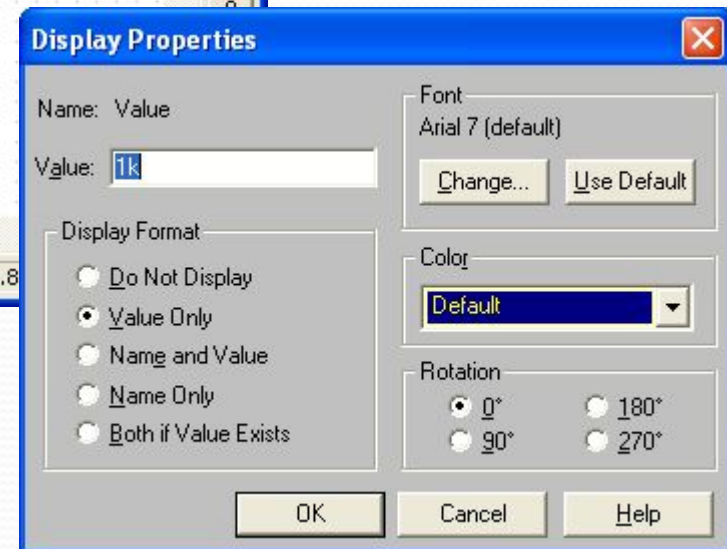
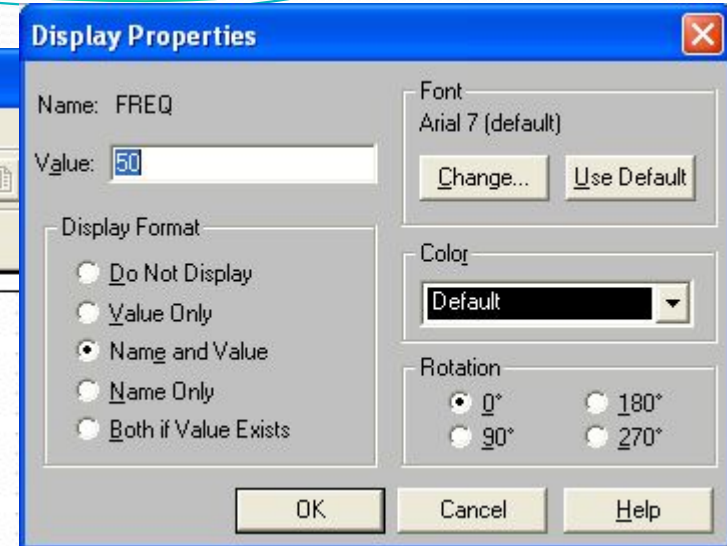
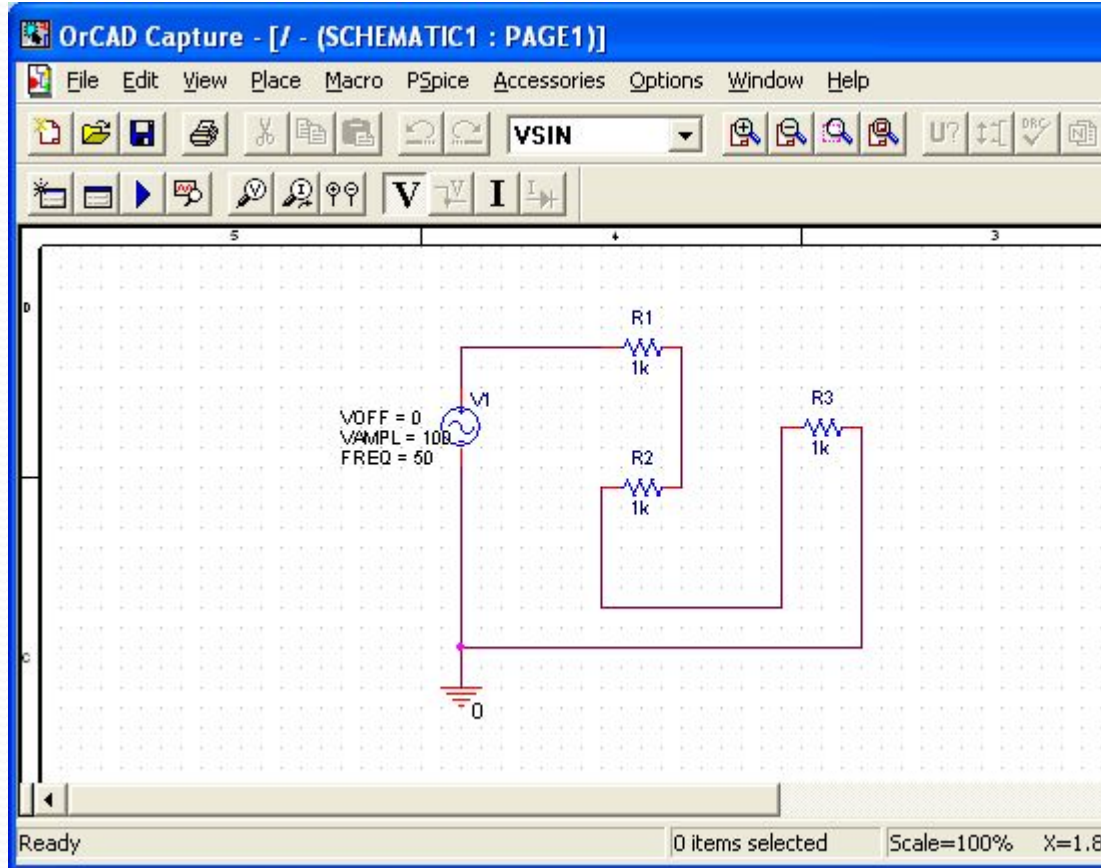


Соединение

Заземление



Создание нового проекта

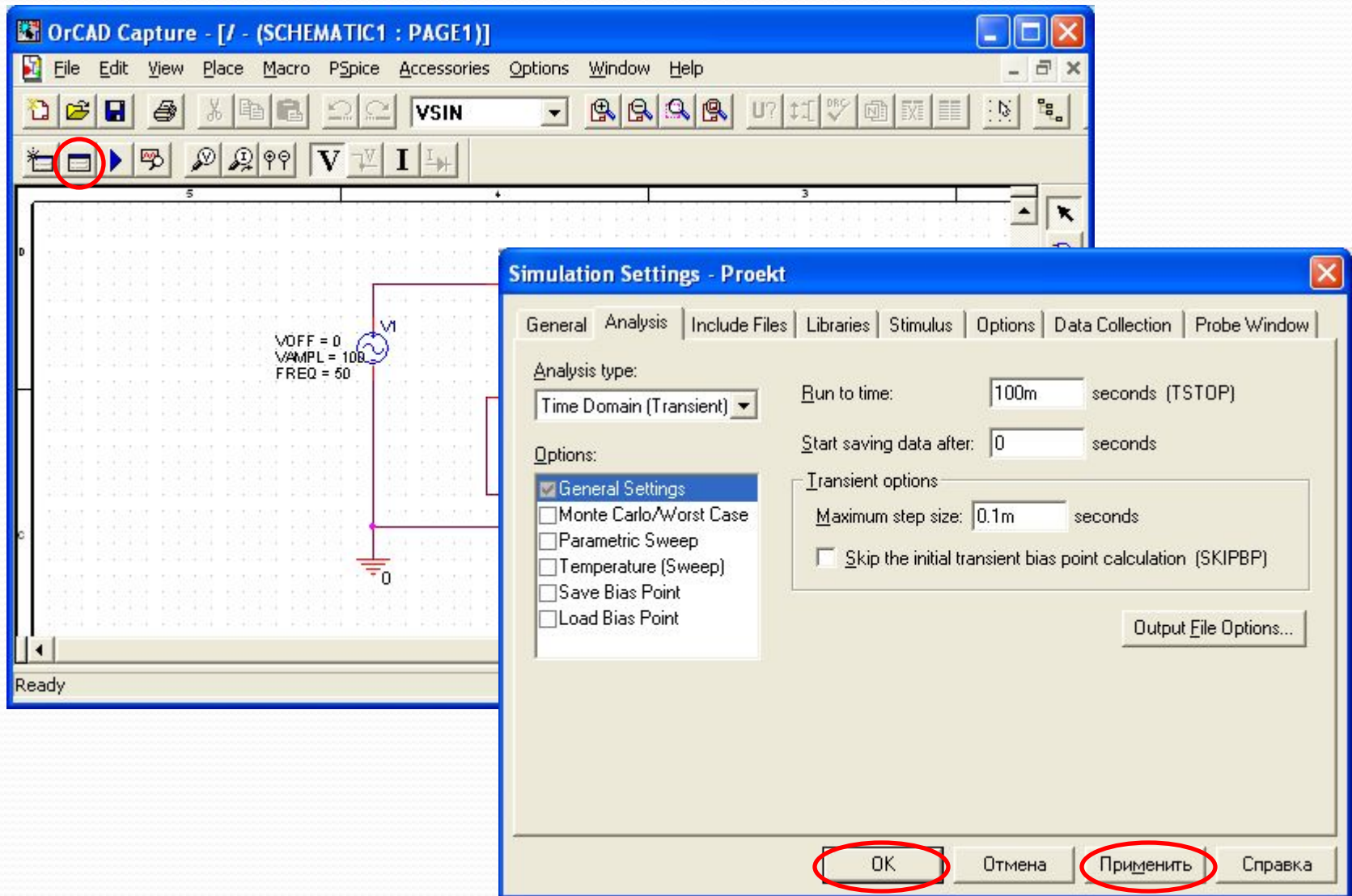


Изменение номинала резистора и настройки источника меняются путем двукратного нажатия левой клавиши мыши

Создание нового проекта

Настройки моделирования во временной области

Жмем



Создание нового проекта

Моделирование и вывод результата

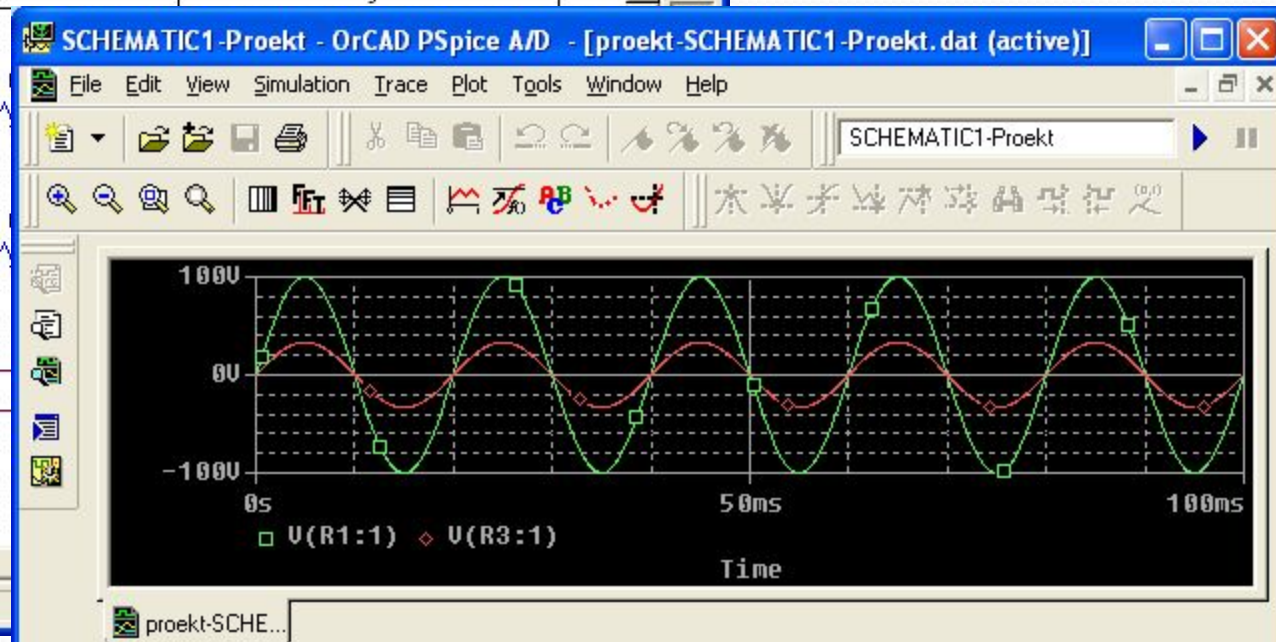
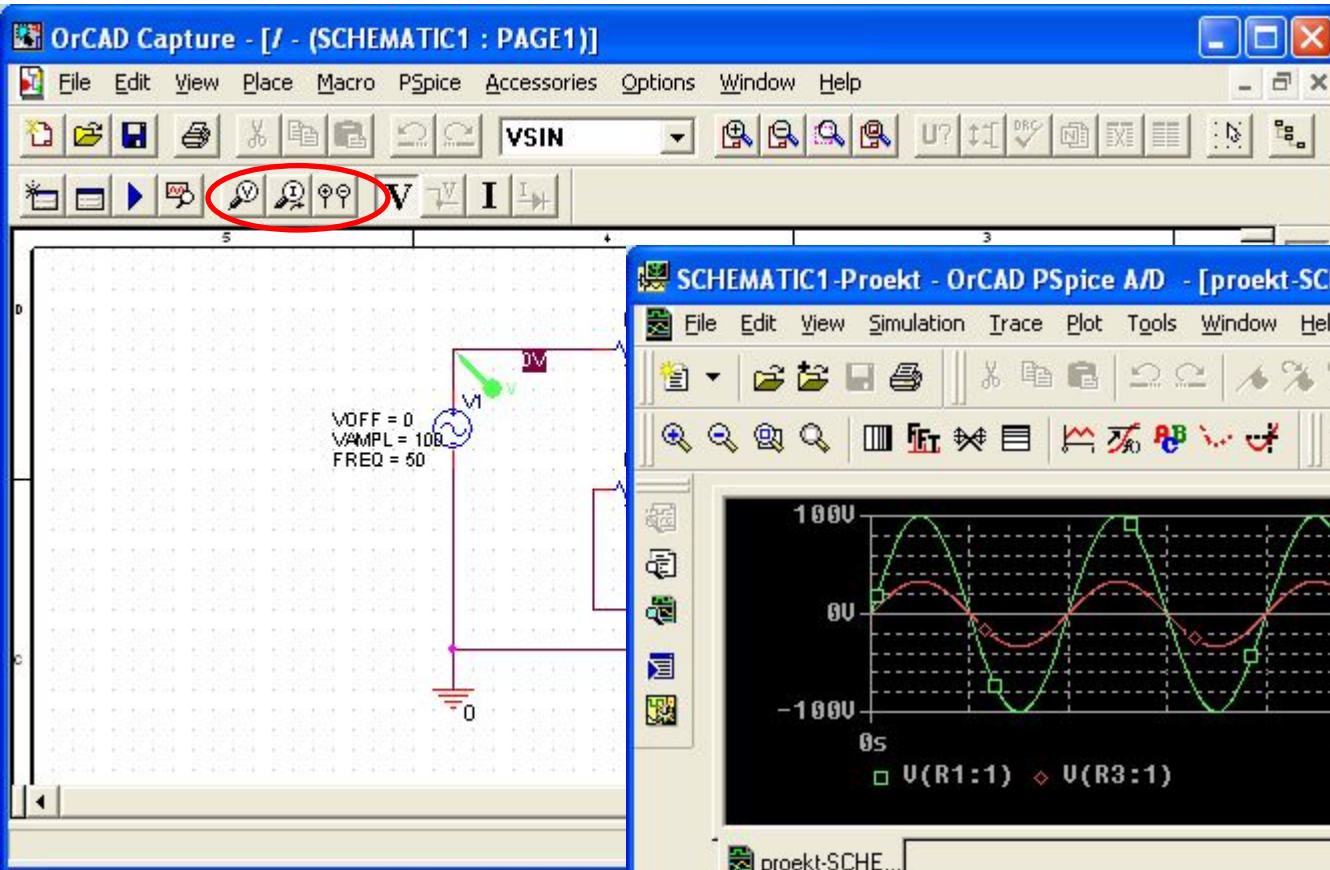
The image displays two overlapping windows from the OrCAD software suite. The top window, titled "OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]", shows a schematic diagram of a circuit. A voltage source is connected to a ground symbol. The voltage source parameters are: $V_{OFF} = 0$, $V_{AMPL} = 100$, 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 very bottom of the PSpice window shows "Time = .1" and "100%".

Создание нового проекта

Моделирование и вывод результата

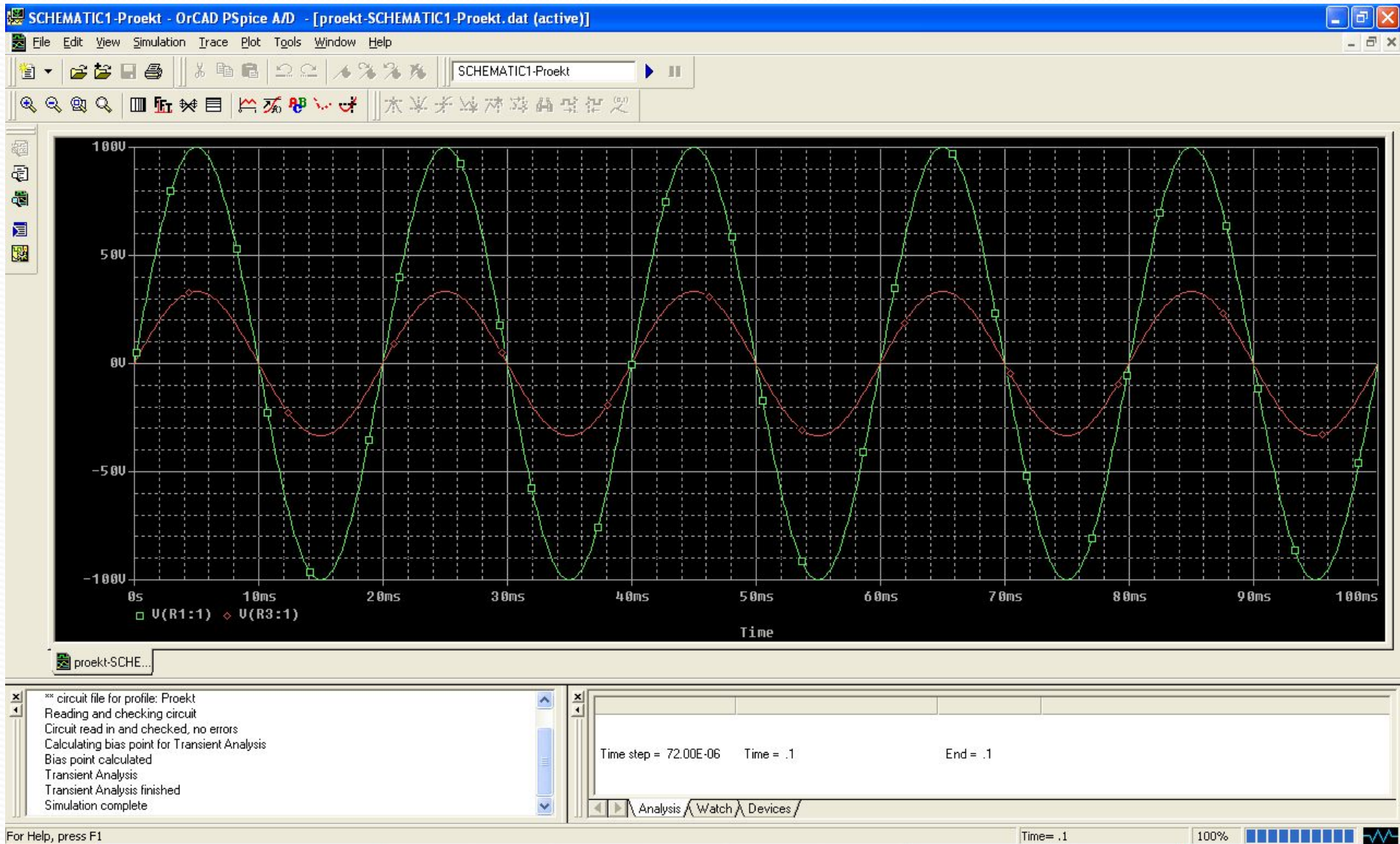


```
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point for Transient Analy
Bias point calculated
Transient Analysis
Transient Analysis finished
Simulation complete
```

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

Создание нового проекта

Моделирование и вывод результата



Создание нового проекта

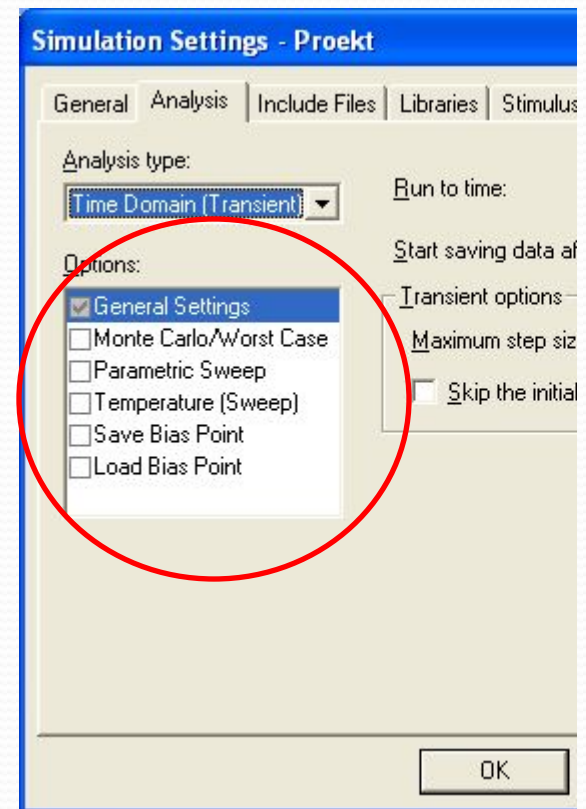
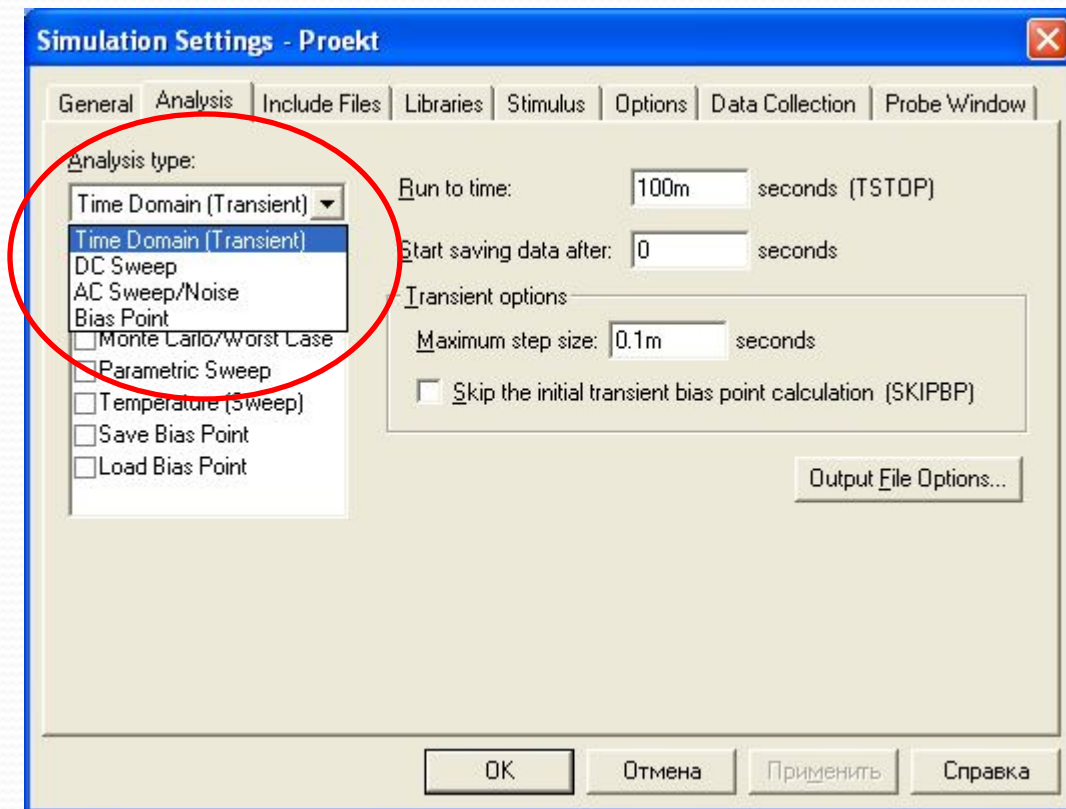
Моделирование и вывод результата

The screenshot shows the OrCAD PSpice A/D interface. The main window displays a simulation plot with a y-axis ranging from -1000 to 1000 and an x-axis labeled '0s'. The plot shows two waveforms: a green one and a red one. A legend at the bottom of the plot indicates 'U(R1:1)' and '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 of the window shows 'Add trace[s] to the selected plot' and 'Time= .1'.

The 'Add Traces' dialog box is shown, allowing the user to select 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 such as ABS(), ARCTAN(), ATAN(), AVG(), AVGX(.), COS(), D(), DB(), ENVMAX(.), ENVMIN(.), EXP(), G(), IMG(), LOG(), LOG10(), M(), and MAX(). The 'Analog' checkbox is checked, and 'Currents' is also checked. The 'Trace Expression' field is empty. The dialog has 'OK', 'Cancel', and 'Help' buttons.

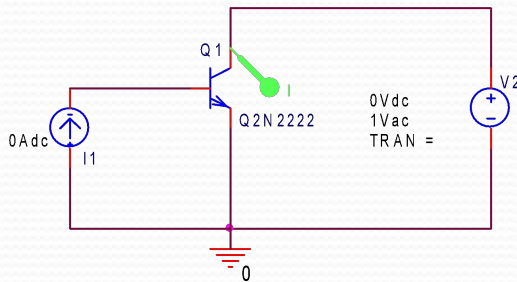
Виды моделирования

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

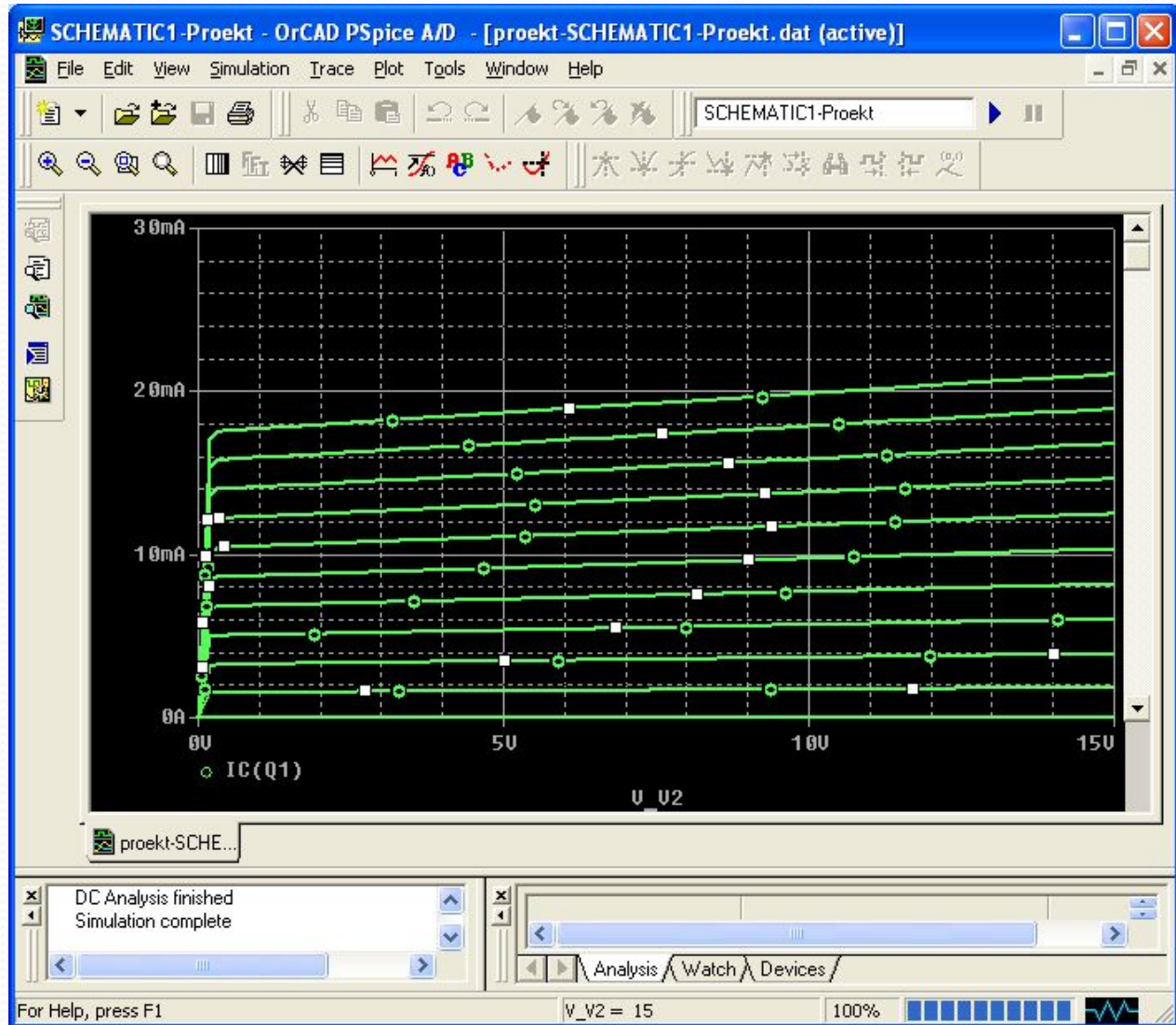
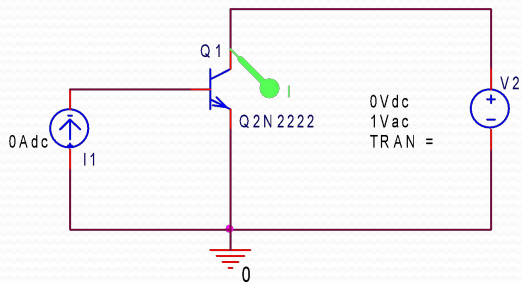
Sweep variable:
 Voltage source Name: I1
 Current source Model type:
 Global parameter Model name:
 Model parameter Parameter name:
 Temperature

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

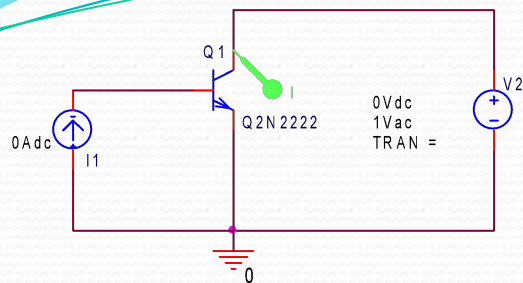
OK Отмена Применить Справка

Виды моделирования

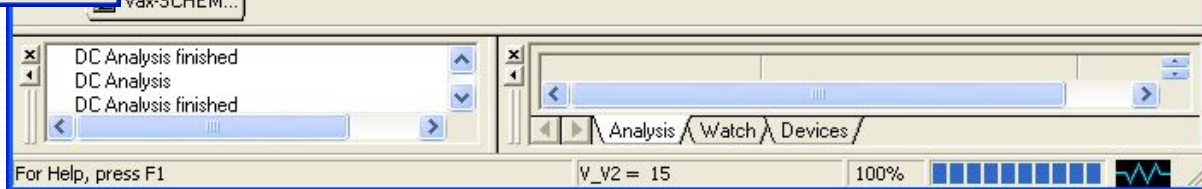
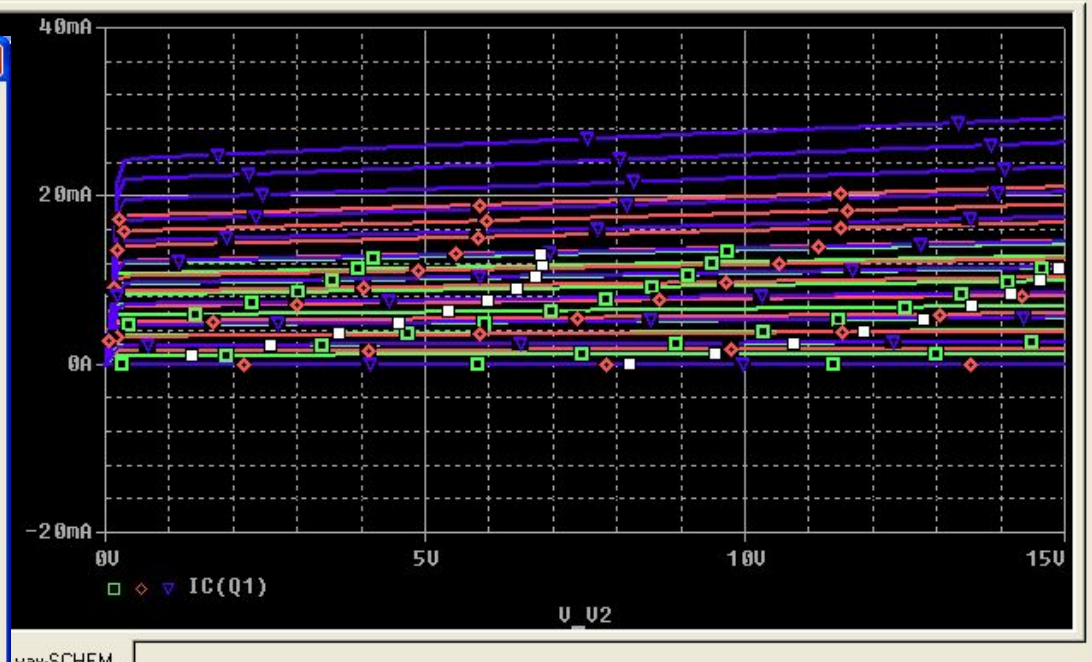
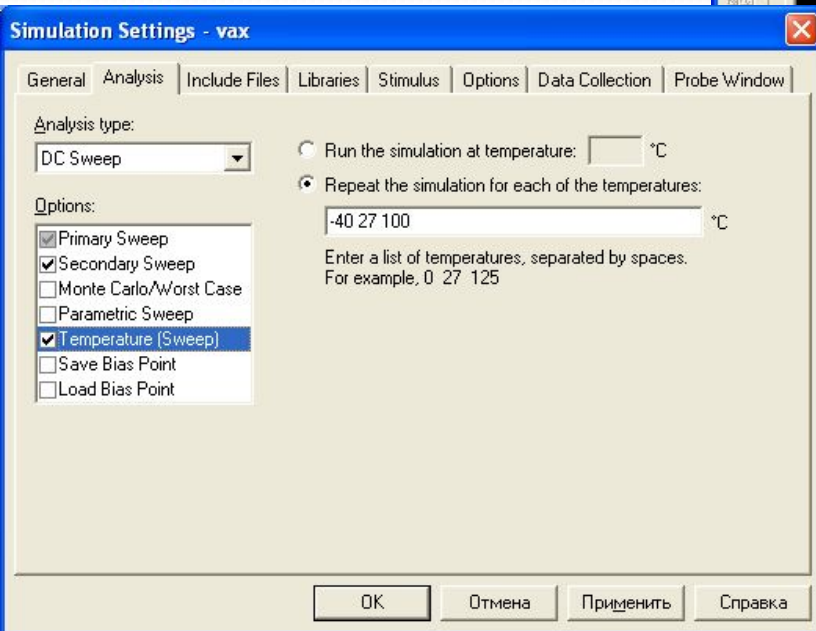
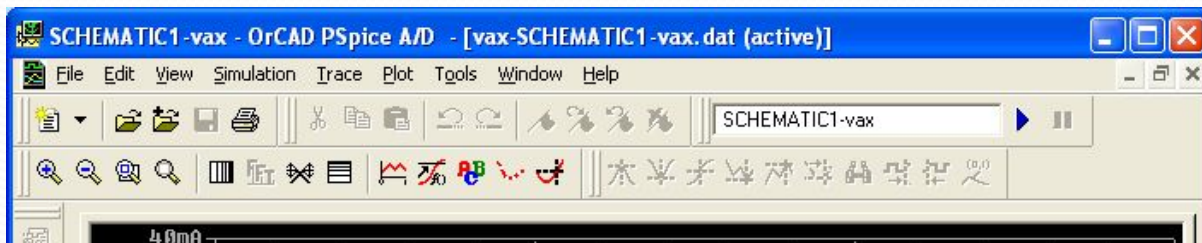
Расчет по постоянному току.
Снятие ВАХ полупроводниковых приборов



Виды моделирования

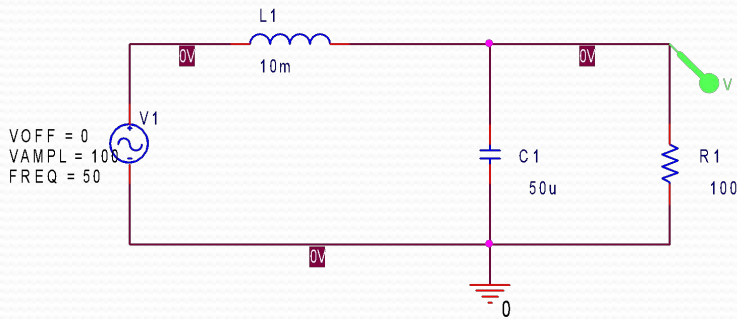


Расчет по постоянному току.
Снятие ВАХ полупроводниковых приборов.
Температурный анализ.

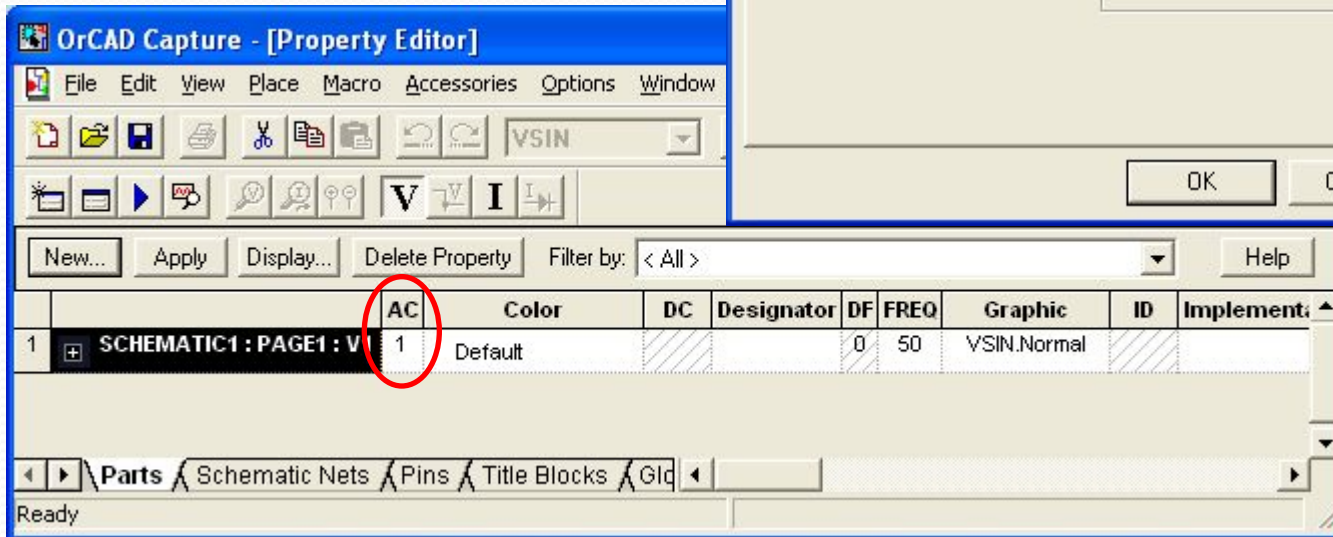
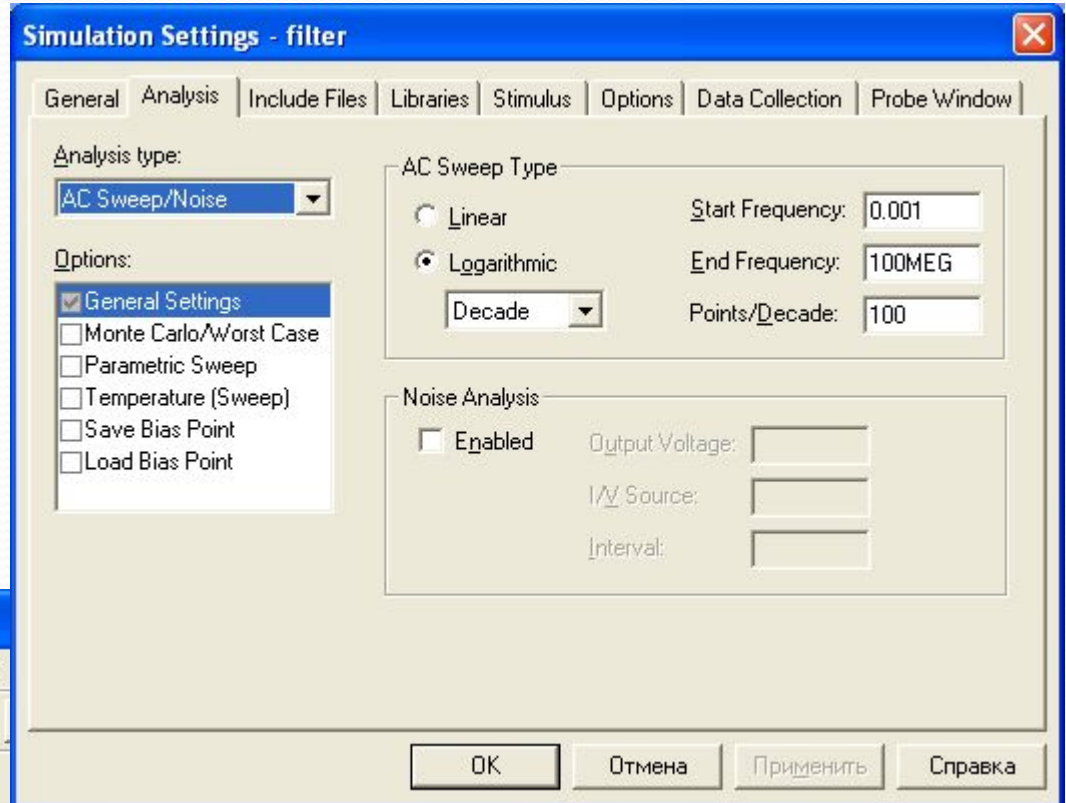


Виды моделирования

Моделирование в частотной области

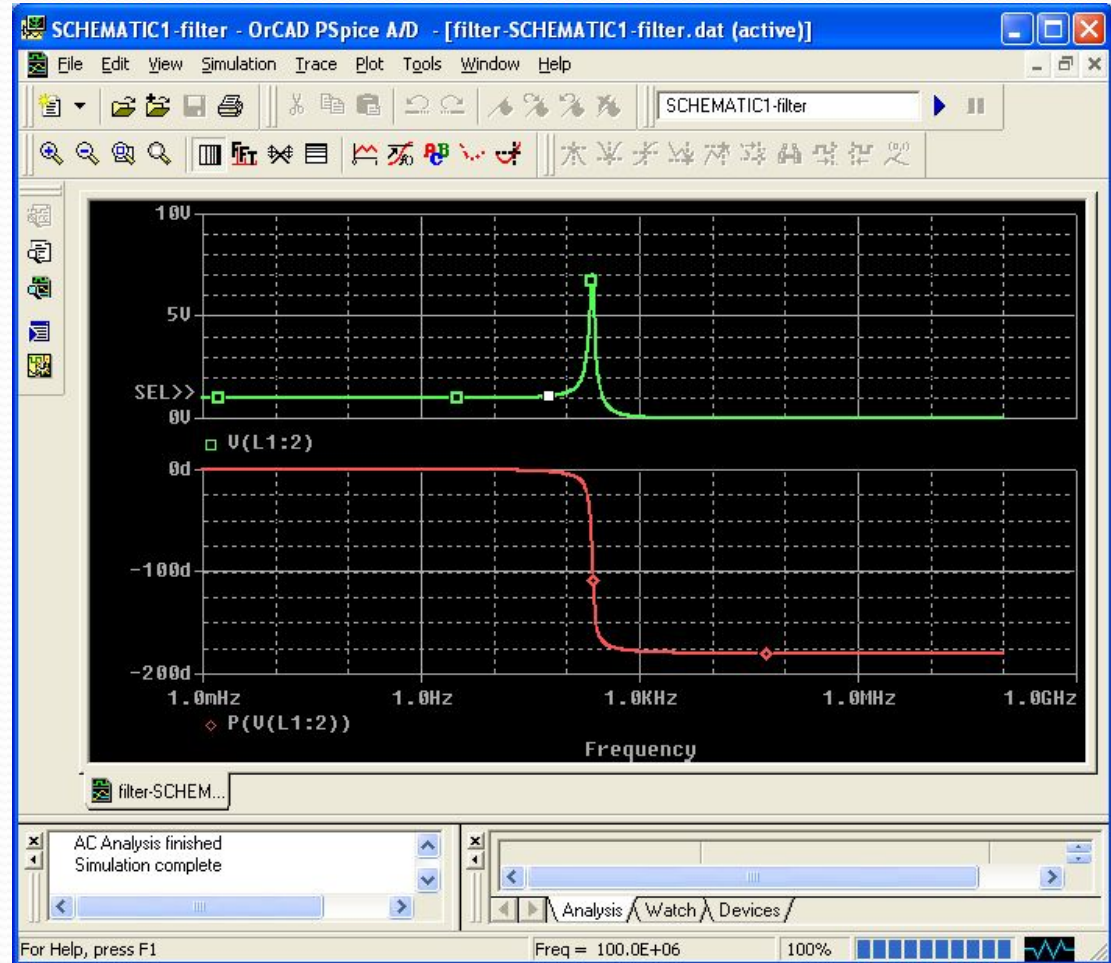
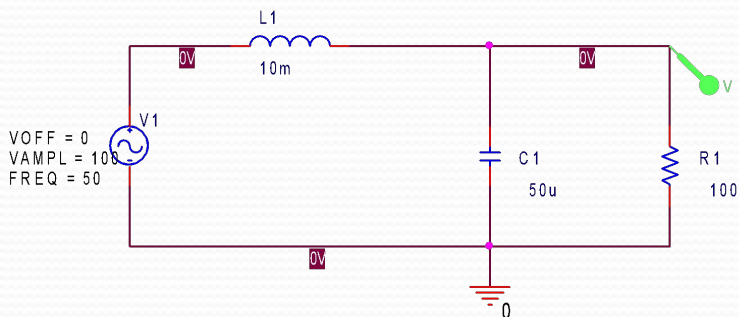


Дополнительные
настройки источника



Виды моделирования

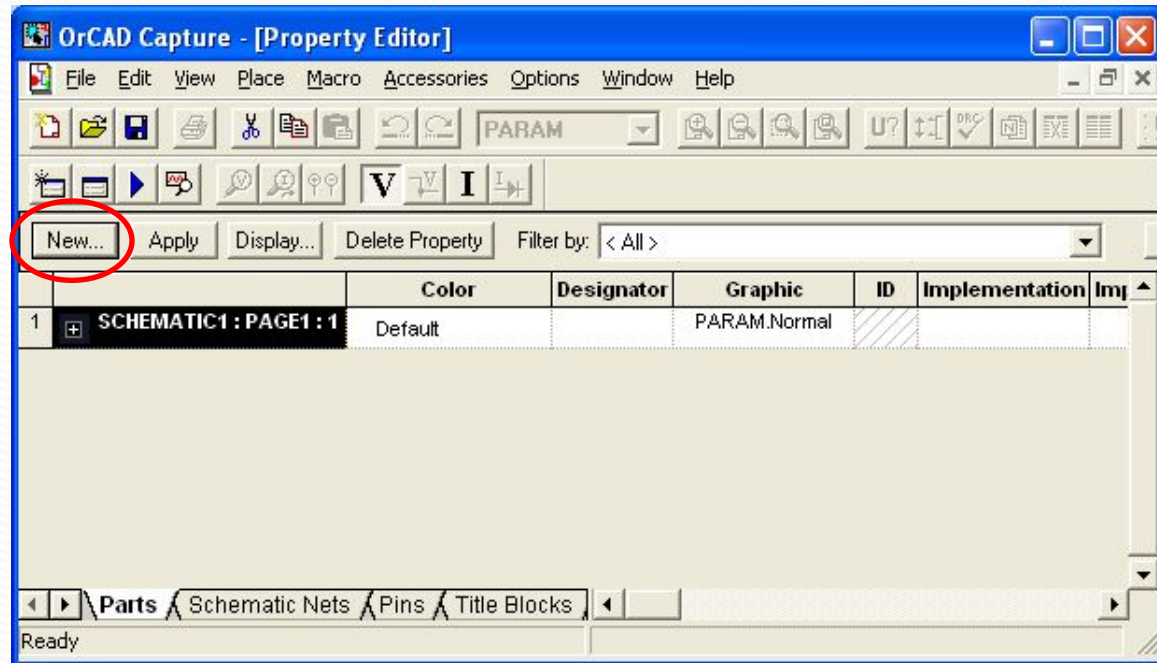
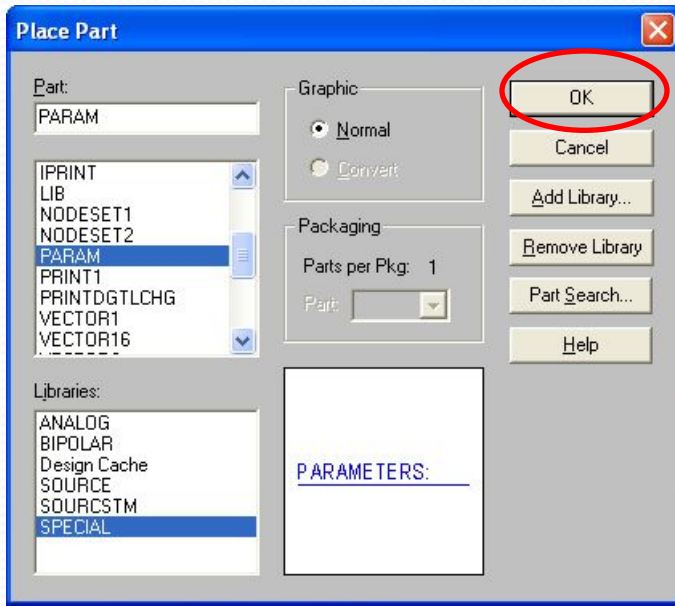
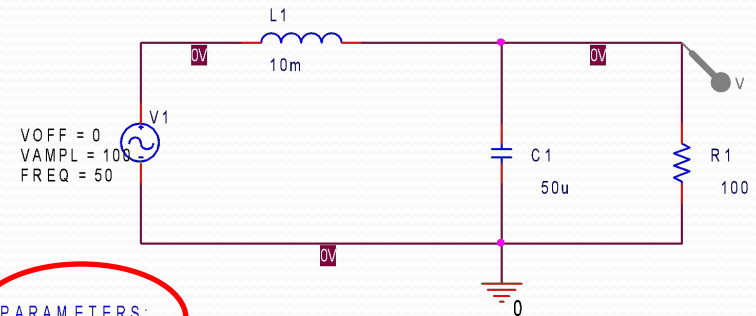
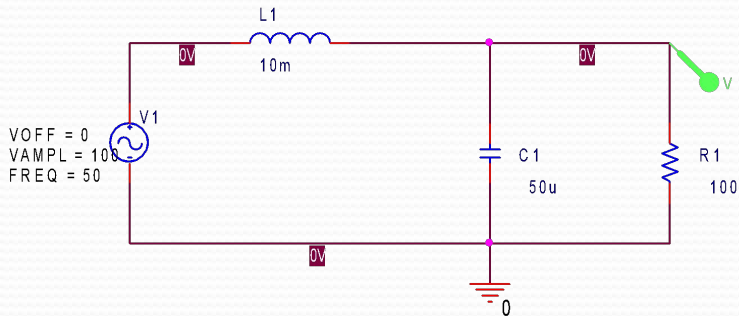
Моделирование в частотной области



АЧХ и ФЧХ Фильтра

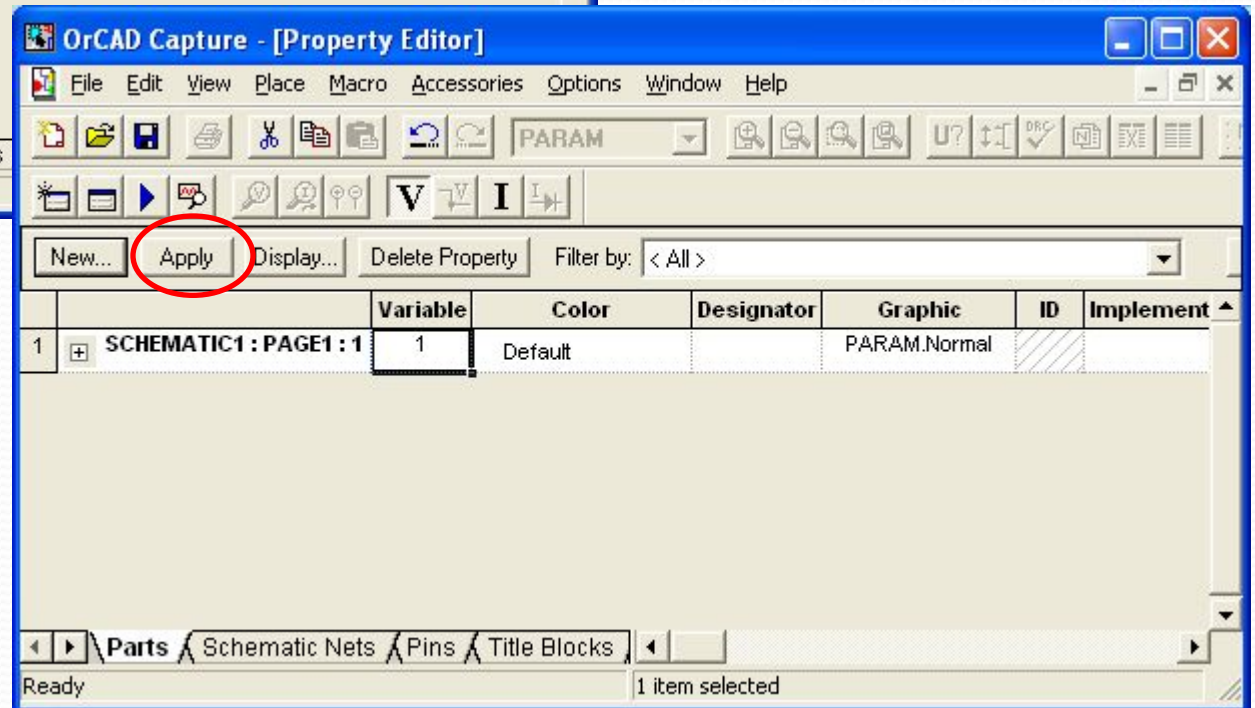
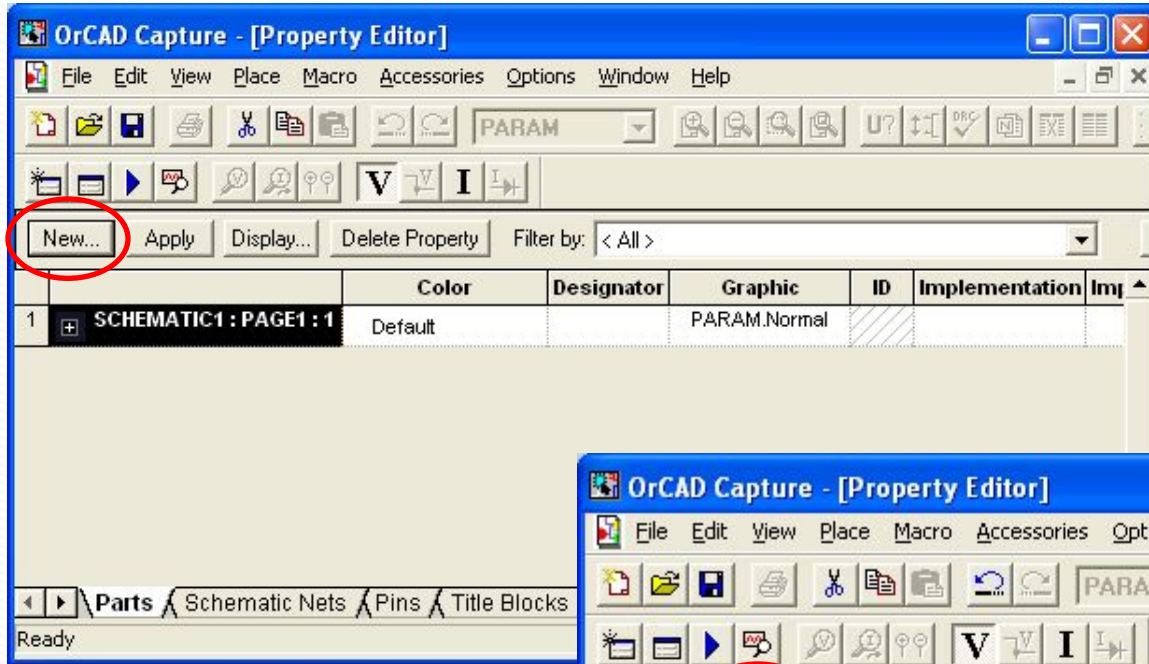
Виды моделирования

Параметрическое моделирование



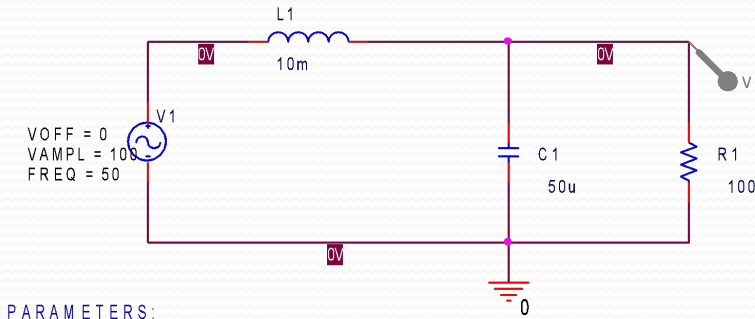
Виды моделирования

Параметрическое моделирование

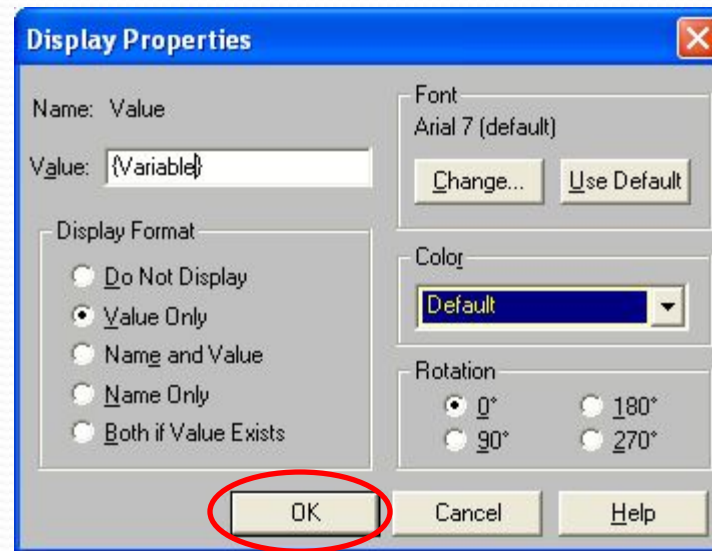


Виды моделирования

Параметрическое моделирование



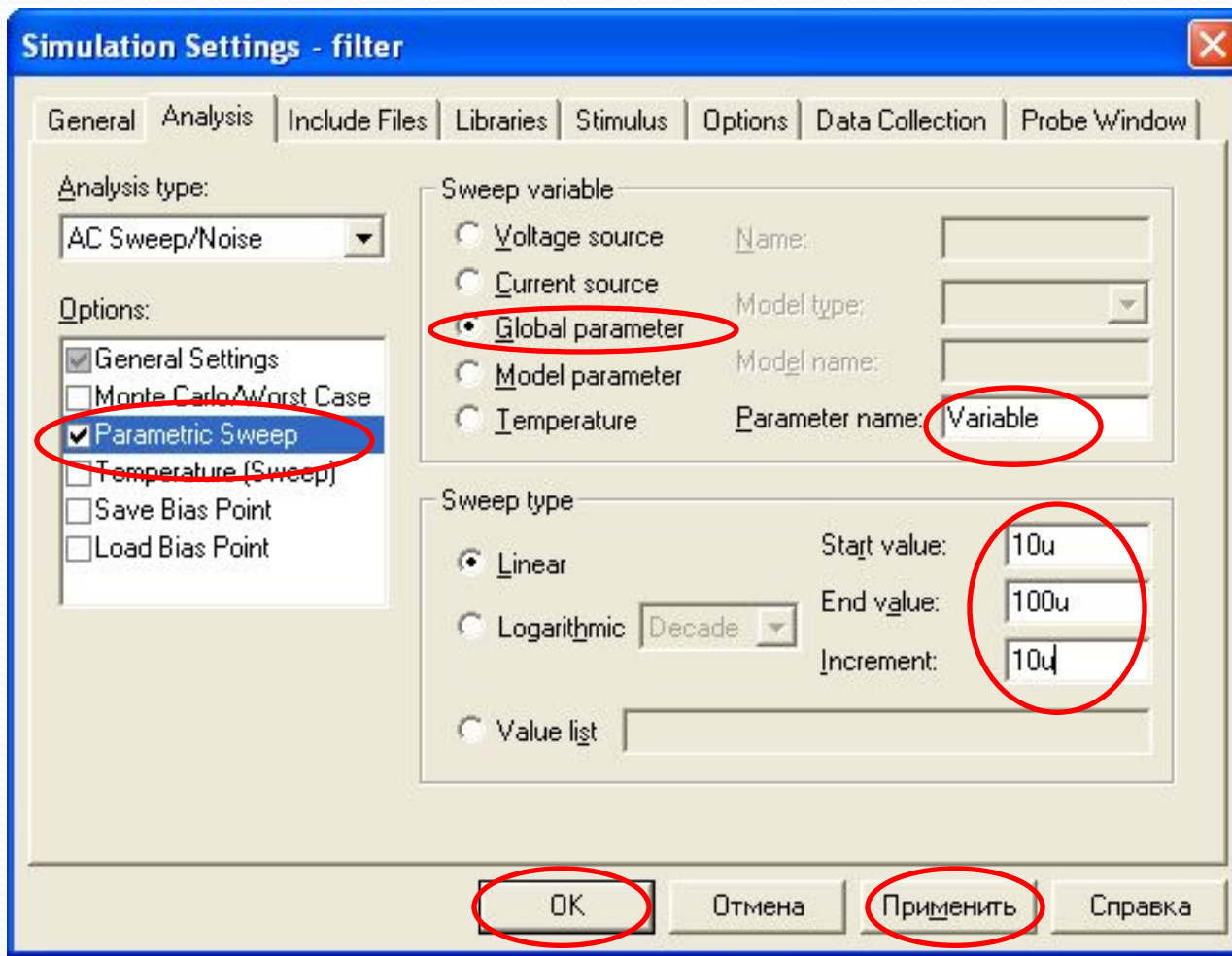
Далее щелкаем на номинале интересующего элемента (например конденсатора) и в фигурных скобках пишем имя переменной которую мы создали выше



Далее идем в настройки параметров моделирования и настраиваем

Виды моделирования

Параметрическое моделирование

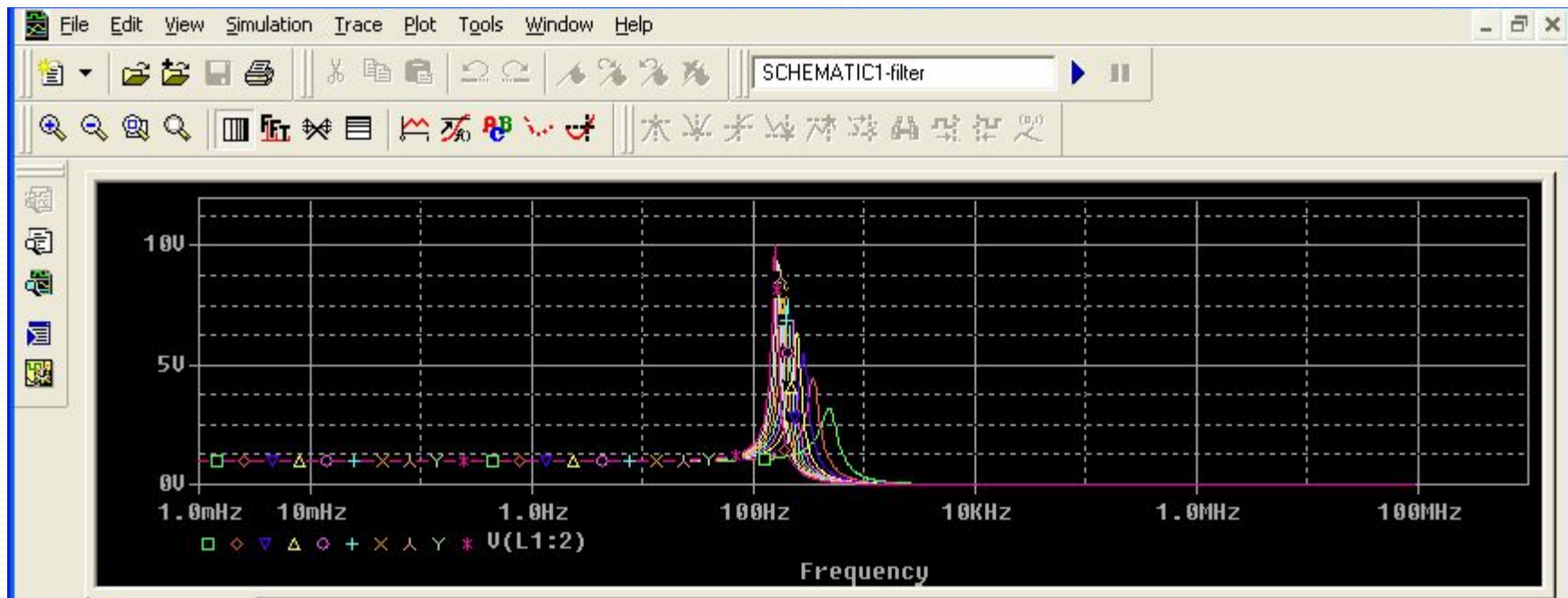
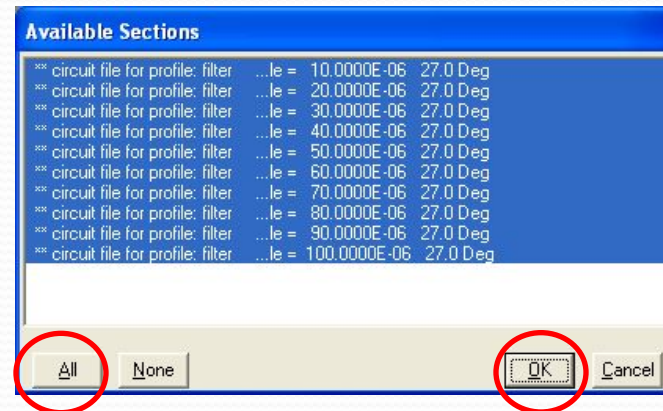


И запускаем схему на расчет

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

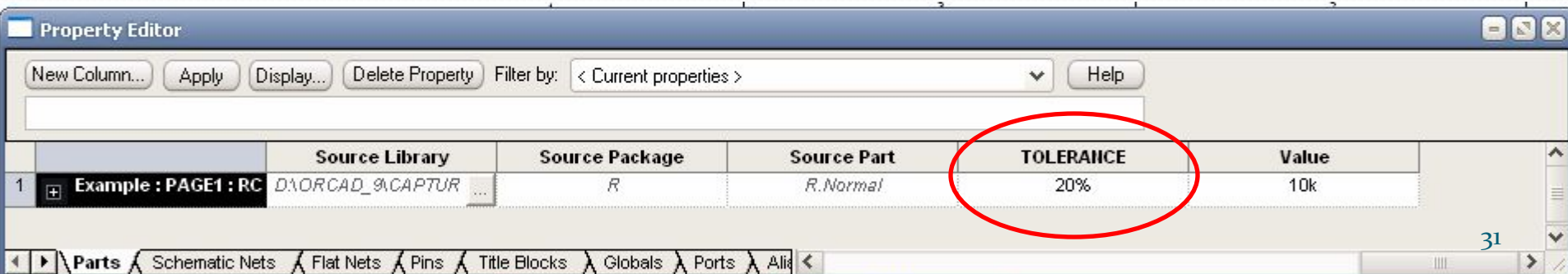
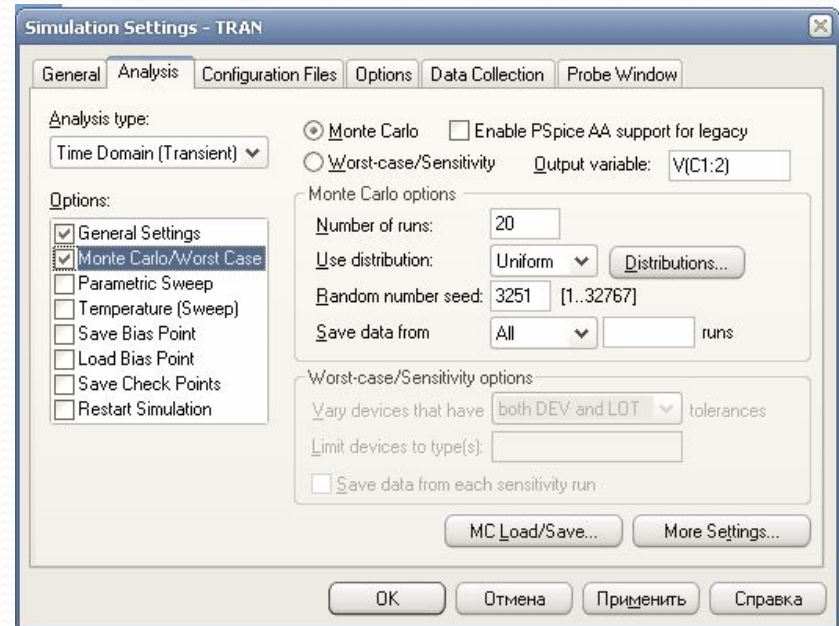
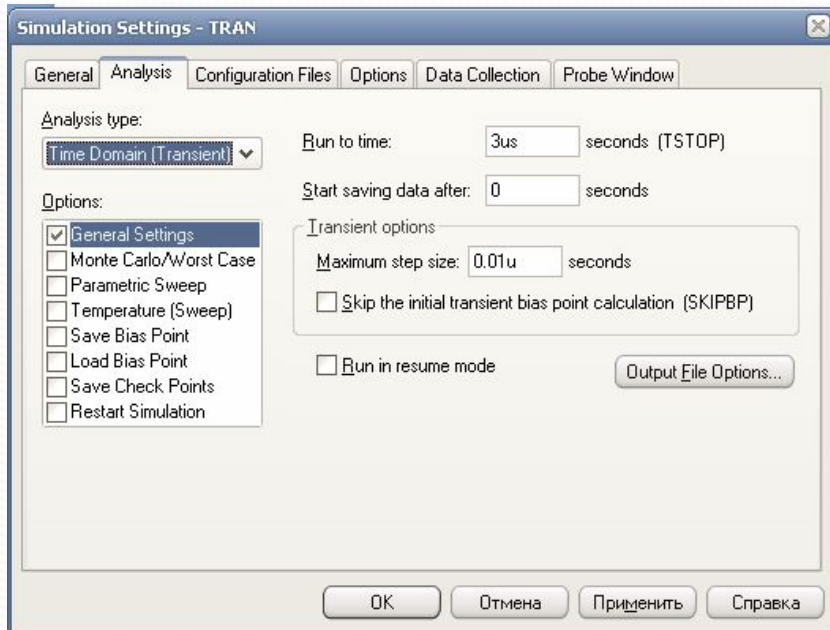
Виды моделирования

Статистическое моделирование методом Монте-Карло
(дифференциальный усилитель)

Настройки расчетов

во временной области

метода Монте-Карло

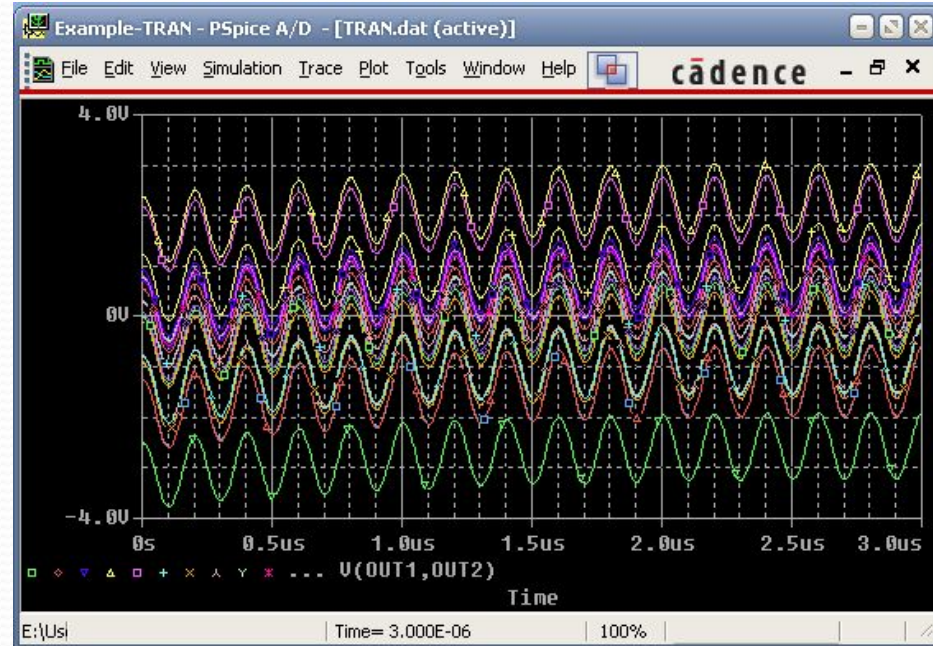
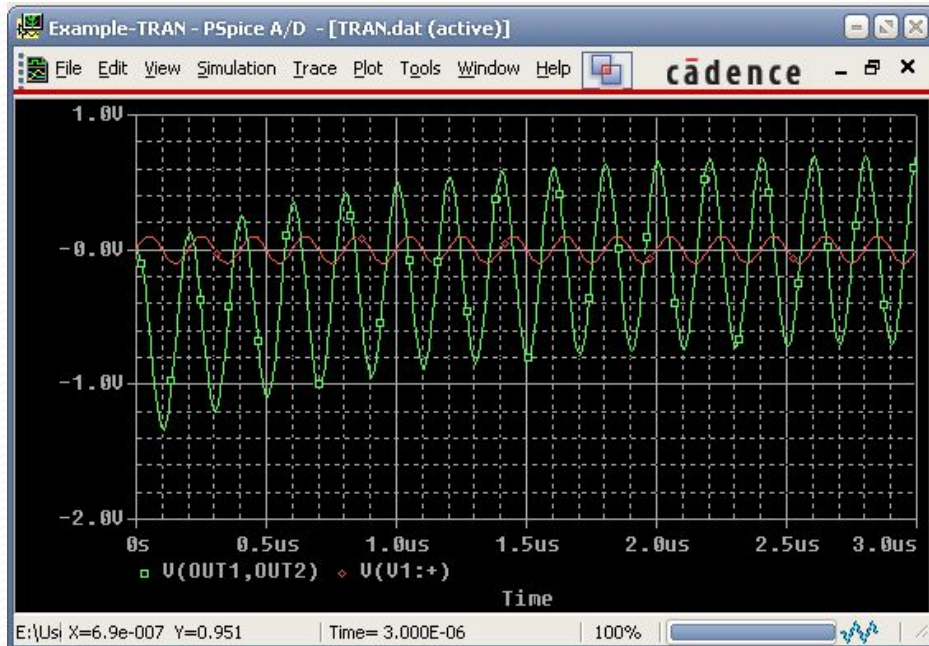


Виды моделирования

Статистическое моделирование методом Монте-Карло
(дифференциальный усилитель)

во временной области

Результаты расчетов
метода Монте-Карло

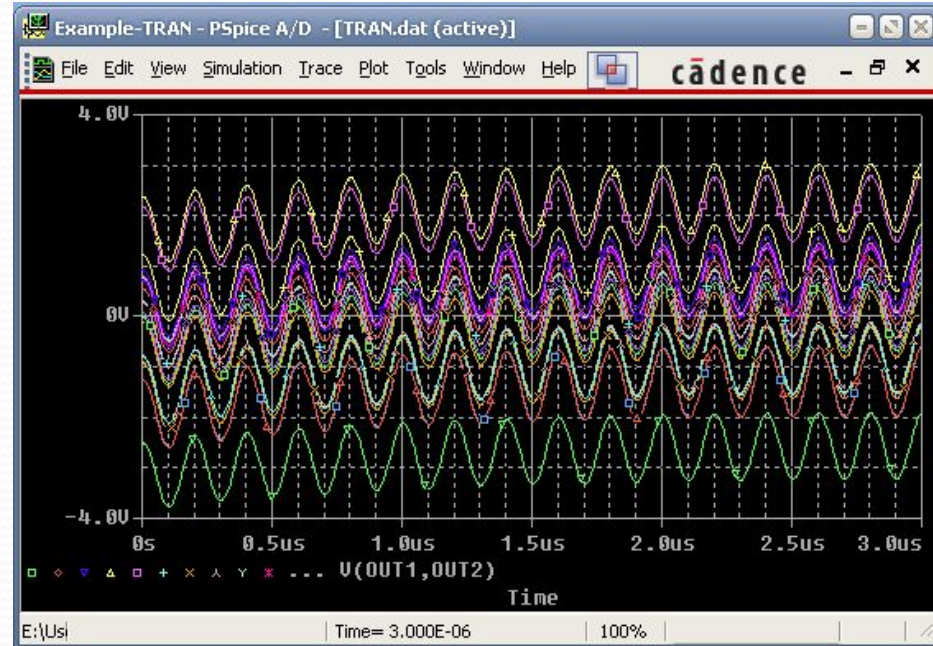
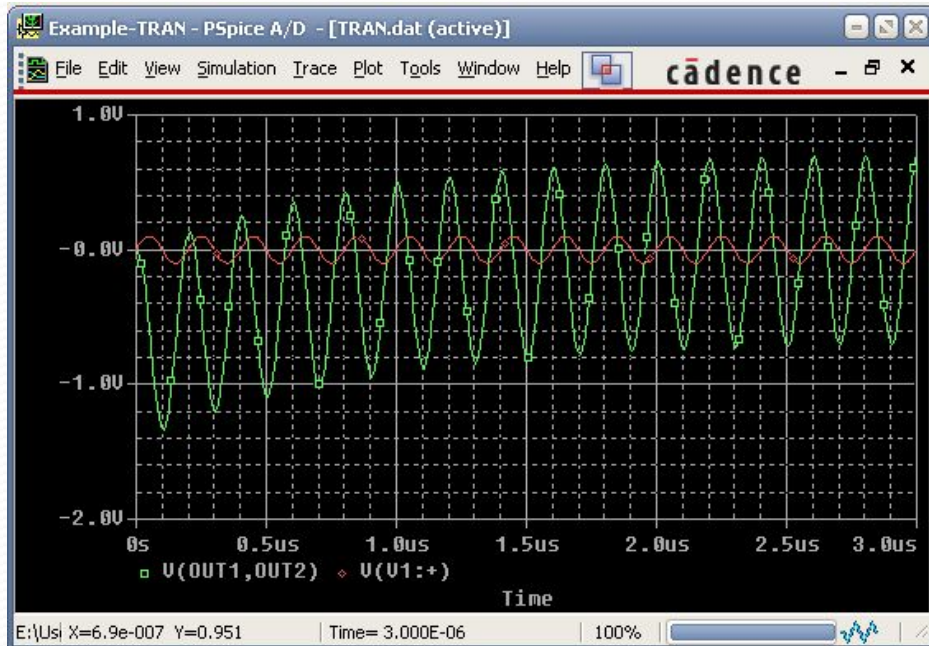


Виды моделирования

Статистическое моделирование методом Монте-Карло
(дифференциальный усилитель)

во временной области

Результаты расчетов
метода Монте-Карло

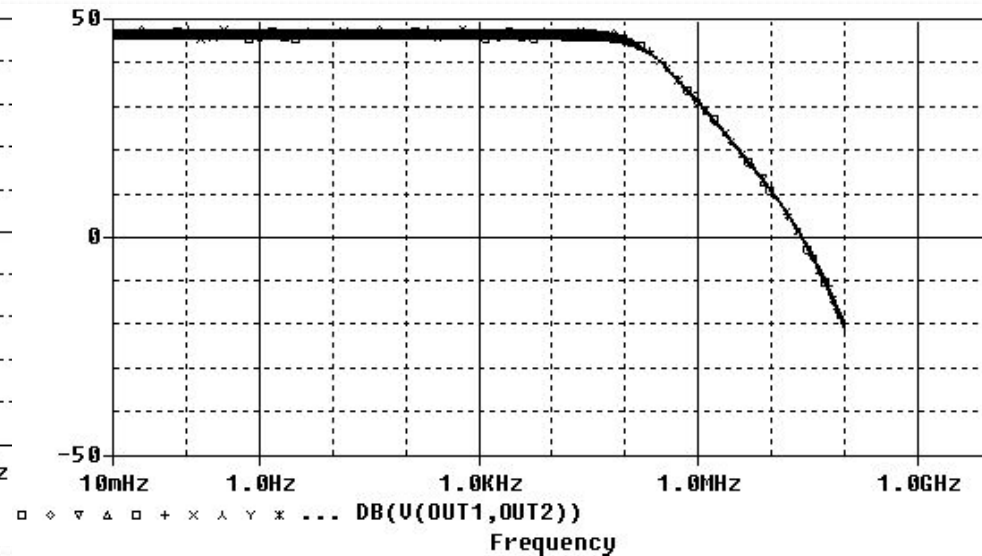
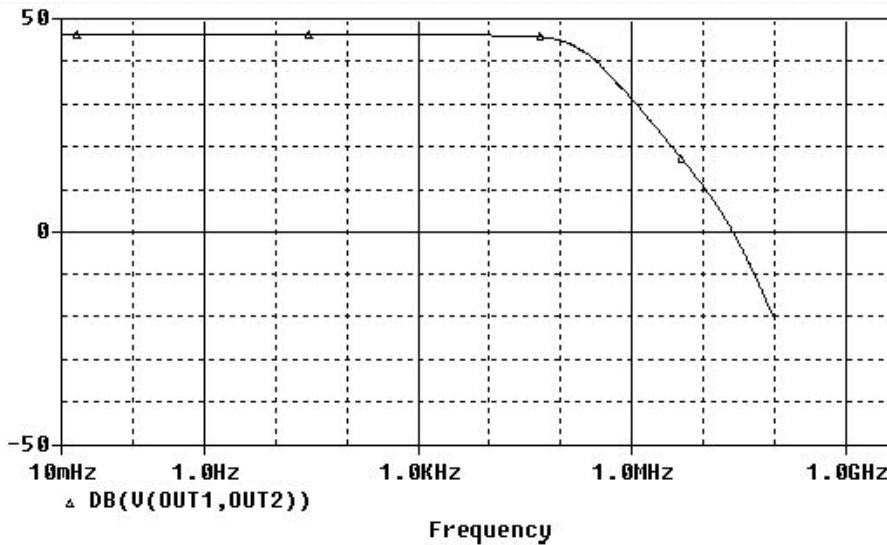


Виды моделирования

Статистическое моделирование методом Монте-Карло
(дифференциальный усилитель)

Результаты расчетов
метода Монте-Карло

В частотной области

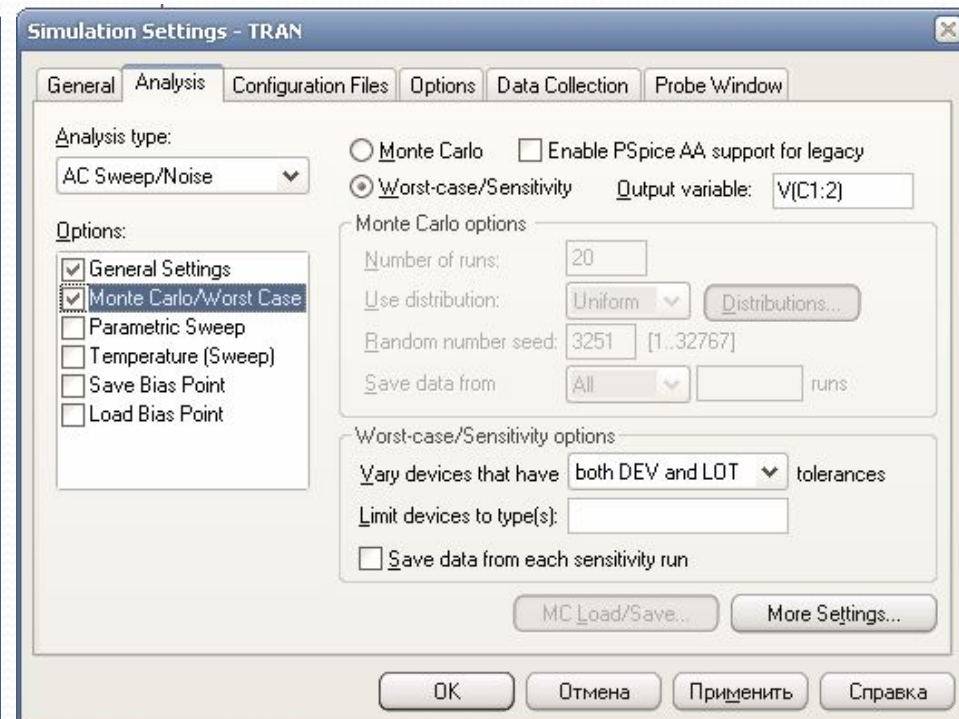
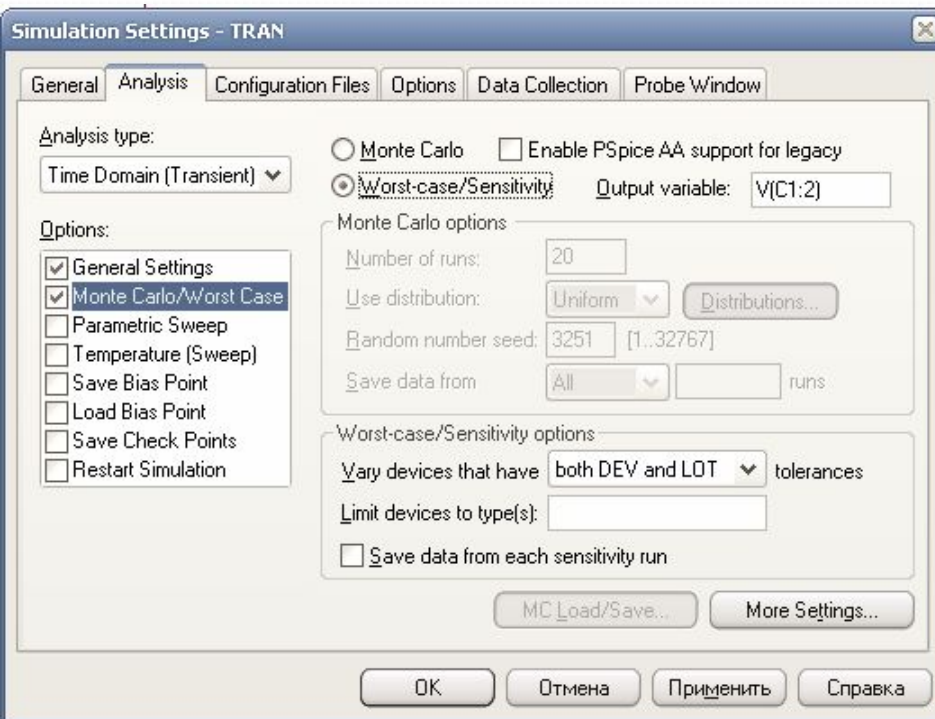


Виды моделирования

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

Настройки расчетов
во временной области

в частотной области

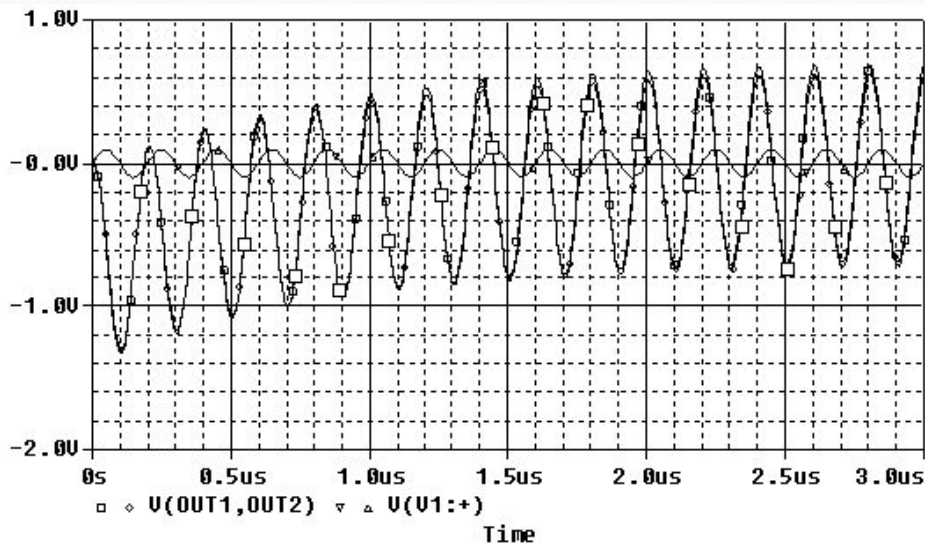


Виды моделирования

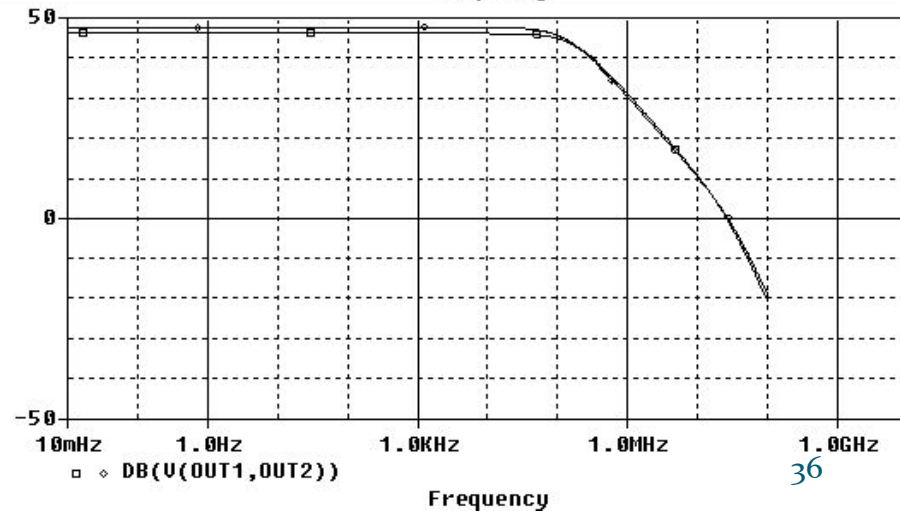
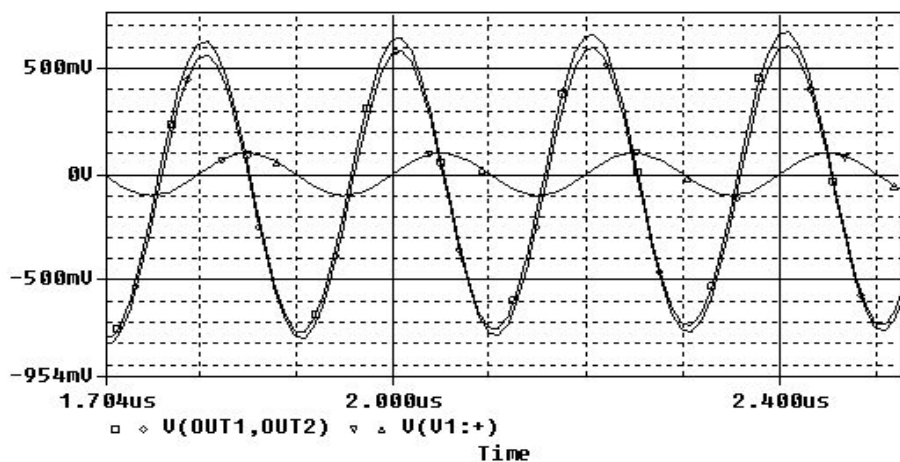
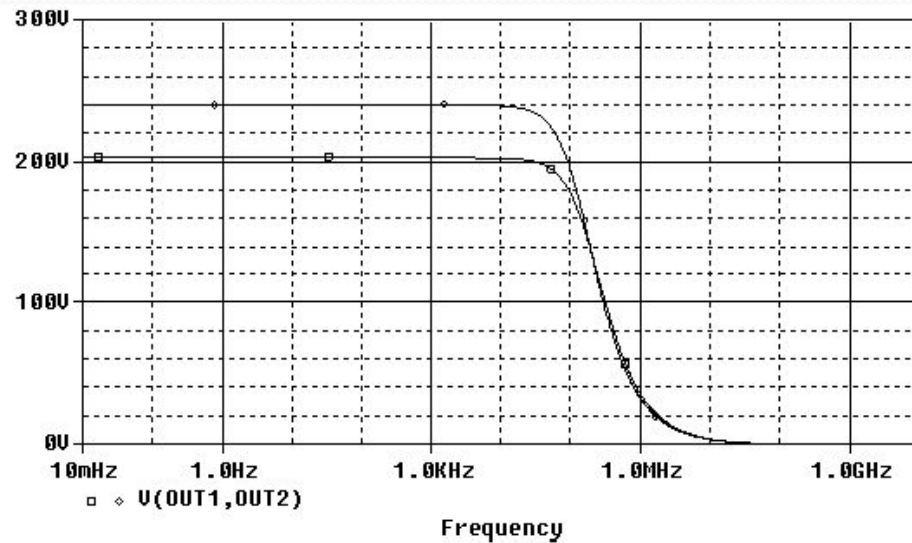
Статистическое моделирование методом. Расчет на наихудший случай
(дифференциальный усилитель)

Результаты расчетов

во временной области

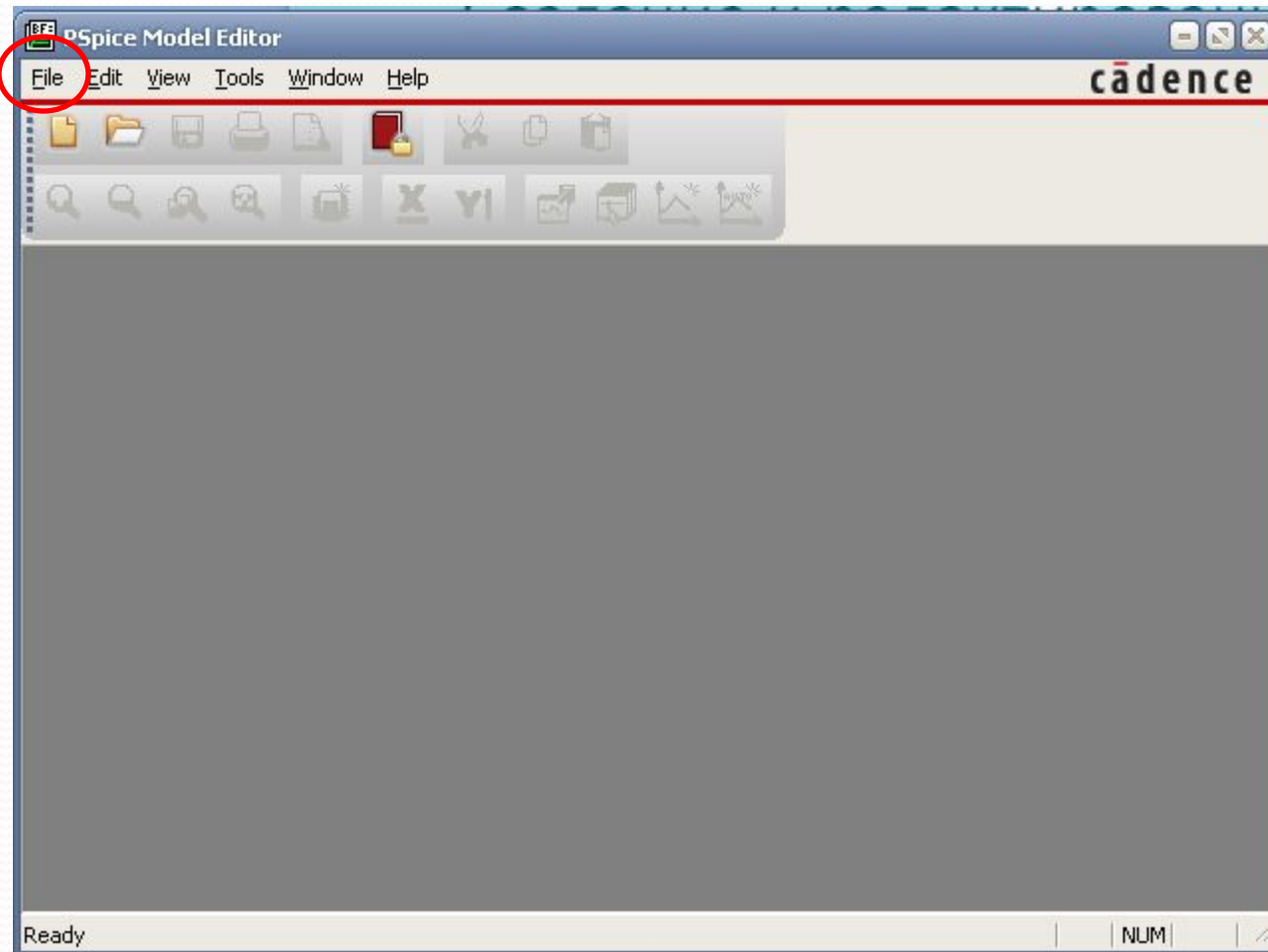


в частотной области (АЧХ, ЛАЧХ)



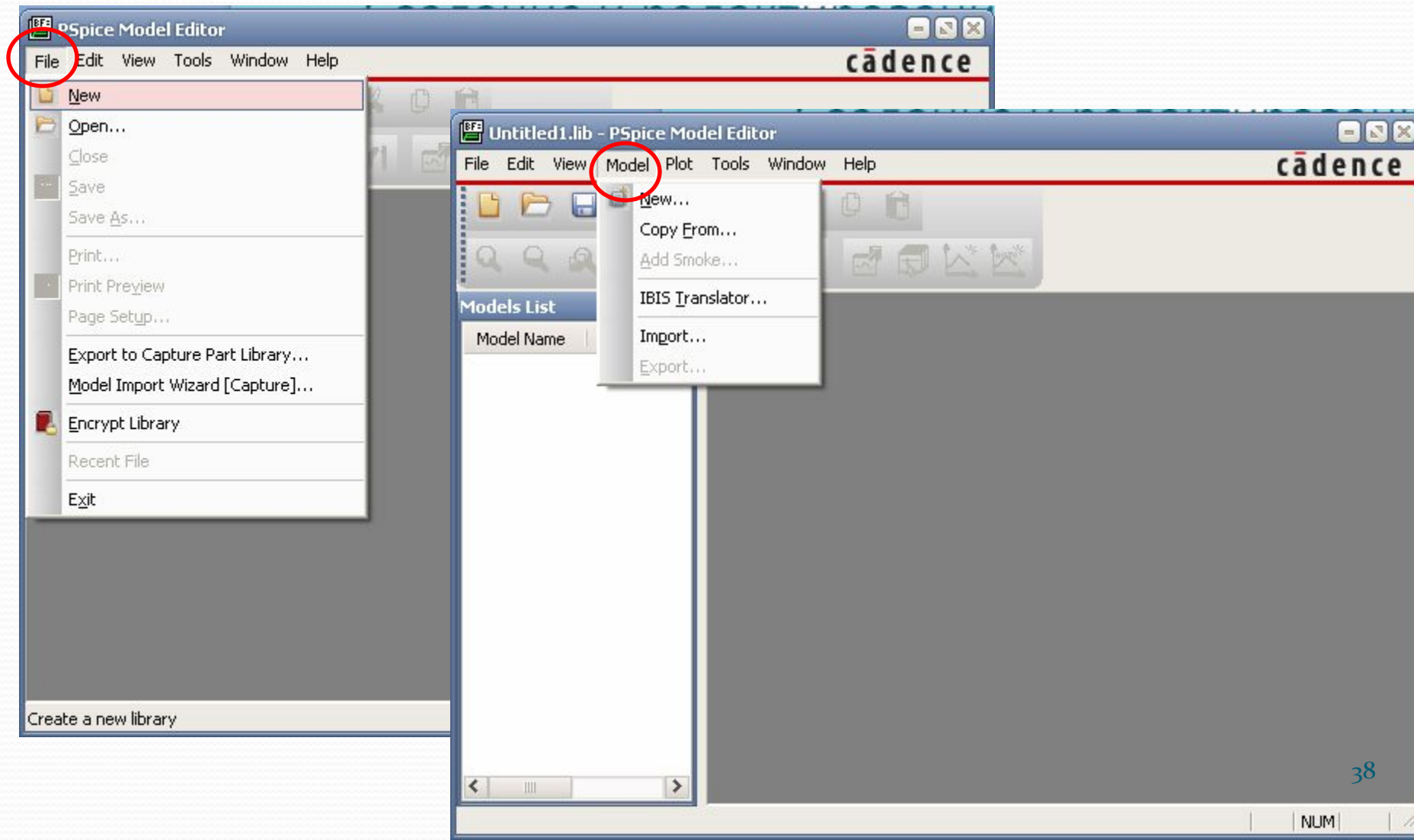
Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

Главное окно приложения

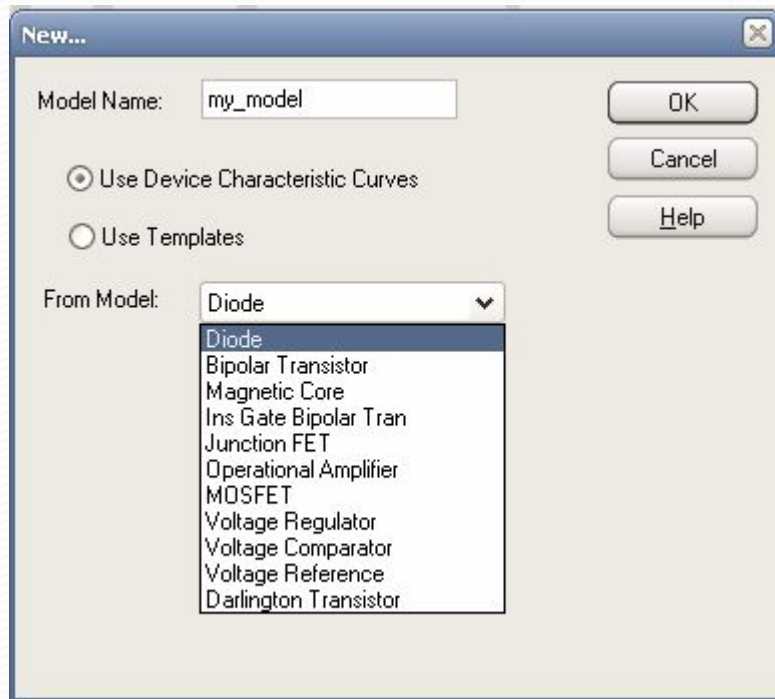


Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

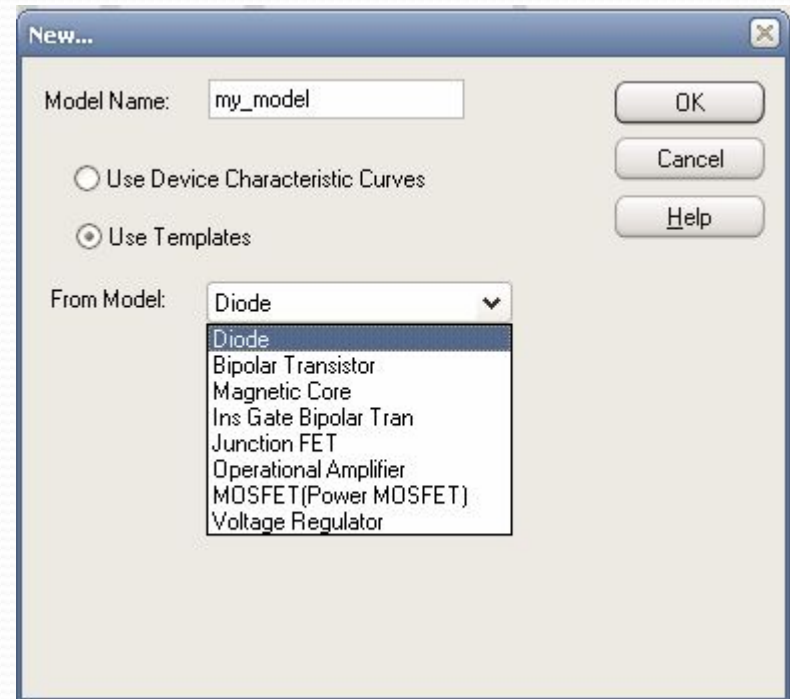
Главное окно приложения



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 'Forward Current' section is active, showing a table for data points and a graph of the forward current vs. forward voltage.

Models List

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

Forward Current

To include this spec in the model extraction please enter two or more data points in the following table:

#	Vfwd	I fwd
1		
2		
3		
4		
5		
6		
7		
8		

Graph: Forward Current vs. Forward Voltage

The graph shows the forward current (I fwd) in Amperes (A) versus the forward voltage (Forward Voltage) in Volts (V). The x-axis ranges from 0.4V to 1.2V, and the y-axis ranges from 0A to 3.00A. A green curve represents the forward current at 27°C.

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"/>

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) showing a reverse recovery characteristic. The plot has a y-axis from -20mA to 20mA and an x-axis from -5ns to 20ns. A green curve shows a step change in current from 0mA to -20mA at 0ns, followed by a reverse recovery transient. Below the plot are buttons for 'Forward Cur...', 'Junction Ca...', 'Reverse Le...', 'Reverse Br...', and 'Reverse Re...'. At the bottom is a 'Parameters' table with columns for Parameter Name, Value, Minimum, Maximum, Default, Active, and Fixed.

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

Reverse Recovery

Trr:

Ifwd:

Irev:

Rf:

20mA
0A
-20mA
-5ns 0s 10ns 20ns
Time
I Ifwd

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 displays the PSpice Model Editor interface for a diode model named 'my_model*'. The 'Reverse Breakdown' section is active, showing input fields for V_z , I_z , and Z_z , all set to 0. A graph plots Reverse Current (A) on the x-axis (0 to 1.00A) against Reverse Voltage (V) on the y-axis (100.00V to 100.30V). The curve shows a sharp increase in current at low voltages, leveling off as voltage increases. The graph is labeled 'Ur (27°C)'.

Models List

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

Reverse Breakdown

V_z : 0
 I_z : 0
 Z_z : 0

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/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 'Недавние документы', 'Рабочий стол', 'Мои документы', 'Мой компьютер', and 'Сетевое'. The 'Имя файла' (File name) is 'my_model.lib' and the 'Тип файла' (File type) is 'Model Library Files (*.lib)'. The 'Сохранить' (Save) and 'Отмена' (Cancel) buttons are visible.

File menu options:

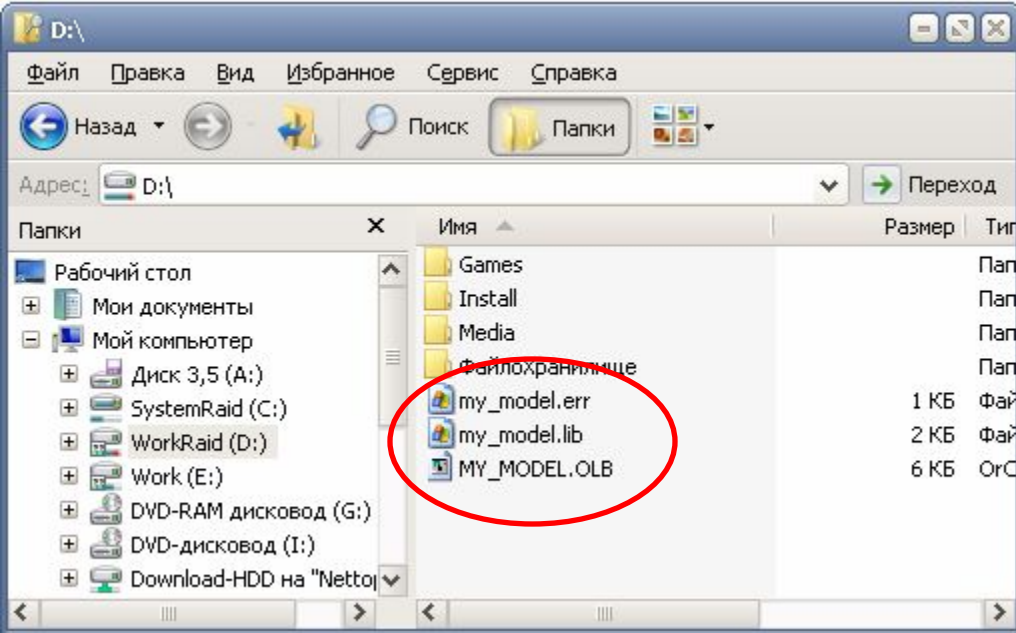
- New
- Open...
- Close
- Save
- Save As...
- Print...
- Print Preview
- Page Setup...
- Export to Capture Part Library...
- Model Import Wizard [Capture]...
- Encrypt Library
- ⌞ D:\my_model.lib
- Exit

Parameters table:

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



```
Lister - [D:\my_model.err]
Файл  Правка  Вид  Кодировка  Справка
STATUS: PSpice Schematics to Capture translator (16.2.0.p001)
STATUS:
STATUS: Translator started at Friday, November 26, 2010 10:45:50
STATUS: C:\ORCAD\ORCAD_16.2\tools\Capture\sch2cap -f "D:\my_model.lib
"D:\my_model.olb" -i C:\ORCAD\ORCAD_16.2\tools\PSpice\PSpice.ini -s m
INFO: Using existing library 'C:\ORCAD\ORCAD_16.2\tools\Capture\Li
\modeled.etc'.
INFO: Created new library 'D:\my_model.olb'.
STATUS: Translator stopped at Friday, November 26, 2010 10:45:50
STATUS: 0 Error messages, 0 Warning messages
```

```
Lister - [D:\my_model.lib]
Файл  Правка  Вид  Кодировка  Справка
100 %
* PSpice Model Editor - Version 16.2.0

*$
*BeginSpec
*IF:
*JC:
*RL:
*RB: Uz=0 Iz=0 Zz=0
*RR: Trr=0 Ifwd=10.000E-3 Irev=10.000E-3 R1=100
*EndSpec

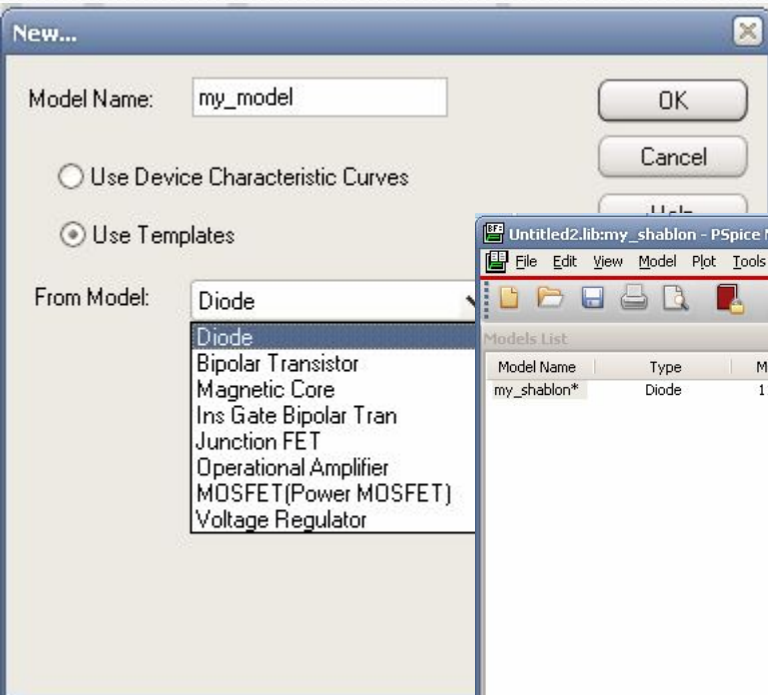
*BeginTrace
*IF: 1,0,.4,1.2000,1,3,0,0,-1 (27)
*JC: 0,1,-.1,10,1,3,0,0,-1 (27)
*RL: 0,0,1,100,1,3,0,0,-1 (27)
*RB: 0,1,100.00E-6,1,1,3,0,0,-1 (27)
*RR: 0,0,-5.0000E-9,20.000E-9,1,3,0,0,-1 (27)
*EndTrace

*BeginParam
*IS=10.000E-15 (10.000E-21,-.1,0)
*N=1 (.2,5,0)
*RS=1.0000E-3 (1.0000E-6,100,0)
*IKF=0 (0,1.0000E3,0)
*XTI=3 (-100,100,0)
*EG=1.1100 (.1,5.5100,0)
*CJO=1.0000E-12 (10.000E-21,1.0000E-3,0)
*M=.3333 (.1,10,0)
*UJ=.75 (.3905,10,0)
*FC=.5 (1.0000E-3,10,0)
*ISR=100.00E-12 (10.000E-21,-.1,0)
*NR=2 (.5,5,0)
*BU=100 (.1,1.0000E6,0)
*IBU=100.00E-6 (1.0000E-9,10,0)
*TT=5.0000E-9 (100.00E-18,1.0000E-3,0)
*EndParam

*DEVICE=my_model,D

* my_model D model
* created using Model Editor release 16.2.0 on 11/26/10 at 10:44
* The Model Editor is a PSpice product.
.MODEL my_model D
+ RS=1.0000E-3
+ CJO=1.0000E-12
+ M=.3333
+ UJ=.75
+ ISR=100.00E-12
+ BU=100
+ IBU=100.00E-6
+ TT=5.0000E-9
*$
```

Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor



Simulation Parameters

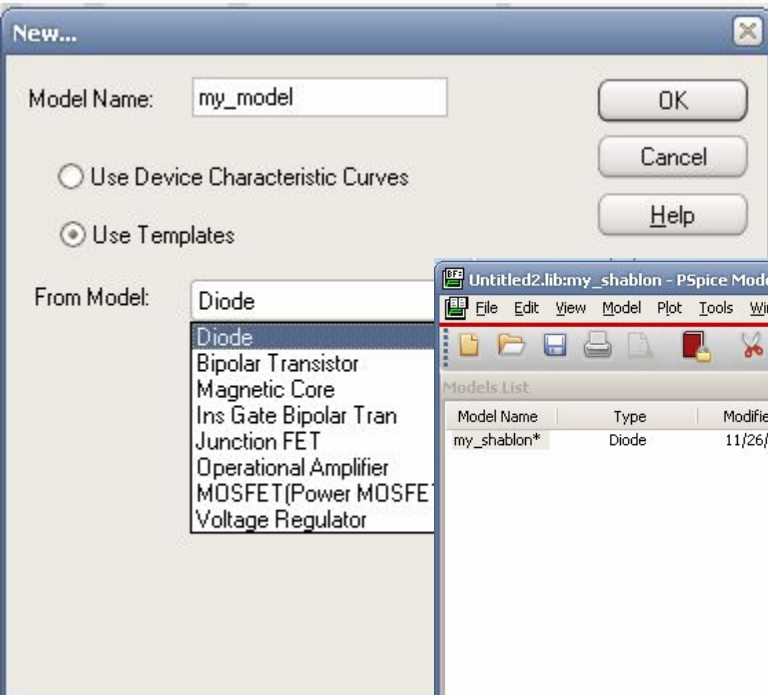
Property Name	Description	Value	Default	Unit	Distribution	Postol	Illegal	Editable
IS	Saturation current	10f	10f	A				<input type="checkbox"/>
RS	Ohmic resistance	0	0	Ohm				<input type="checkbox"/>
N	Emission coefficient	1	1					<input type="checkbox"/>
TT	Transit time	0	0	sec				<input type="checkbox"/>
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



Untitled2.lib:my_shablon - PSpice Model Editor - [Smoke Parameters]

File Edit View Model Plot Tools Window Help

cadence

Models List

Model Name	Type	Modified
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

Simulation Smoke

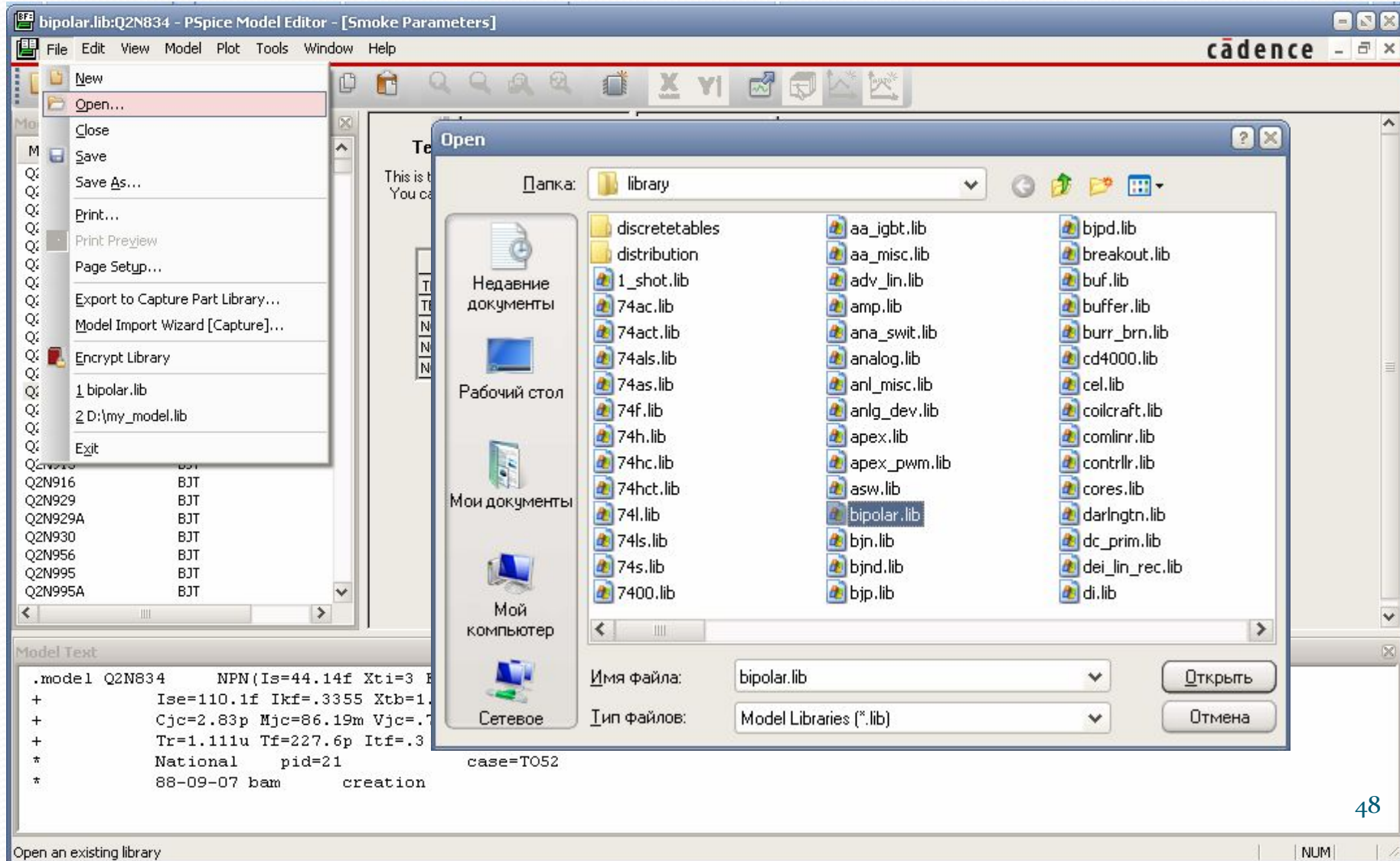
Model Text (Read Only)

```
* 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
```

Ready

2

Создание и редактирование моделей компонентов ЭС в PSpice Model Editor



Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

The screenshot shows 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 panes:

- Models List:** A list of BJT models, with Q2N2222 selected.
- Test Node Mapping:** A section for defining node and port mappings, containing a table:

Node	Port
TERM_IC	C
TERM_IB	B
NODE_VC	C
NODE_VB	B
NODE_VE	E

- Smoke Parameters:** A section for defining device maximum operating condition parameters, containing a table:

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		%

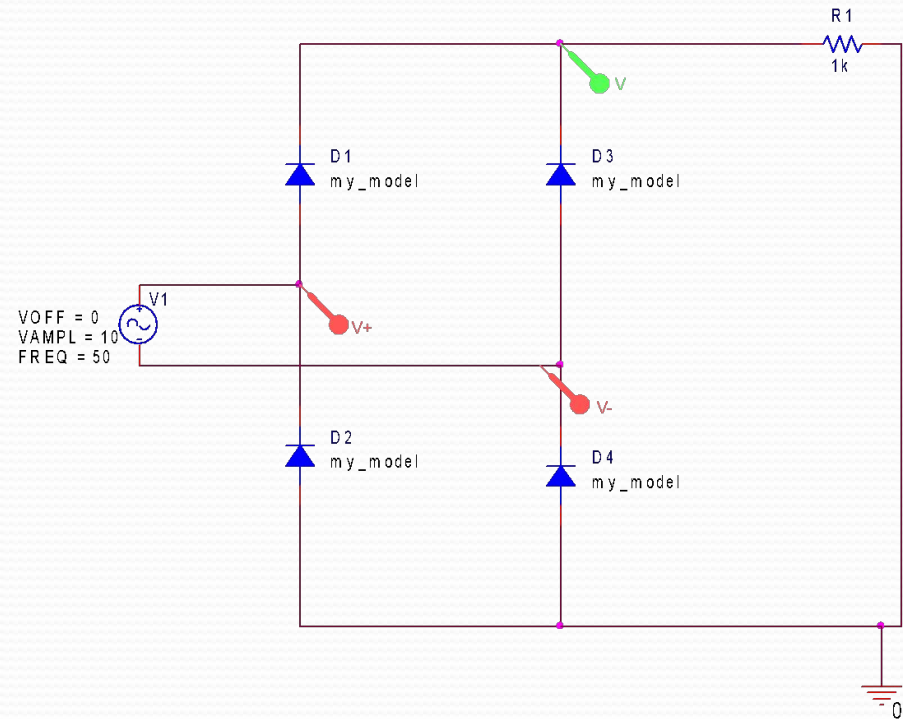
At the bottom, the **Model Text** pane shows the model definition code:

```
.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 (parameters: 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	A
Document Number	<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

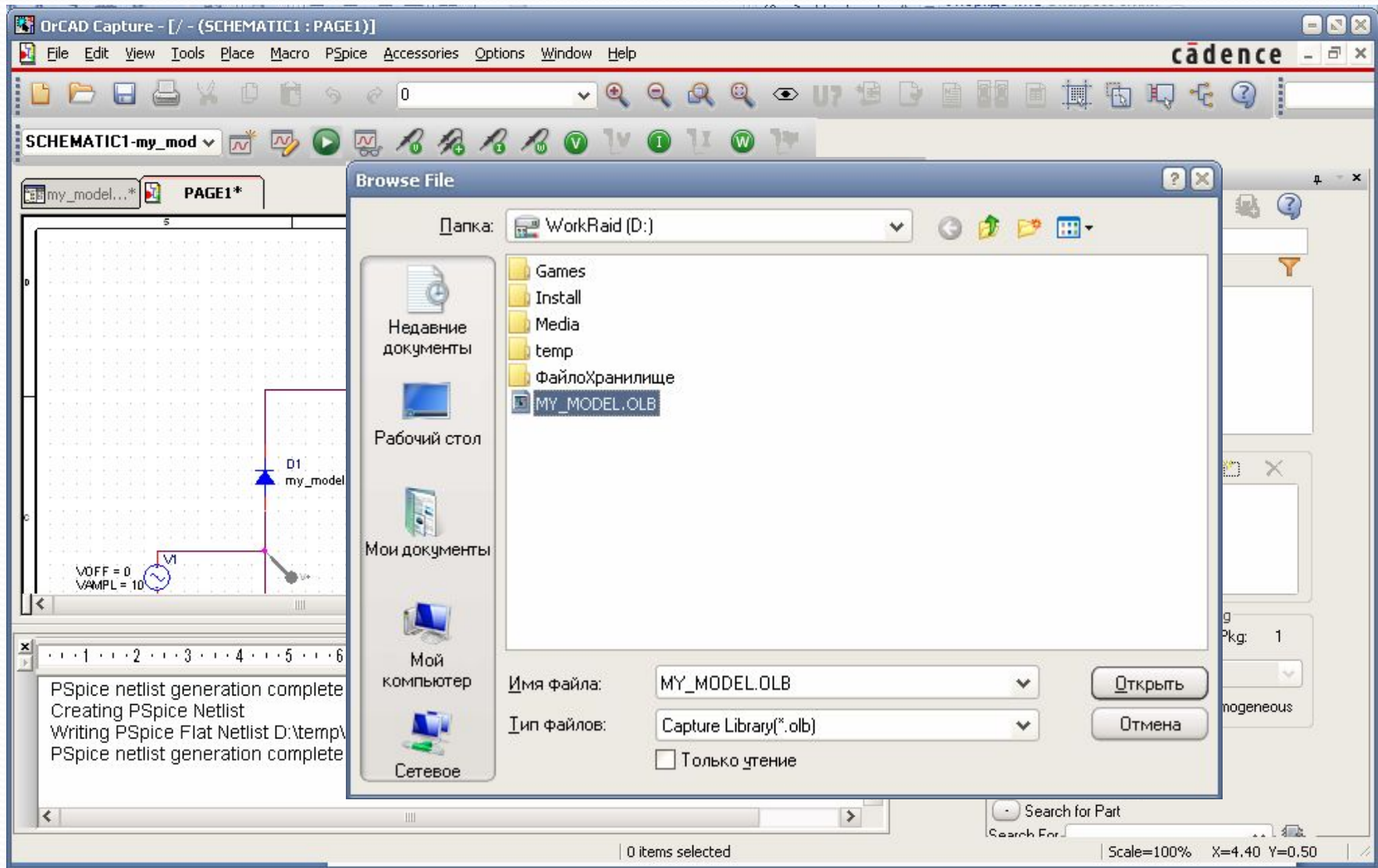
The screenshot displays the OrCAD Capture interface. The main workspace shows a circuit diagram with a voltage source V1 (V_{OFF} = 0, V_{AMPL} = 10), two diodes D1 and D3 (both labeled 'my_model'), and a resistor R1 (1k). The status bar at the bottom indicates '0 items selected'.

The 'Place Part' dialog box is open on the right side. It features a search bar, a 'Part List' section with the text 'Добавить библиотеку' (Add library), and a 'Libraries' list containing 'ANALOG', 'Design Cache', 'DIG_PRIM', and 'SOURCE'. A red circle highlights a small icon in the 'Libraries' section. Below the libraries, there are fields for 'Packaging', 'Parts per Pkg:' (set to 1), 'Part:' (dropdown), and 'Type: Homogeneous'. At the bottom of the dialog, there are radio buttons for 'Normal' and 'Convert', and a 'Search for Part' button.

The bottom status bar shows 'Scale=100% X=4.40 Y=0.50'.

```
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 $VOFF = 0$ and $VAMPL = 10$. Two diodes, $D1$ and $D3$, are placed and 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`. The `Normal` radio button is selected, and the `Convert` radio button is unselected. The `Search for Part` field is empty. The `Scale=100%` and `X=0 Y=0` are displayed at the bottom right of the dialog.

The status bar at the bottom of the window indicates `0 items selected`.

The command 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

The image shows a screenshot of the PSpice Model Editor interface. The main window displays a circuit diagram with four diode components labeled D1, D2, D3, and D4, all using the model '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, and the 'Browse...' button is highlighted with a red circle. The 'Configured Files' list contains 'nom.lib'. The 'Library Path' is set to 'C:\OrCAD\OrCAD_16.2\tools\PSpice\UserLib'. The dialog box has buttons for 'Add as Global', 'Add to Design', 'Add to Profile', 'Edit', 'Change', and 'Browse...'. The status bar at the bottom shows 'Ready' and the page number '55'.

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

D1 my_model

D2 my_model

D3 my_model

D4 my_model

V1

VOFF = 0
VAMPL = 10
FREQ = 50

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:\

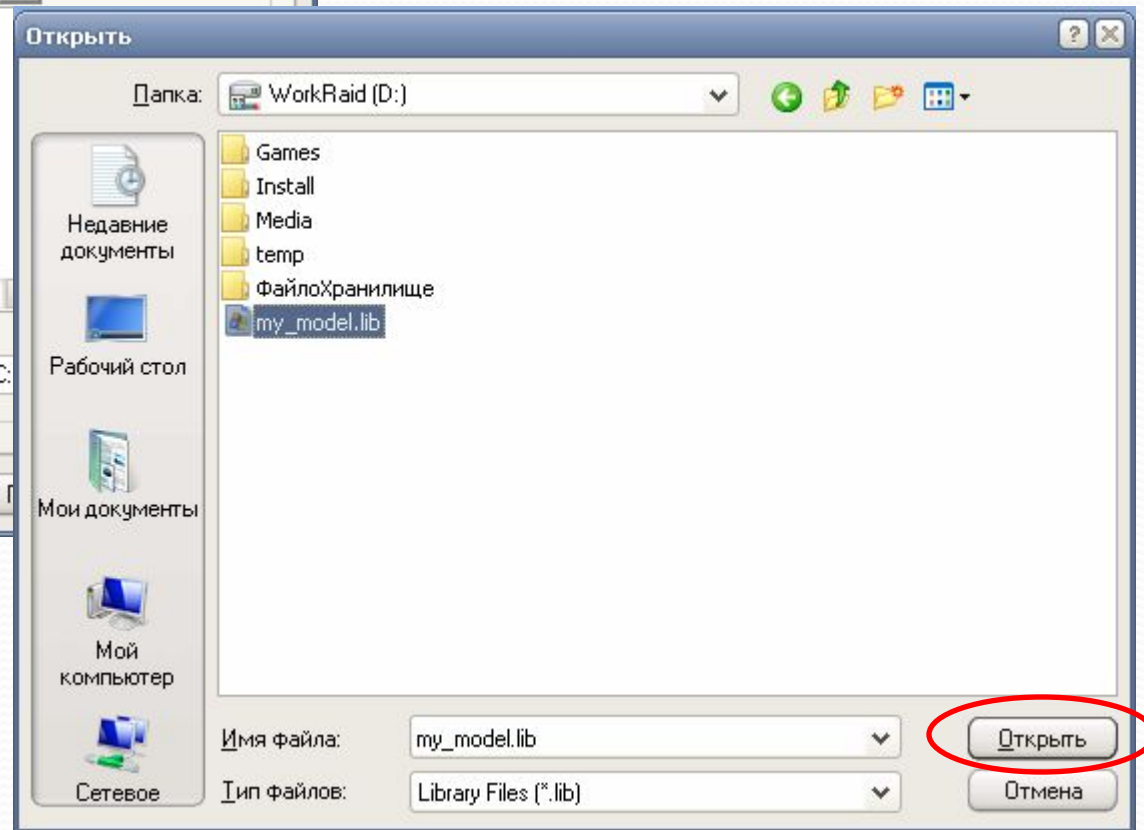
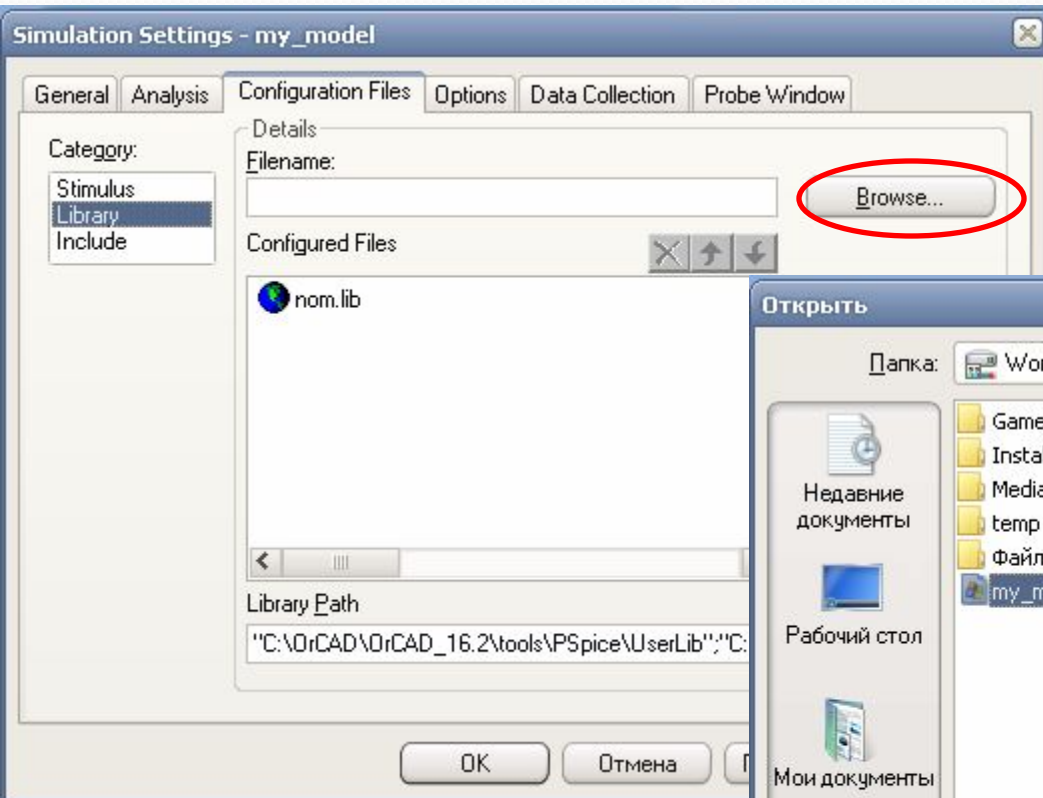
Browse...

OK Отмена Применить Справка

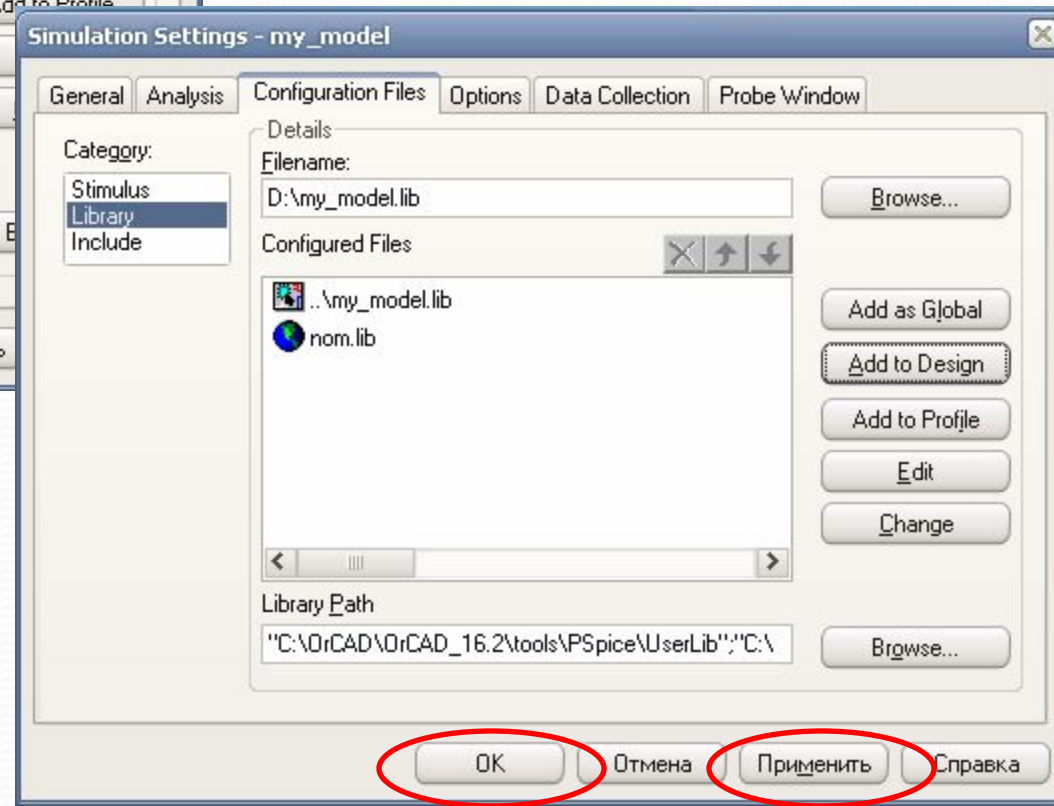
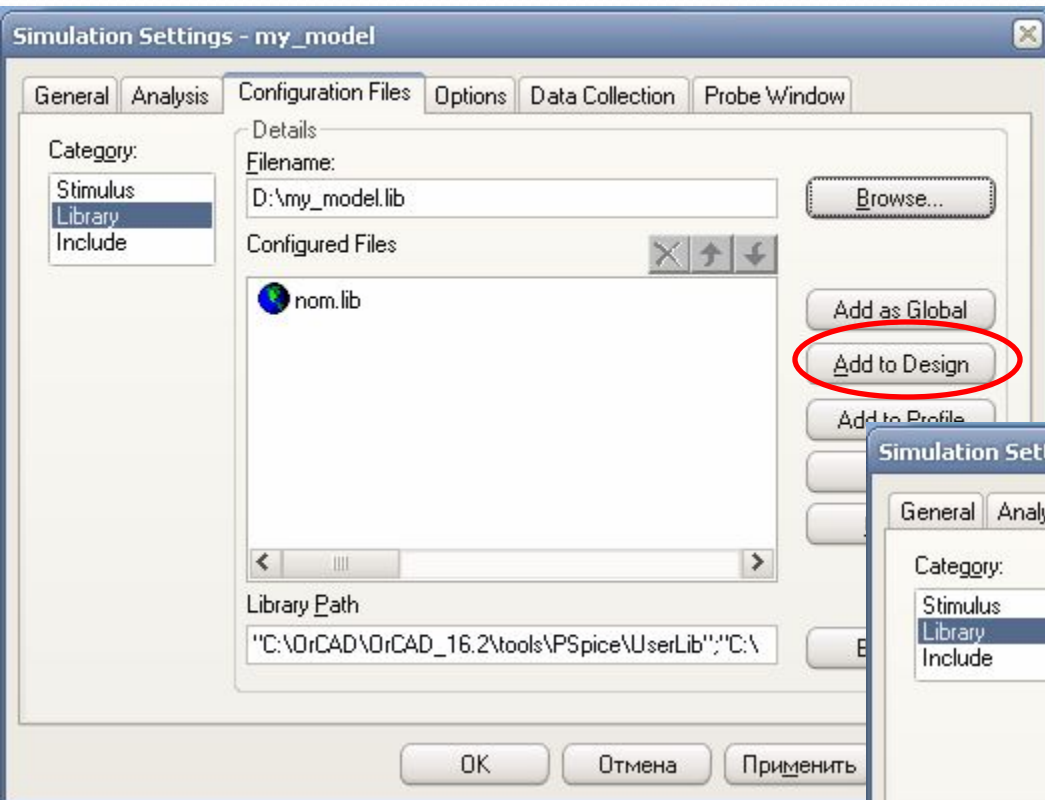
Ready

55

Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

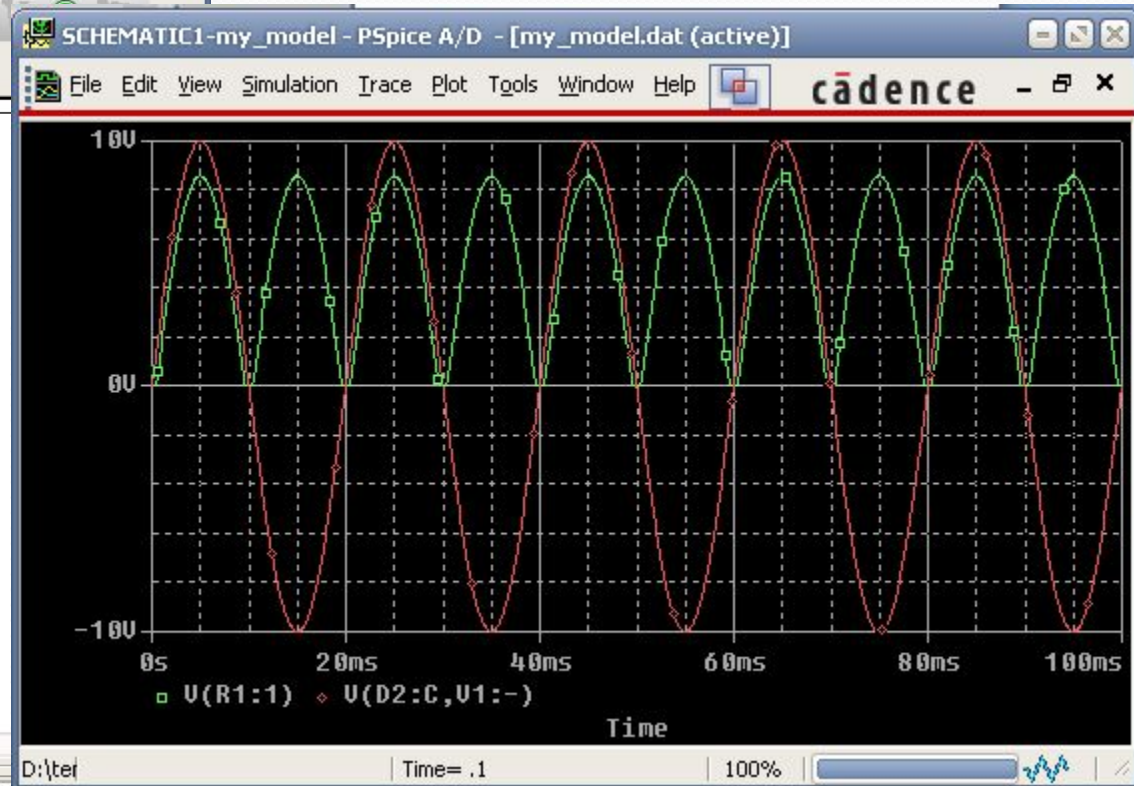
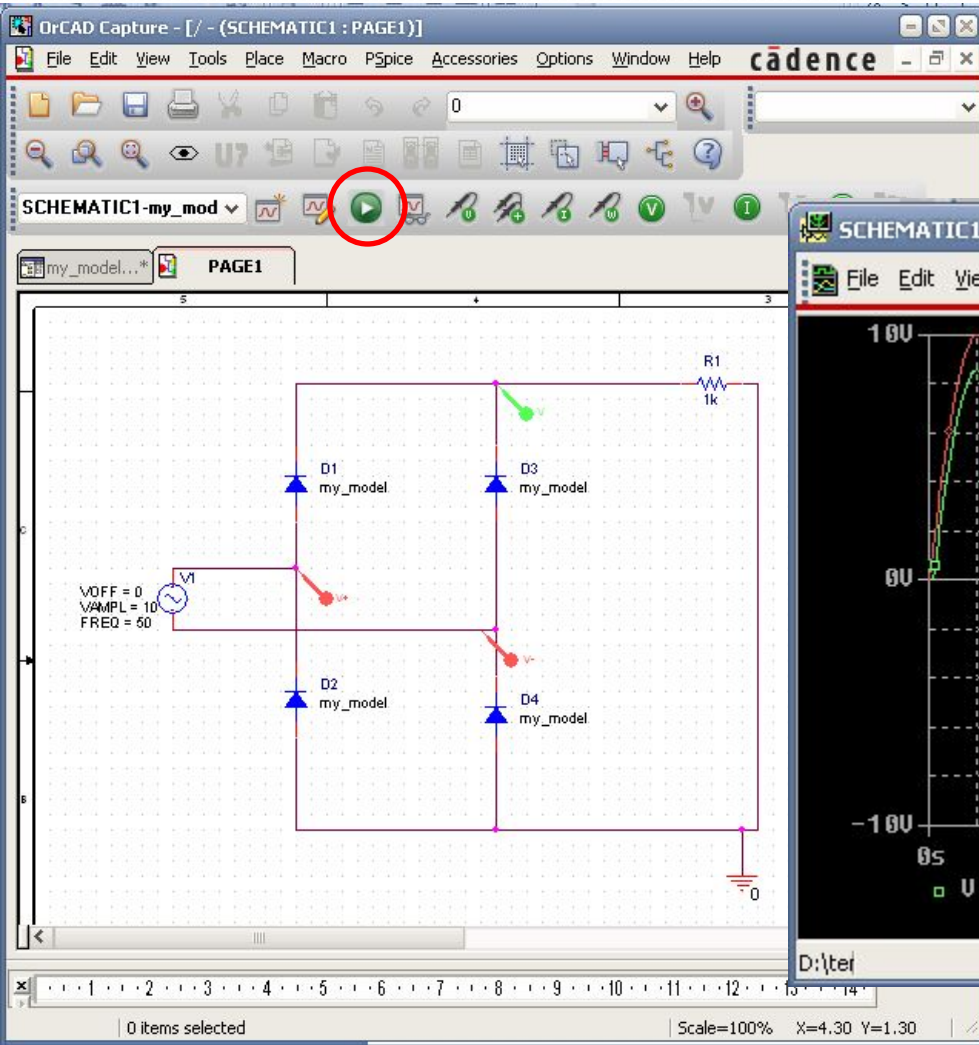



Создание и редактирование моделей КОМПОНЕНТОВ ЭС в PSpice Model Editor



Создание и редактирование моделей компонентов ЭС в PSpice Model Editor

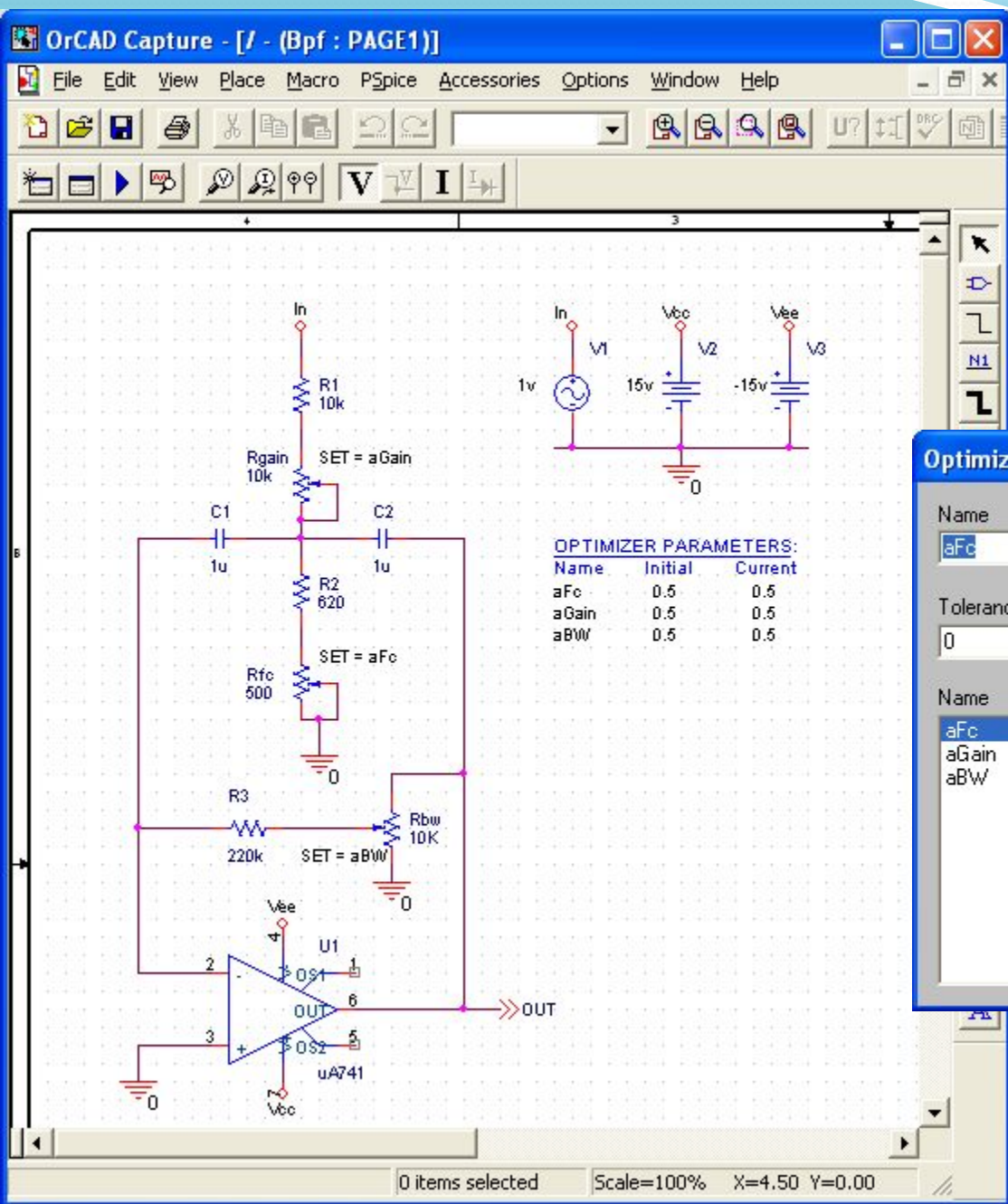
Проверка работоспособности созданной модели





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

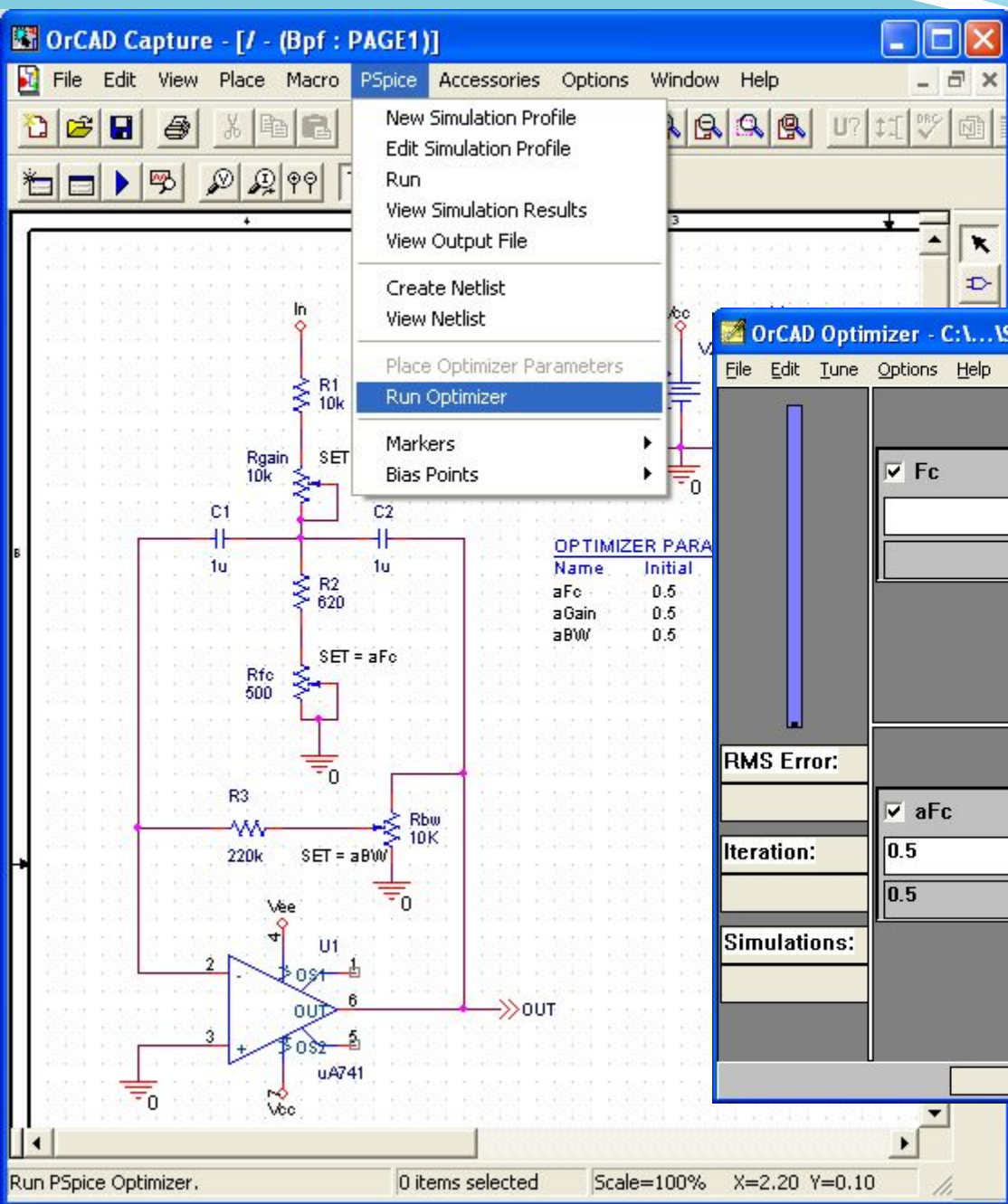
Add

Change

Delete

OK

Cancel



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

File Edit Tune Options Help

Specifications

Fc BW Gain

Parameters

aFc aGain aBW

Iteration: 0.5 0.5 0.5

0.5 0.5 0.5

RMS Error:

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:\... \SAMPLESOPTIMIZE\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:\... \SAMPLESOPTIMIZE\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

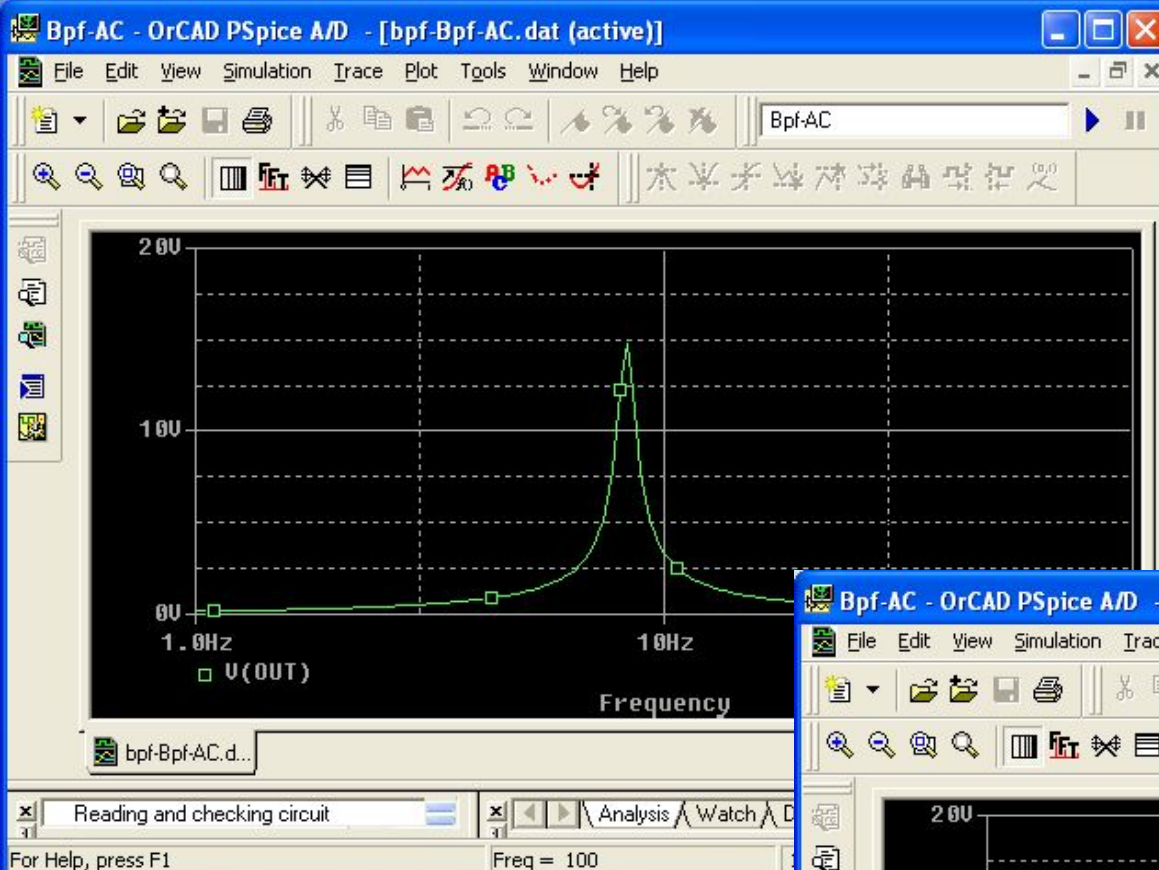
Parameters

<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

Iteration: 3

Simulations: 9

Schematic updated.



До оптимизации

После
оптимизации

