

Лабораторная работа №1

Знакомство с OrCad Capture

New Project ✕

Name

Lab1

OK

Cancel

Help

Create a New Project Using

-  Analog or Mixed A/D
-  PC Board Wizard
-  Programmable Logic Wizard
-  Schematic

Tip for New Users

Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.

Location

C:\LABS\Lab1

Browse...

Create PSpice Project



Create based upon an existing project

AnalogGNDsymbol.opj



Create a blank project

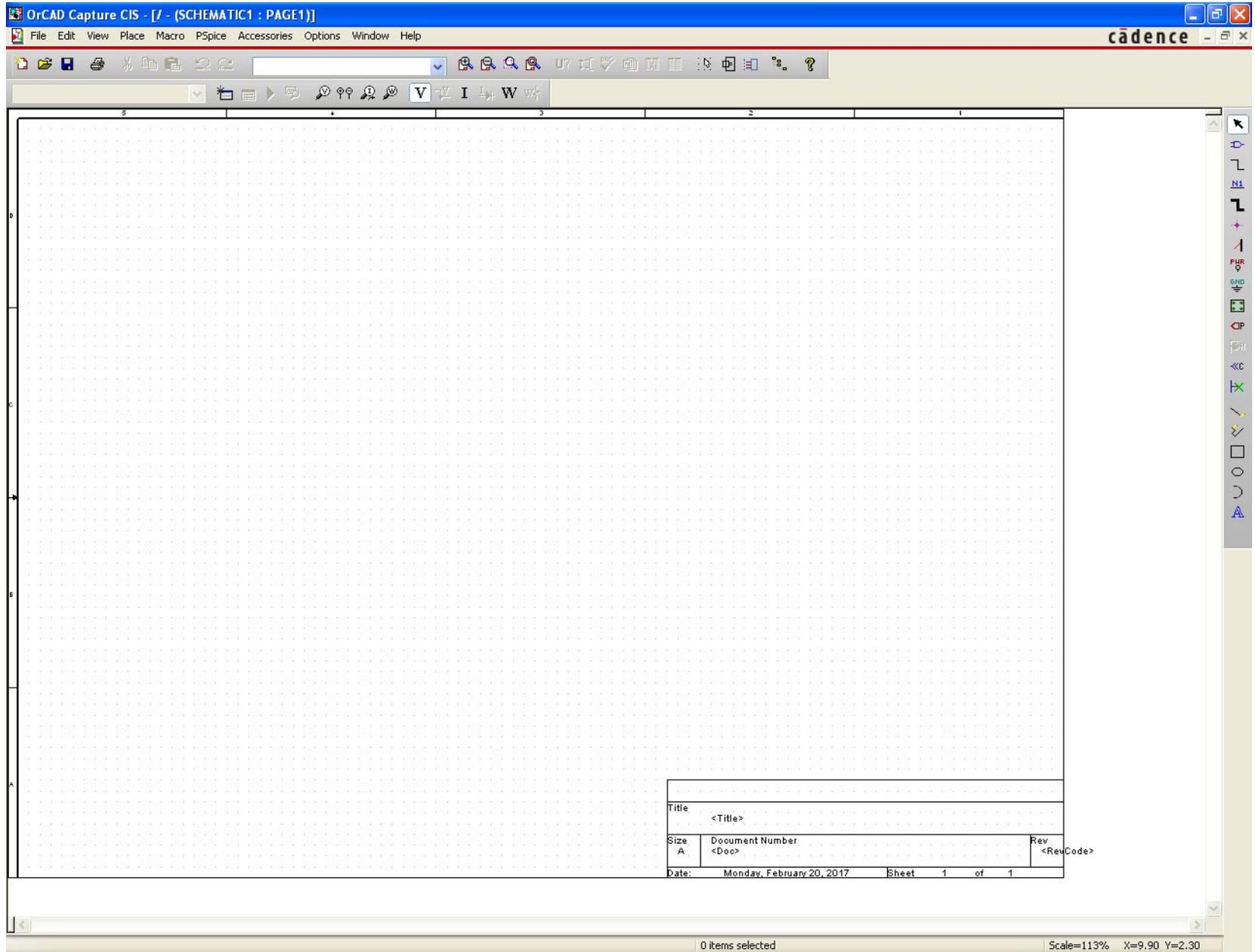
OK

Browse...

Cancel

Help

Рабочее окно Capture



Создание режима МОДЕЛИРОВАНИЯ

New Simulation

Name:
Lab1

Inherit From:
none

Root Schematic: SCHEMATIC1

Create

Cancel

Simulation Settings - Lab1

General Analysis Configuration Files Options Data Collection Probe Window

Analysis type:
Time Domain (Transient)

Options:

- General Settings
- Monte Carlo/Worst Case
- Parametric Sweep
- Temperature (Sweep)
- Save Bias Point
- Load Bias Point
- Save Check Points
- Restart Simulation

Run to time: 40ms seconds (TSTOP)

Start saving data after: 0 seconds

Transient options

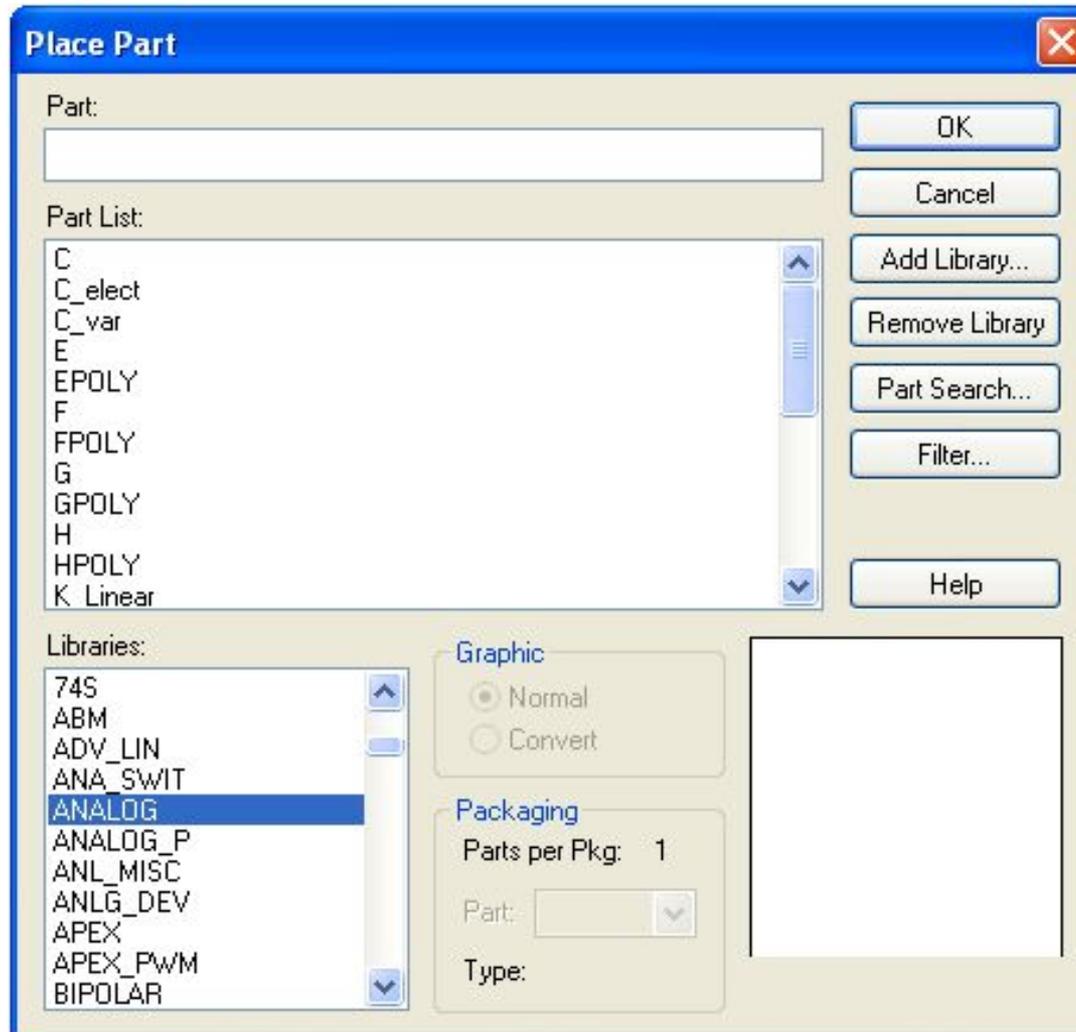
- Skip the initial transient bias point calculation (SKIPBP)
- Run in resume mode

Maximum step size: 0.1ms seconds

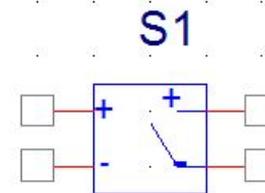
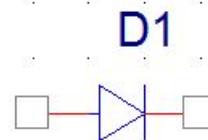
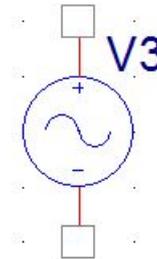
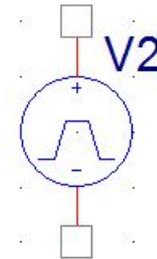
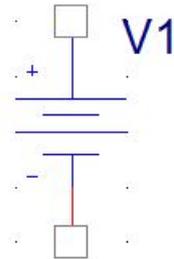
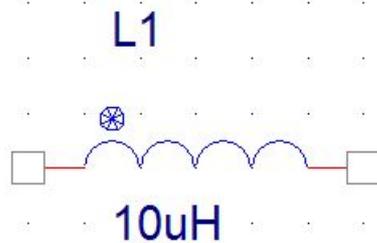
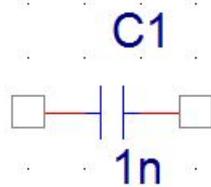
Output File Options...

OK Отмена Применить Справка

Добавление библиотек ЭЛЕМЕНТОВ



Основные электротехнические элементы



Изменение параметров элементов

	A
	SCHEMATIC1 : PAGE1
Color	Default
Designator	
DIST	FLAT
Graphic	R.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	<none>
Location X-Coordinate	270
Location Y-Coordinate	220
MAX_TEMP	RTMAX
Name	INS39
Part Reference	R1
PCB Footprint	AX/RC05
POWER	RMAX
Power Pins Visible	<input type="checkbox"/>
Primitive	DEFAULT
PSpiceTemplate	R*@REFDES %1 %2 ?TOL
Reference	R1
SLOPE	RSMAX
Source Library	C:\ICADENCE\SPB_16...
Source Package	R
Source Part	R.Normal
TC1	0
TC2	0
TOLERANCE	
Value	1k
VOLTAGE	RVMAX

Display Properties

Name: Value

Value:

Font: Arial 5 (default)

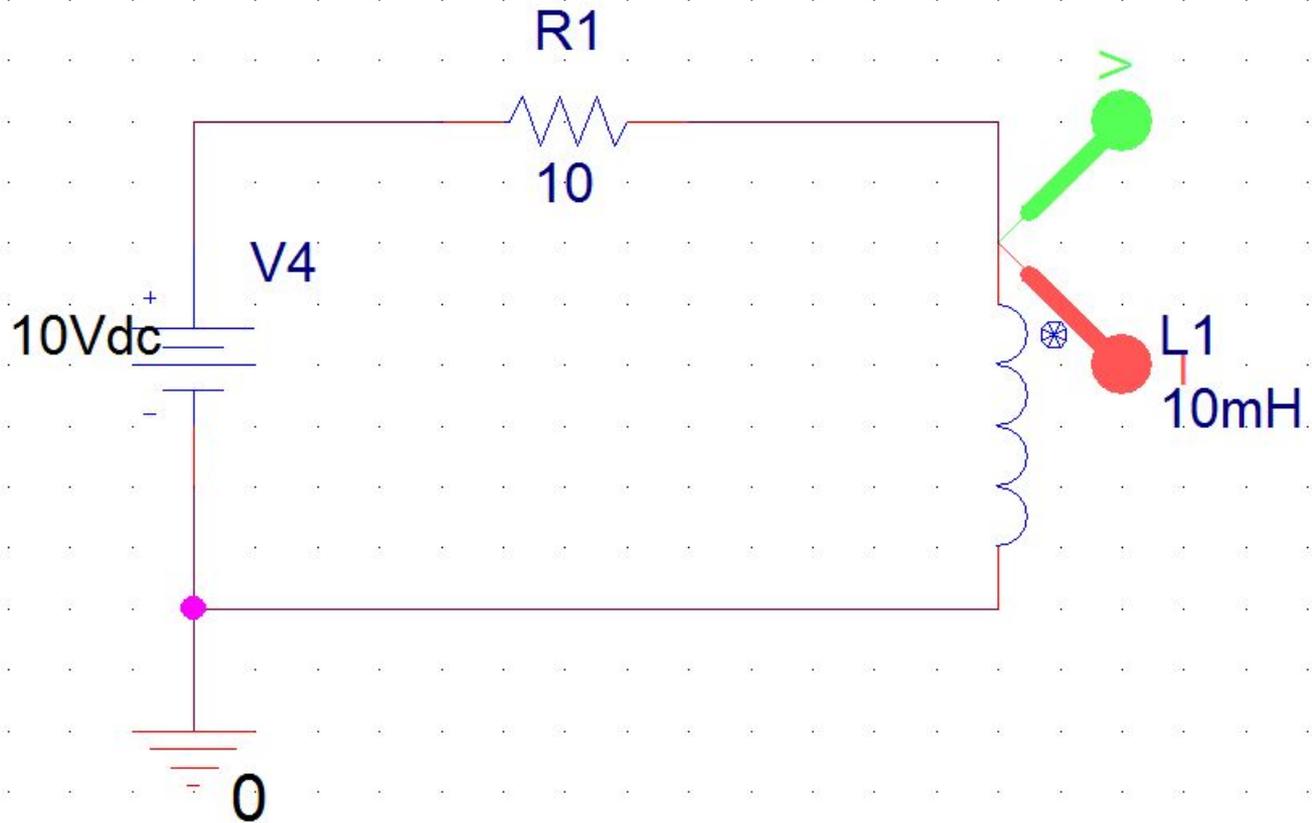
Display Format

Do Not Display
 Value Only
 Name and Value
 Name Only
 Both if Value Exists

Color:

Rotation: 0° 180°
 90° 270°

Создание электрической схемы



Окно ошибки

The screenshot shows the Cadence PSpice interface. The main window displays a netlist with the following content:

```
11
12
13
14 ** Creating circuit file "Lab1.cir"
15 ** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT SIMULATIONS
16
17 *Libraries:
18 * Profile Libraries :
19 * Local Libraries :
20 * From [PSpICE NETLIST] section of C:\Users\Николай\AppData\Roaming\SPB_Data\cdssetup\OrCAD_PSpice\16.6.0\PSpice.ini file:
21 .lib "noa.lib"
22
23 *Analysis directives:
24 .TRAN 0 10ms 0 0.1ms SKIPBP
25 .OPTIONS ADVCONV
26 .PROBE64 V(alias(*)) I(alias(*)) W(alias(*)) D(alias(*)) NOISE(alias(*))
27 .INC "..\SCHEMATIC1.net"
28
29
30
31 **** INCLUDING SCHEMATIC1.net ****
32 * source LAB1
33 R_R1      N00366 N00373 10 TC=0,0
34 L_L1      N00373 N00380 10mH
35 V_V4      N00366 N00380 10Vdc
36
37 **** RESUMING Lab1.cir ****
38 .END
39
40 ERROR(ORPSIM-15142): Node N00366 is floating
41 ERROR(ORPSIM-15142): Node N00373 is floating
42 ERROR(ORPSIM-15142): Node N00380 is floating
43
44
45 *
```

The error window at the bottom right displays the following message:

```
Reading and checking circuit
■ ERROR(ORPSIM-15142): Node N00366 is floating
■ ERROR(ORPSIM-15142): Node N00373 is floating
■ ERROR(ORPSIM-15142): Node N00380 is floating
Circuit has errors ... run aborted
See output file for details
INFO(DRPROBE-3188): Simulation aborted
```

The Command Window at the bottom left shows the following text:

```
Ps spice> Initializing Scripting...
Loading C:/Cadence/SPB_16.6/tools/psp
Ps spice>
```

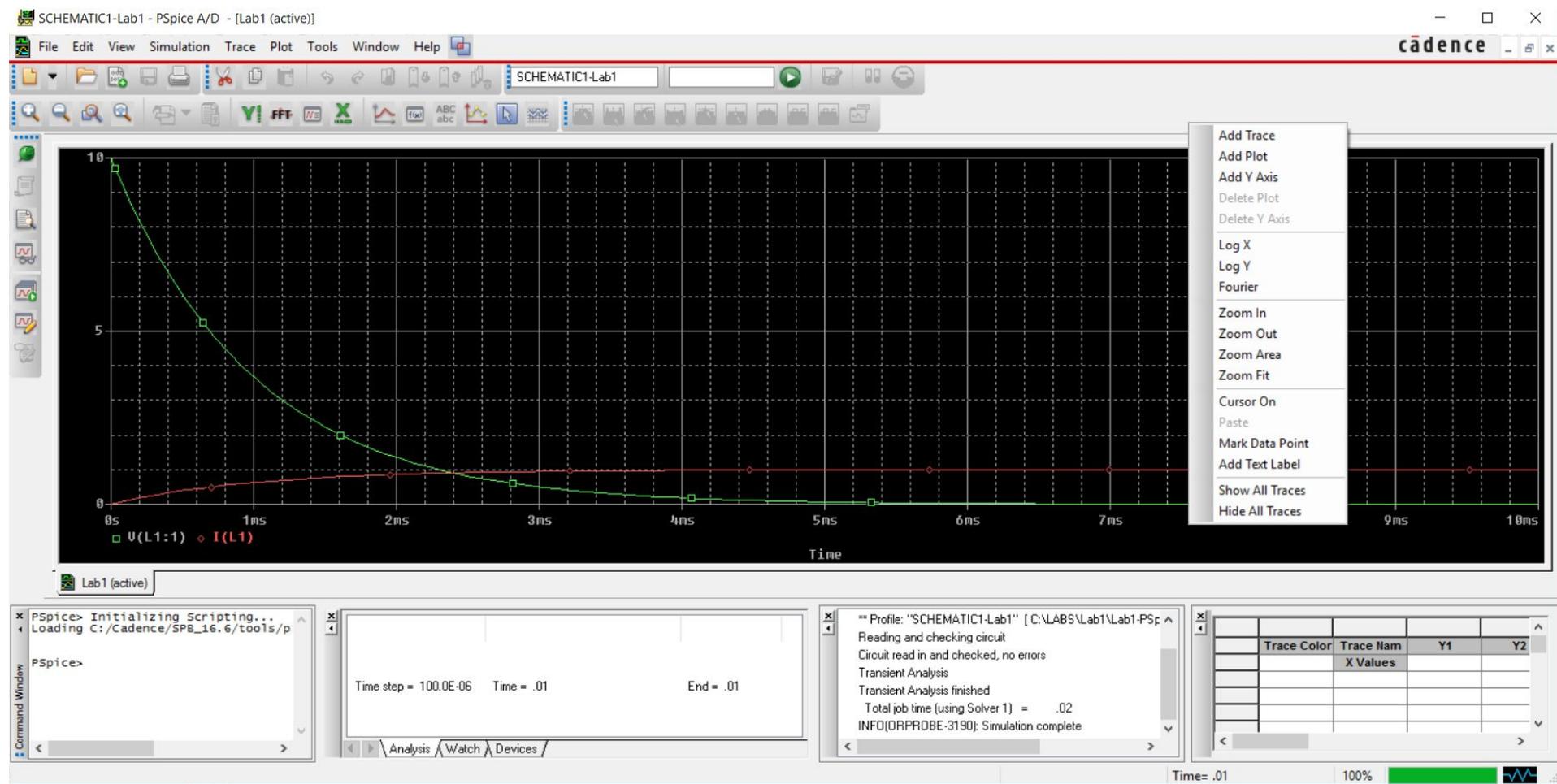
The Trace Window at the bottom right shows a table with the following columns: Trace Color, Trace Nam, Y1, and Y2. The table is currently empty.

Trace Color	Trace Nam	Y1	Y2
	X Values		

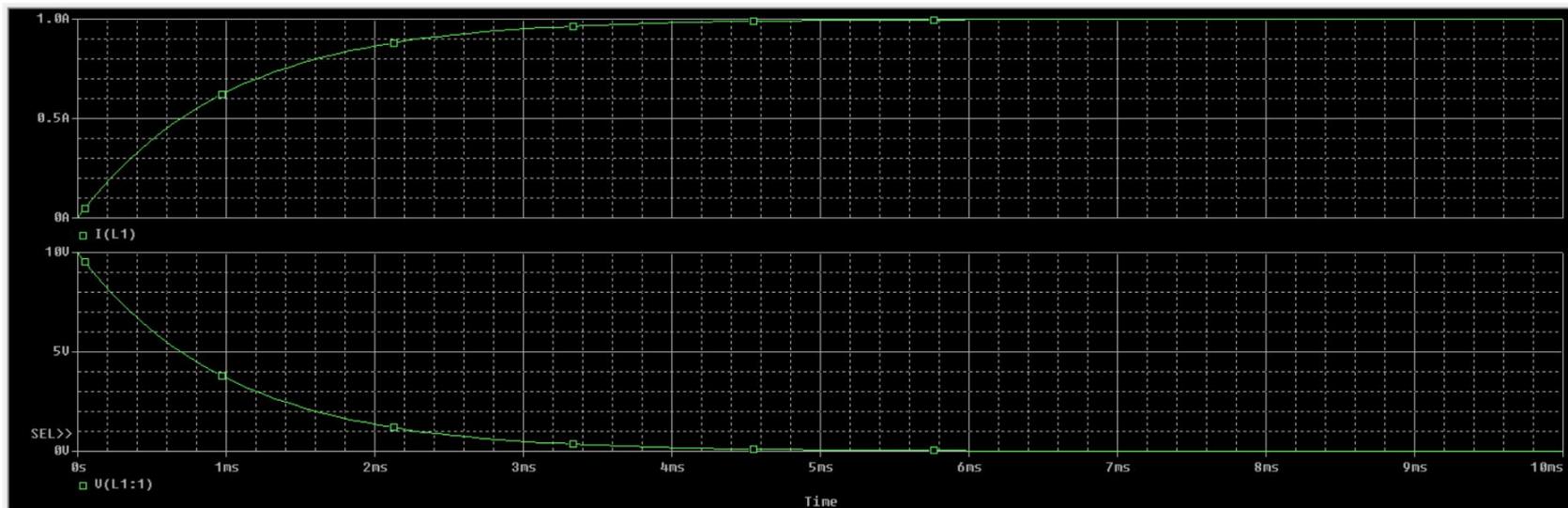
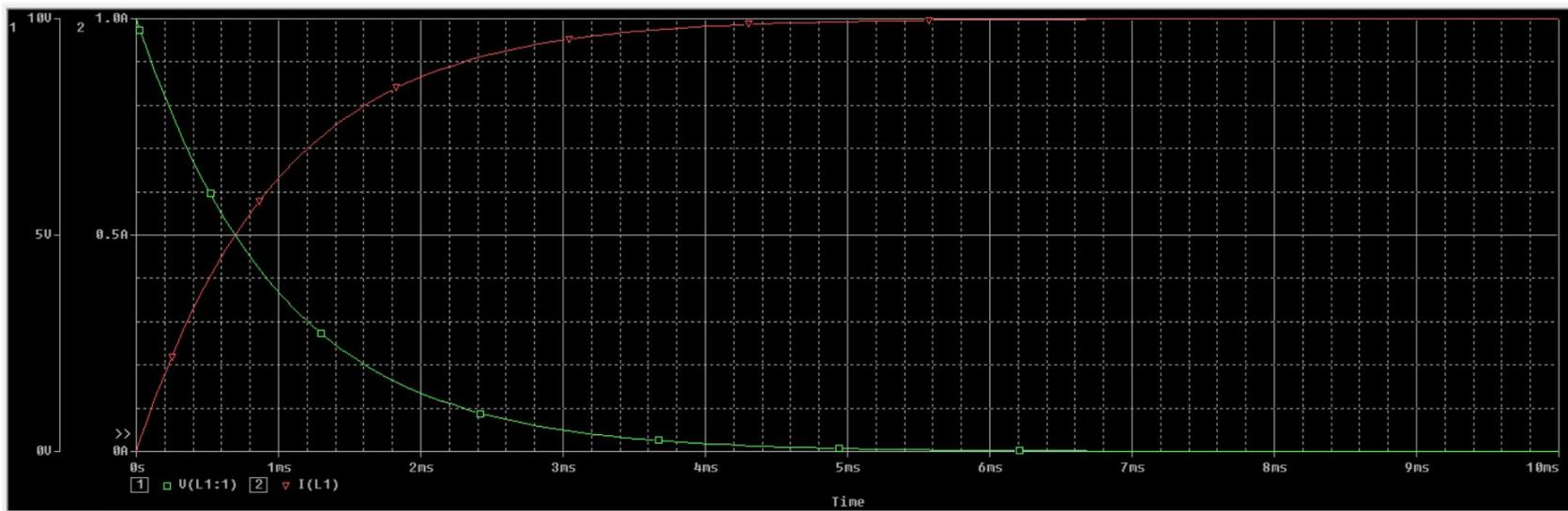
Position cursor at next peak value

Time: .01

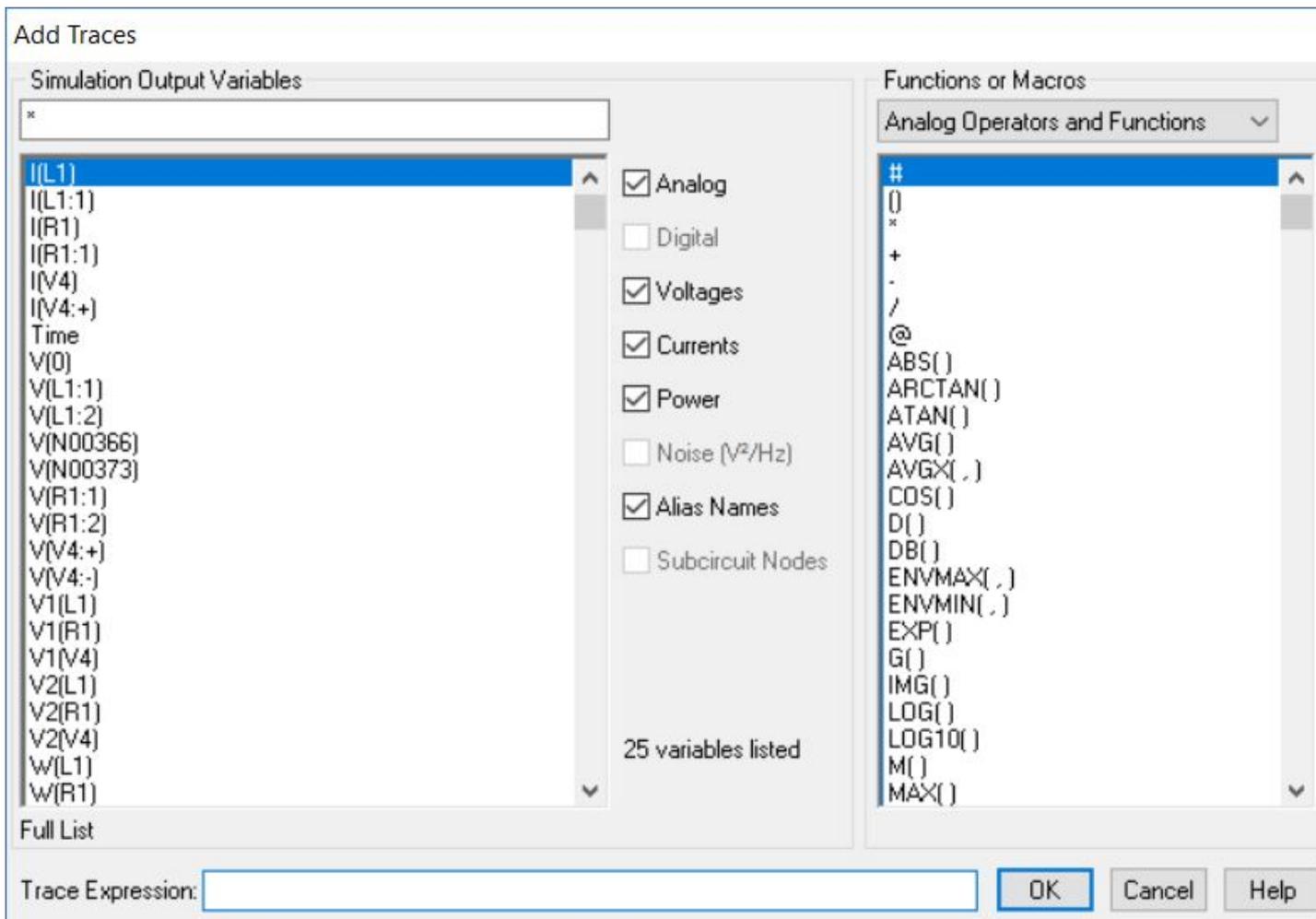
Результаты моделирования



Результаты моделирования

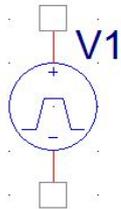


Результаты моделирования

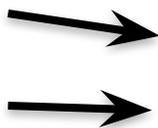


Источник импульсного напряжения

V1 =
 V2 =
 TD =
 TR =
 TF =
 PW =
 PER =



Время нарастания
 Первый уровень напряжения



A	
	SCHEMATIC1 : PAGE1
AC	
Color	Default
DC	
Designator	
Graphic	VPULSE.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	390
Location Y-Coordinate	200
Name	INS568
Part Reference	V1
PCB Footprint	
PER	
Power Pins Visible	<input type="checkbox"/>
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	V*@REFDES %+ %- ?DCID
PW	
Reference	V1
Source Library	C:\CADENCE\SPB_16...
Source Package	VPULSE
Source Part	VPULSE.Normal
TD	
TF	
TR	
V1	
V2	
Value	VPULSE

Период



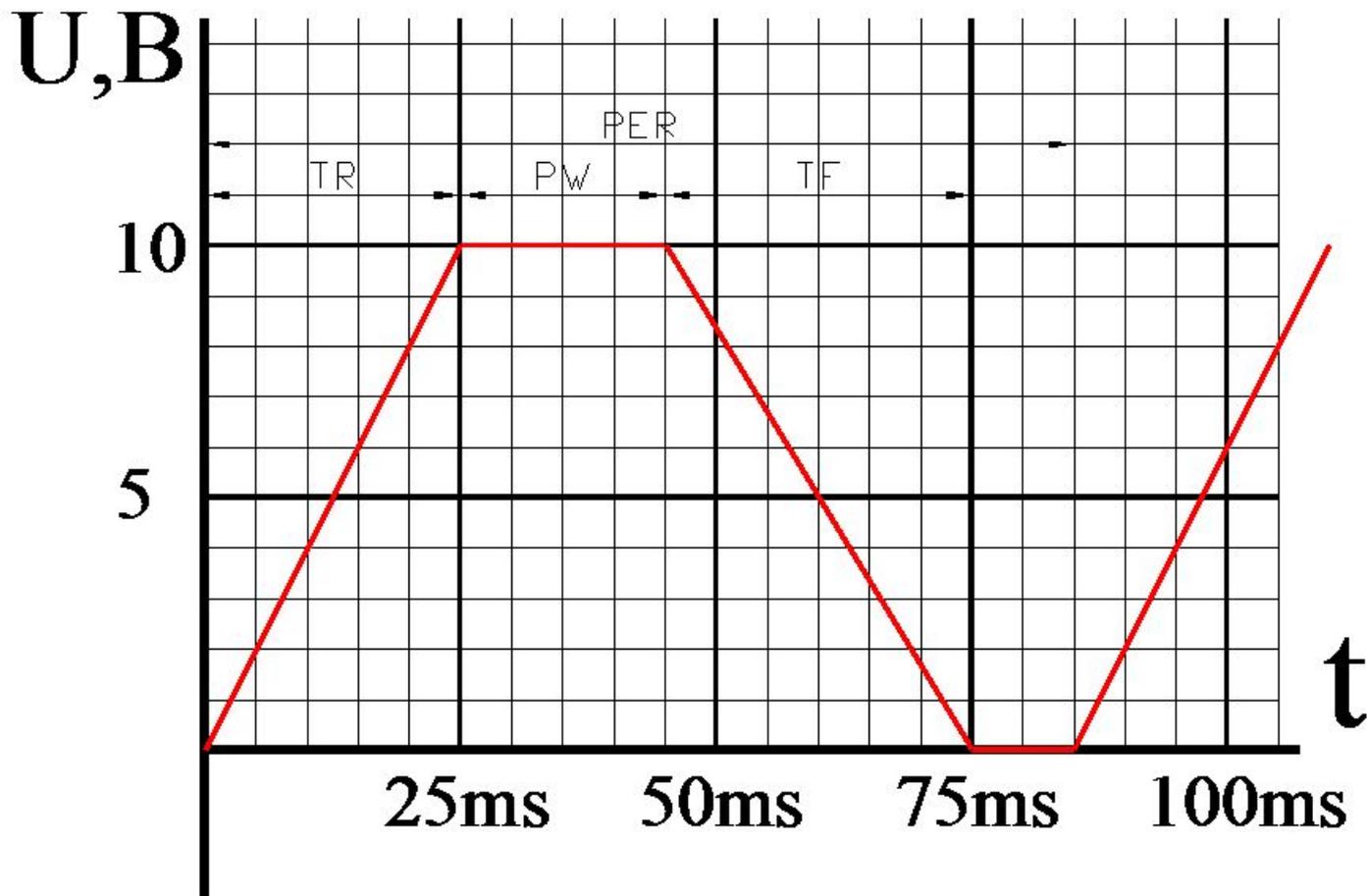
Длительность импульса



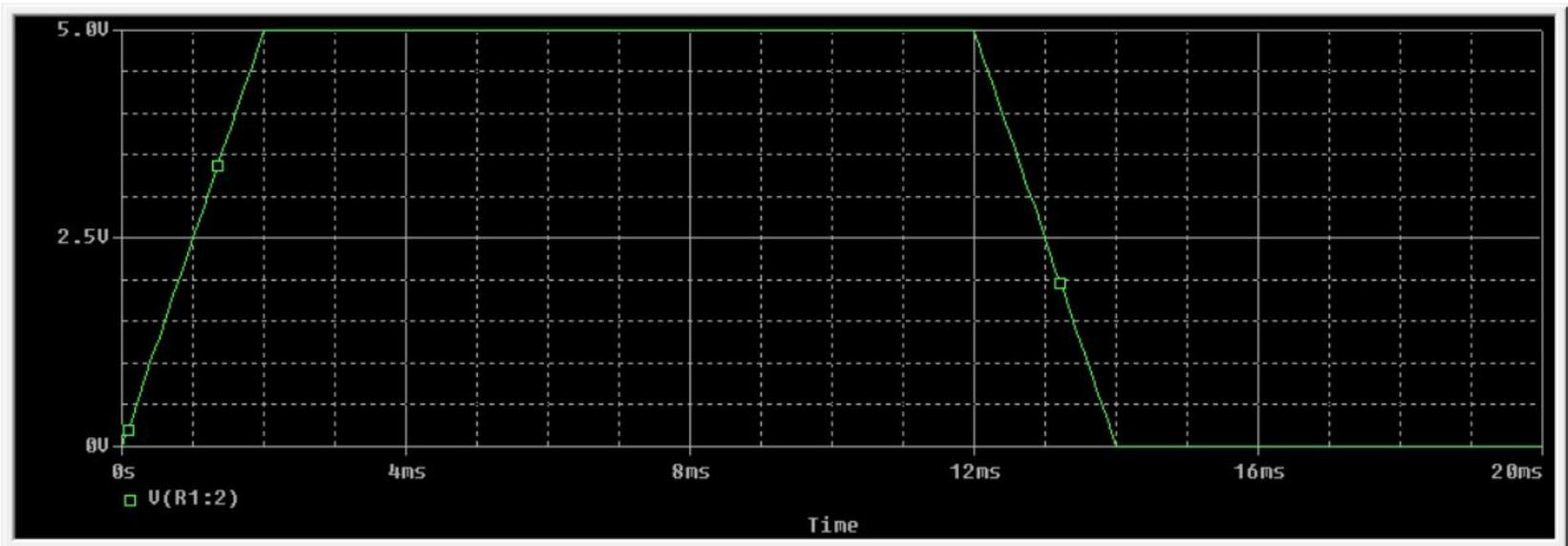
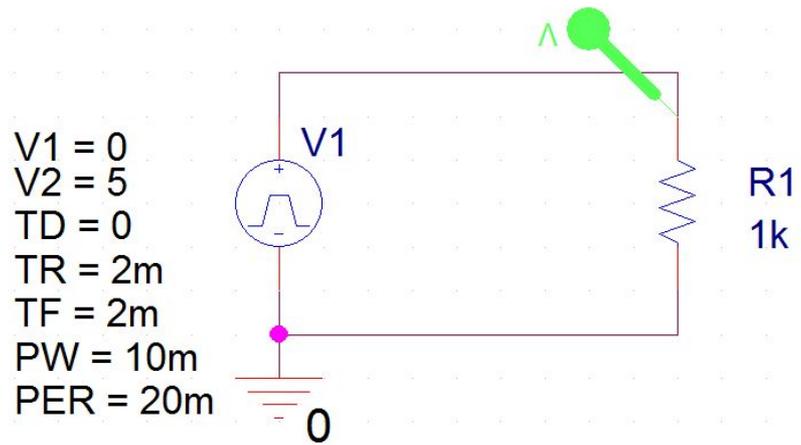
Время задержки
 Время спада
 Второй уровень напряжения



ИСТОЧНИК ИМПУЛЬСНОГО НАПРЯЖЕНИЯ

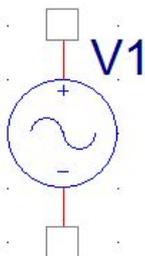


ИСТОЧНИК ИМПУЛЬСНОГО



Источник синусоидального напряжения

VOFF =
 VAMPL =
 FREQ =
 AC =



A	
SCHEMATIC1 : PAGE1	
AC	
Color	Default
DC	
Designator	
DF	0
FREQ	
Graphic	VSIN.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	410
Location Y-Coordinate	200
Name	INST86
Part Reference	V1
PCB Footprint	
PHASE	0
Power Pins Visible	<input type="checkbox"/>
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	V*@REFDES %+ %- ?DCID
Reference	V1
Source Library	C:\CADENCE\SPB_16
Source Package	VSIN
Source Part	VSIN.Normal
TD	0
Value	VSIN
VAMPL	
VOFF	

← Частота

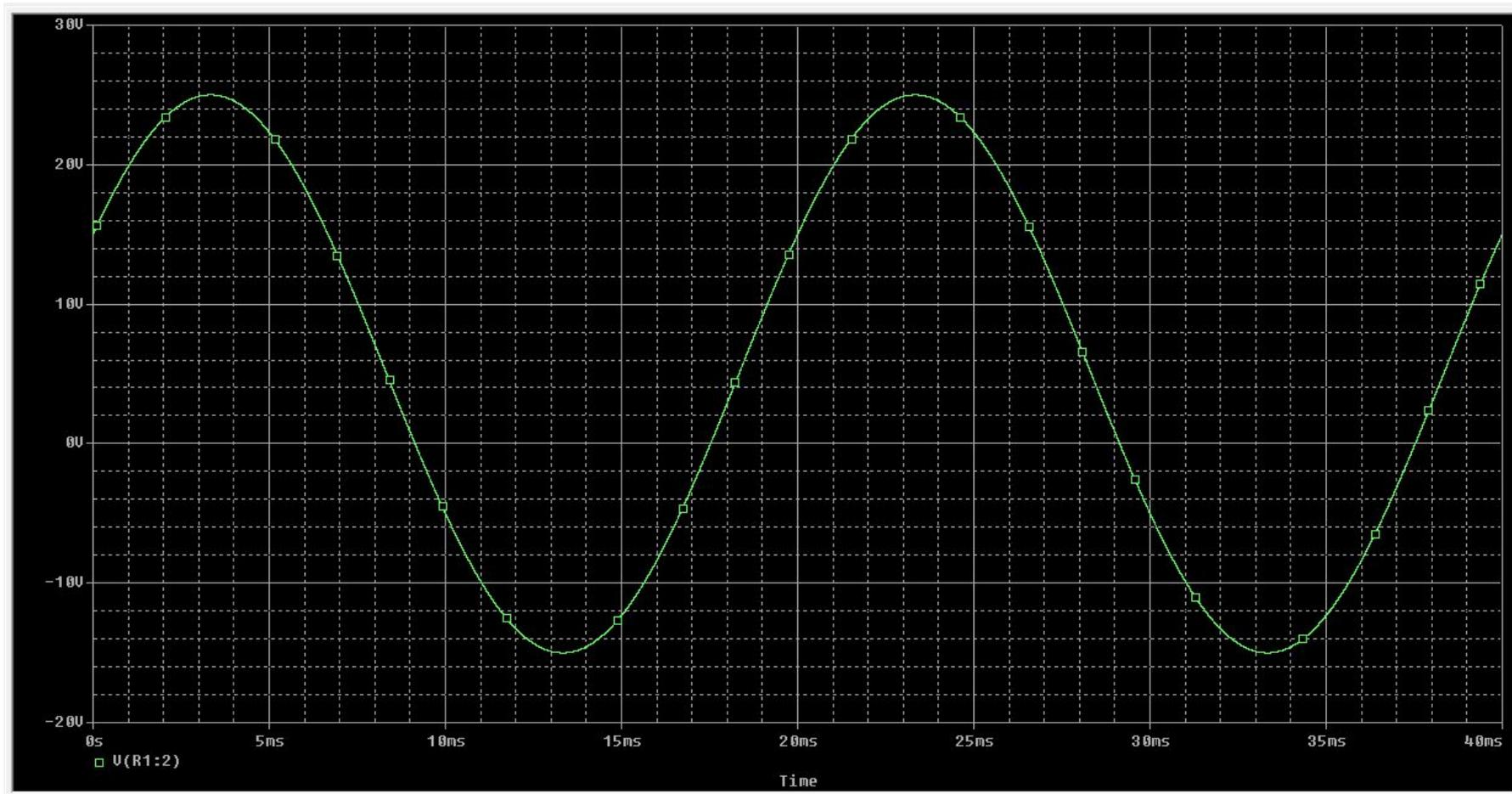
← Фазовый сдвиг

← Амплитудное значение

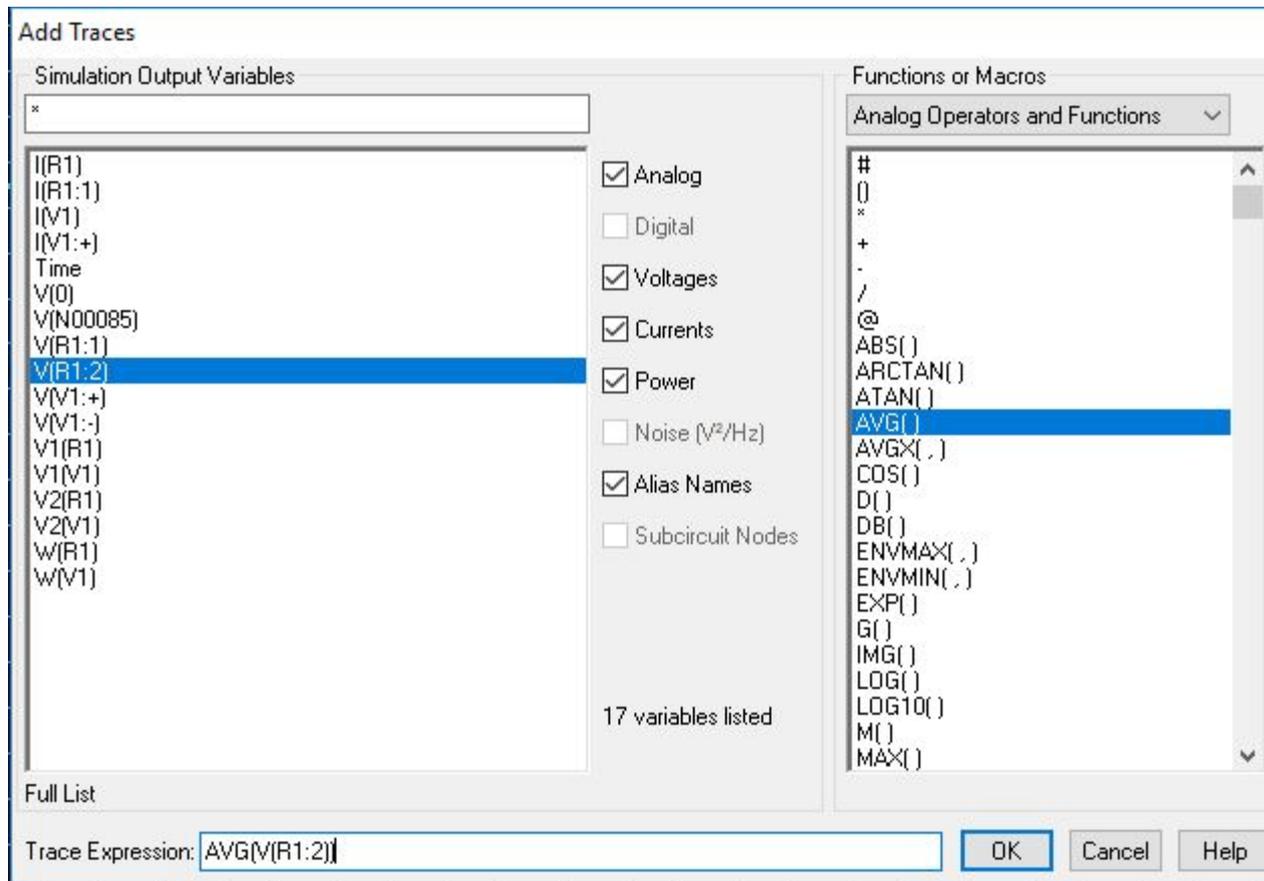
Постоянная составляющая



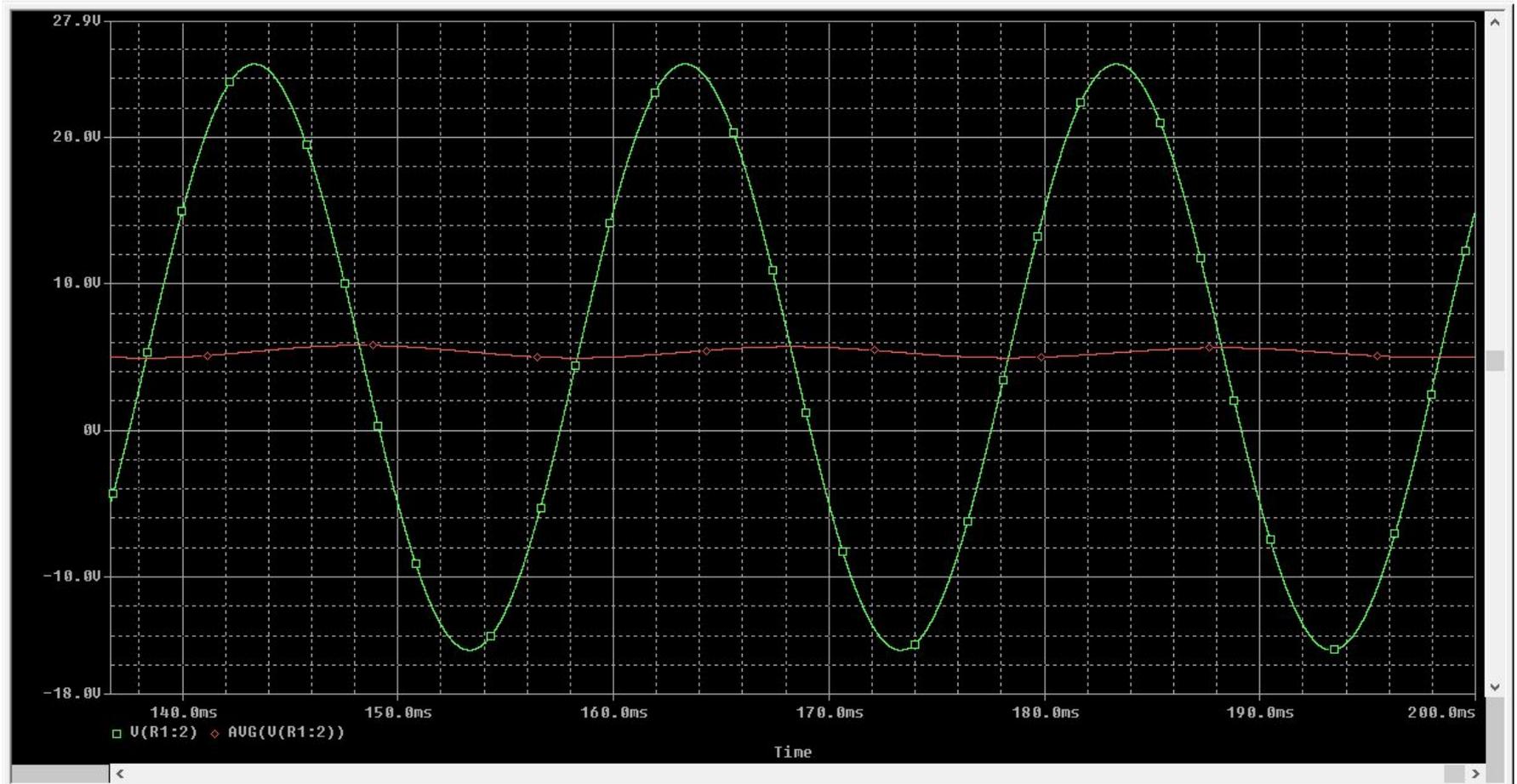
Источник синусоидального напряжения



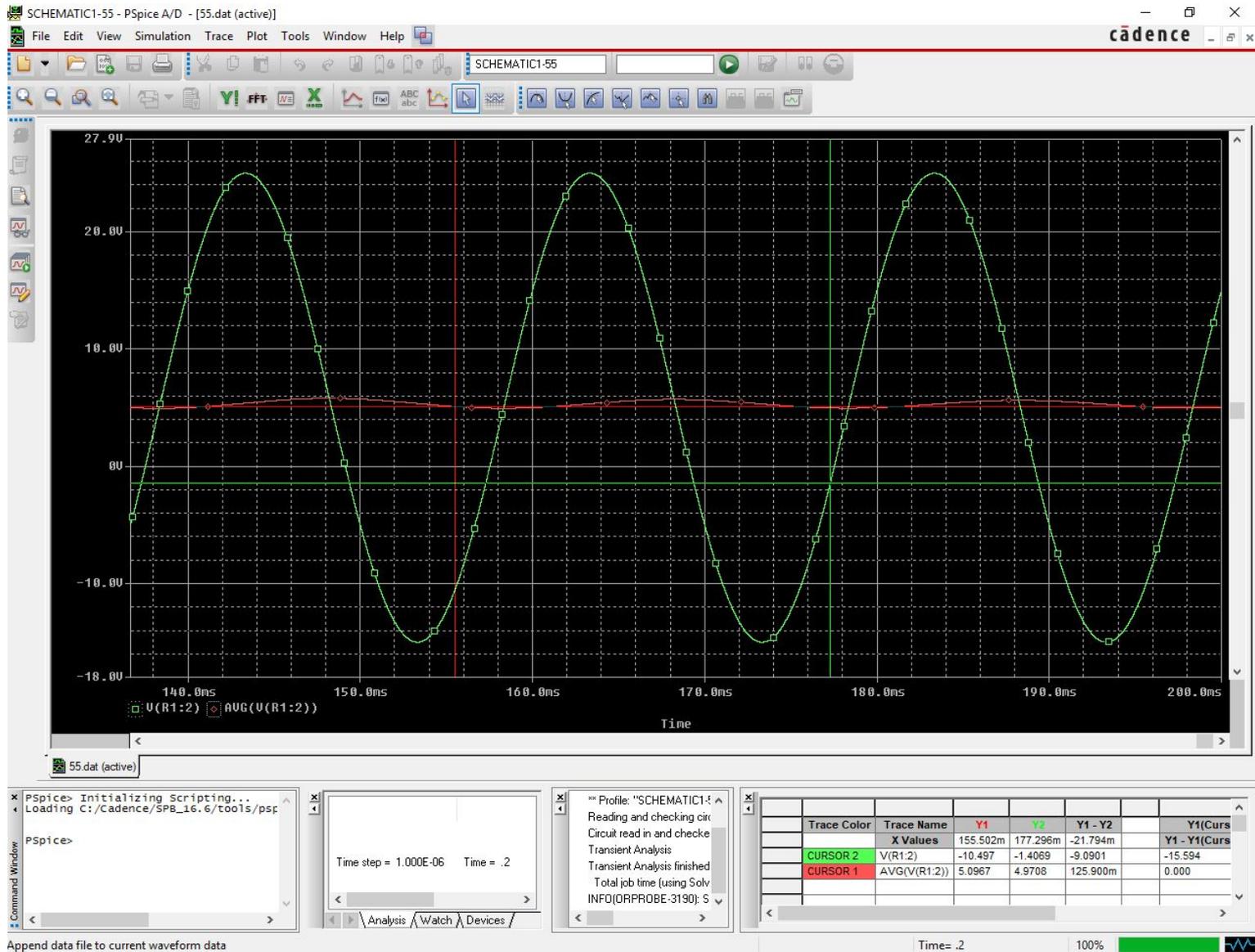
Среднее значение величины



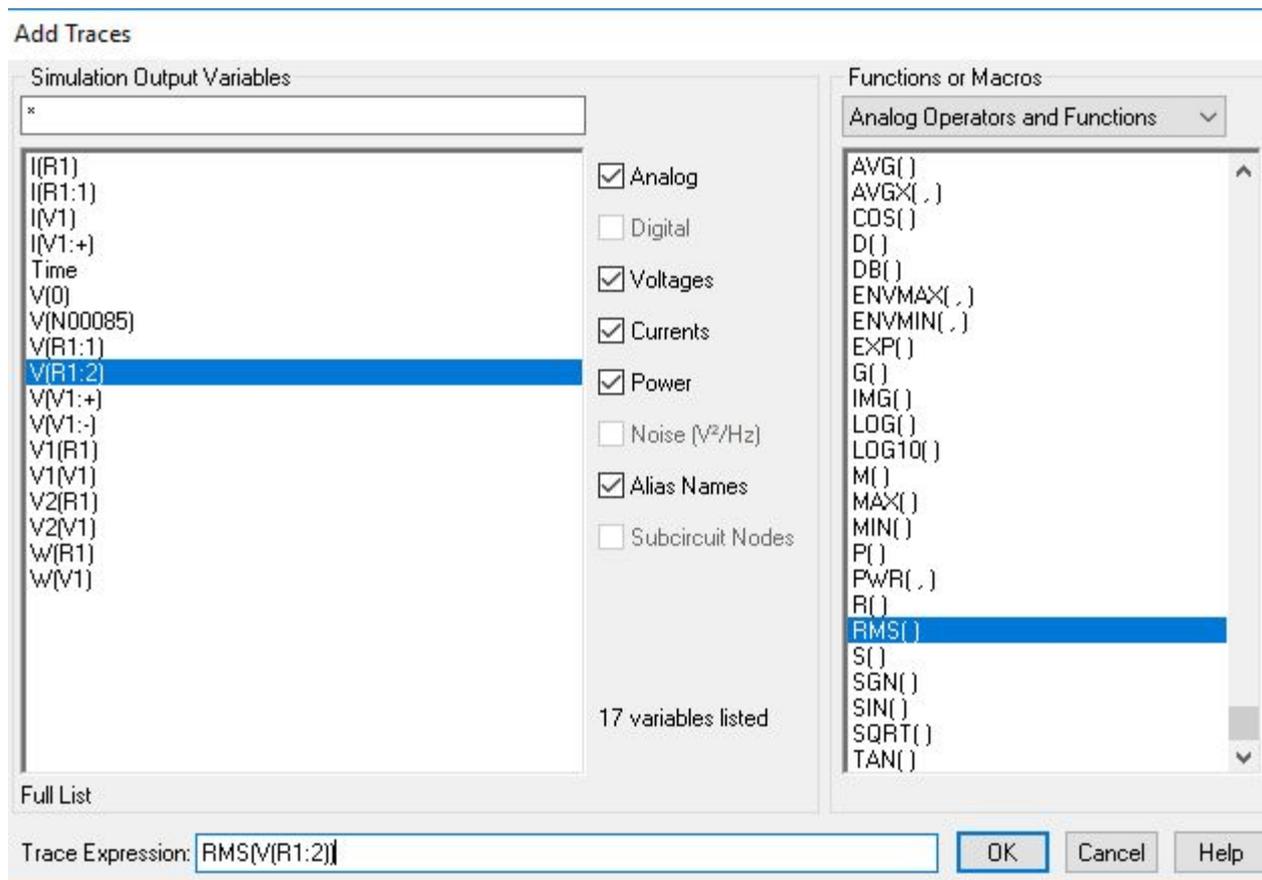
Среднее значение синусоидального напряжения



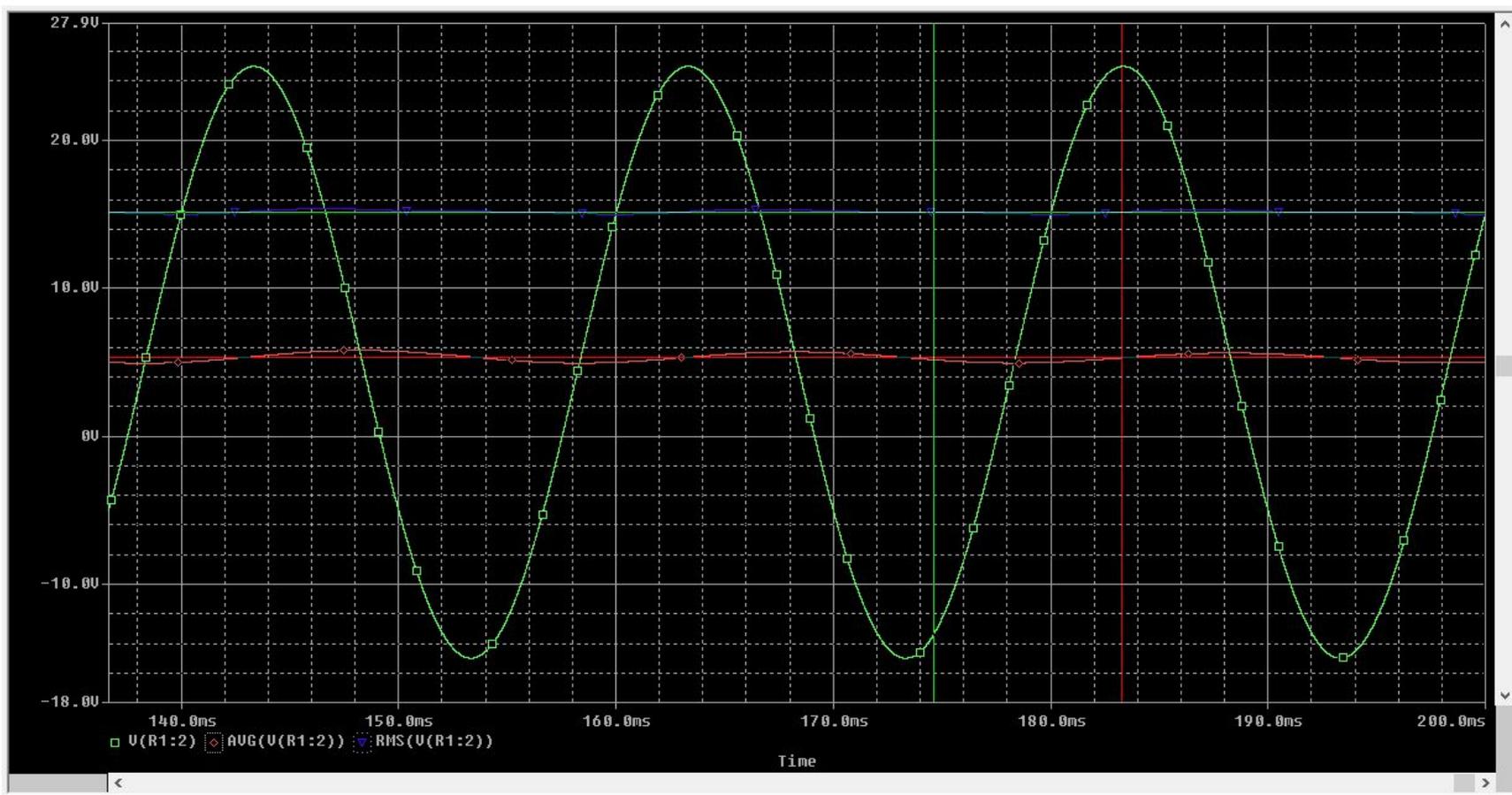
Использование курсоров



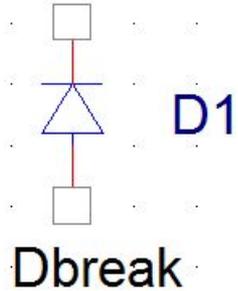
Среднее квадратичное значение величины



Среднее квадратичное (Действующее) значение напряжения



Идеализированный диод



The screenshot shows the Cadence PSpice Model Editor interface. The window title is "LAB1:Dbreak - PSpice Model Editor - [Model Text]". The menu bar includes "File", "Edit", "View", "Model", "Plot", "Tools", "Window", and "Help". The toolbar contains various icons for file operations and simulation. The "Models List" panel on the left shows a table with the following content:

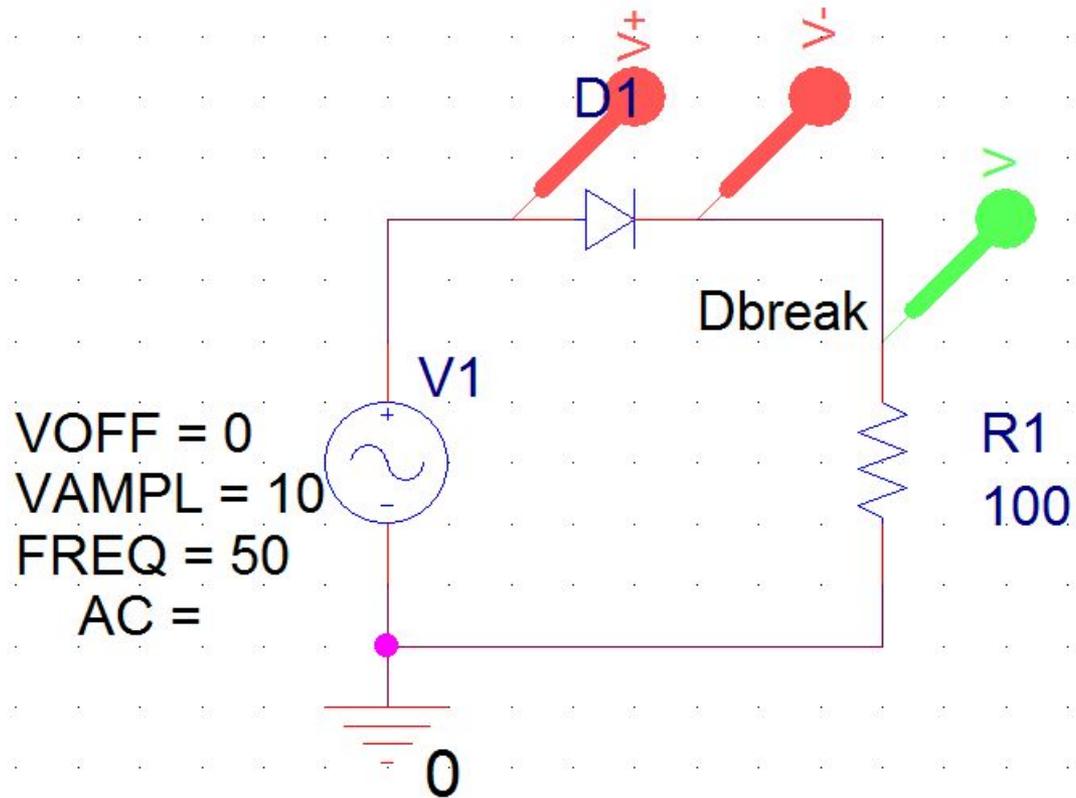
Model Name	Type
Dbreak*	Diode

The main editor area contains the following model definition text:

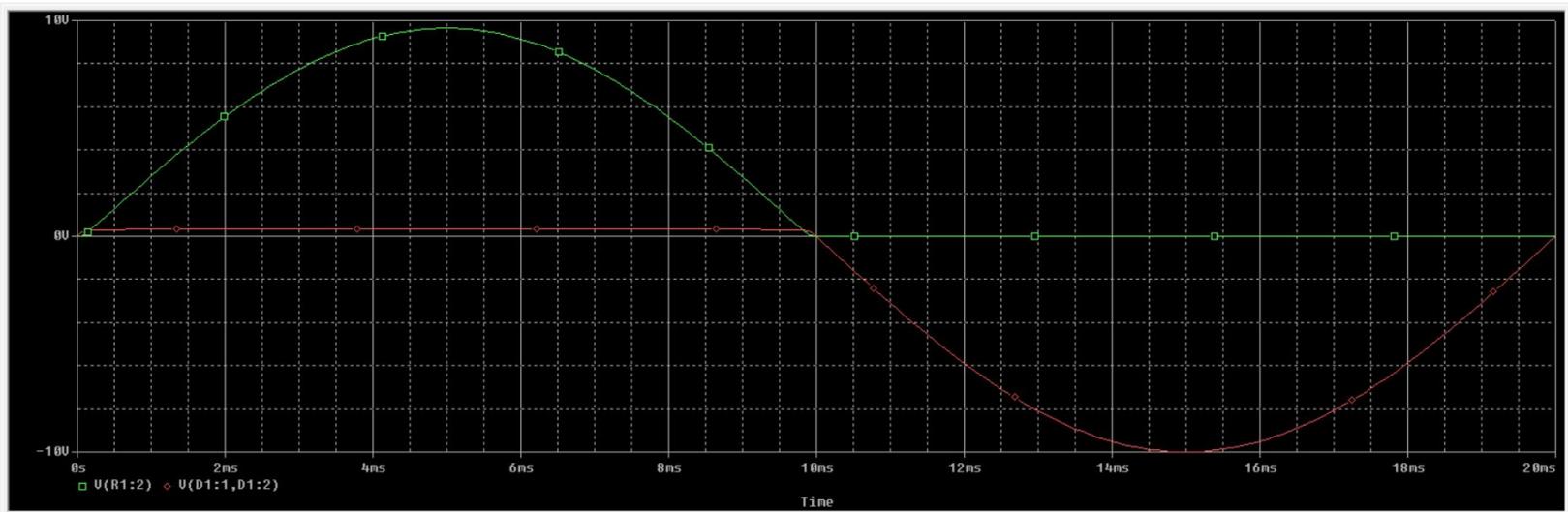
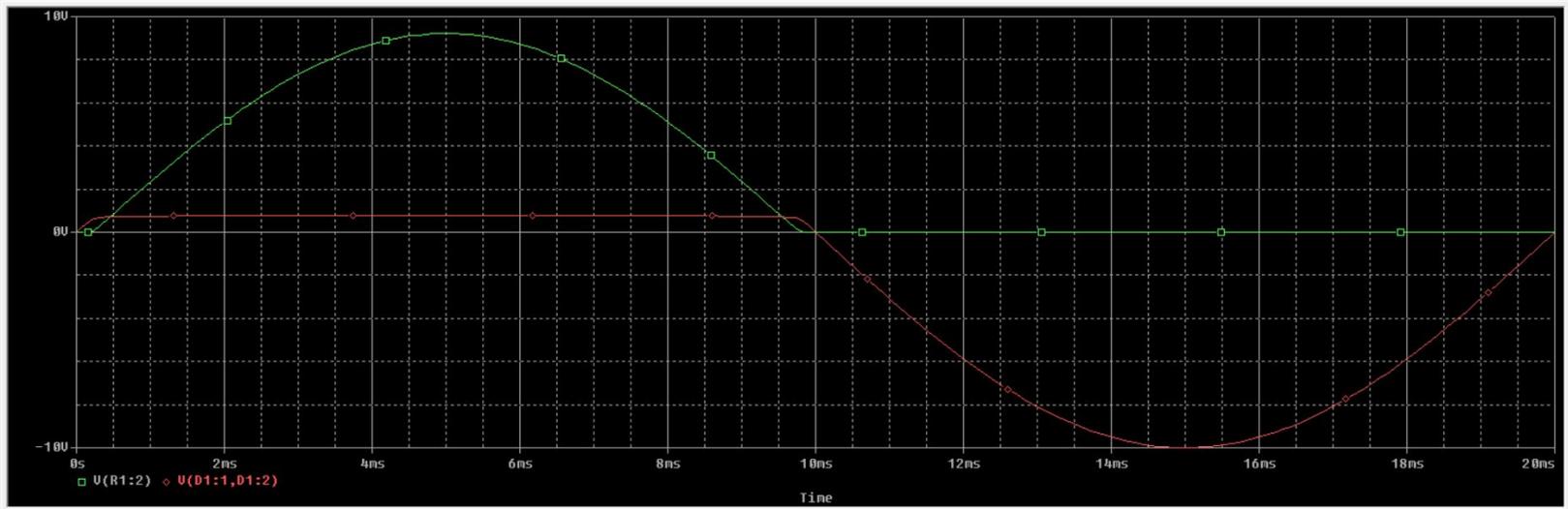
```
.model Dbreak D Is=1e-14 Cjo=.1pF Rs=.1
```

The status bar at the bottom indicates "Ready" and "NUM".

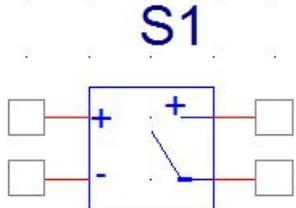
Идеализированный диод



Идеализированный диод



Идеализированный ключ



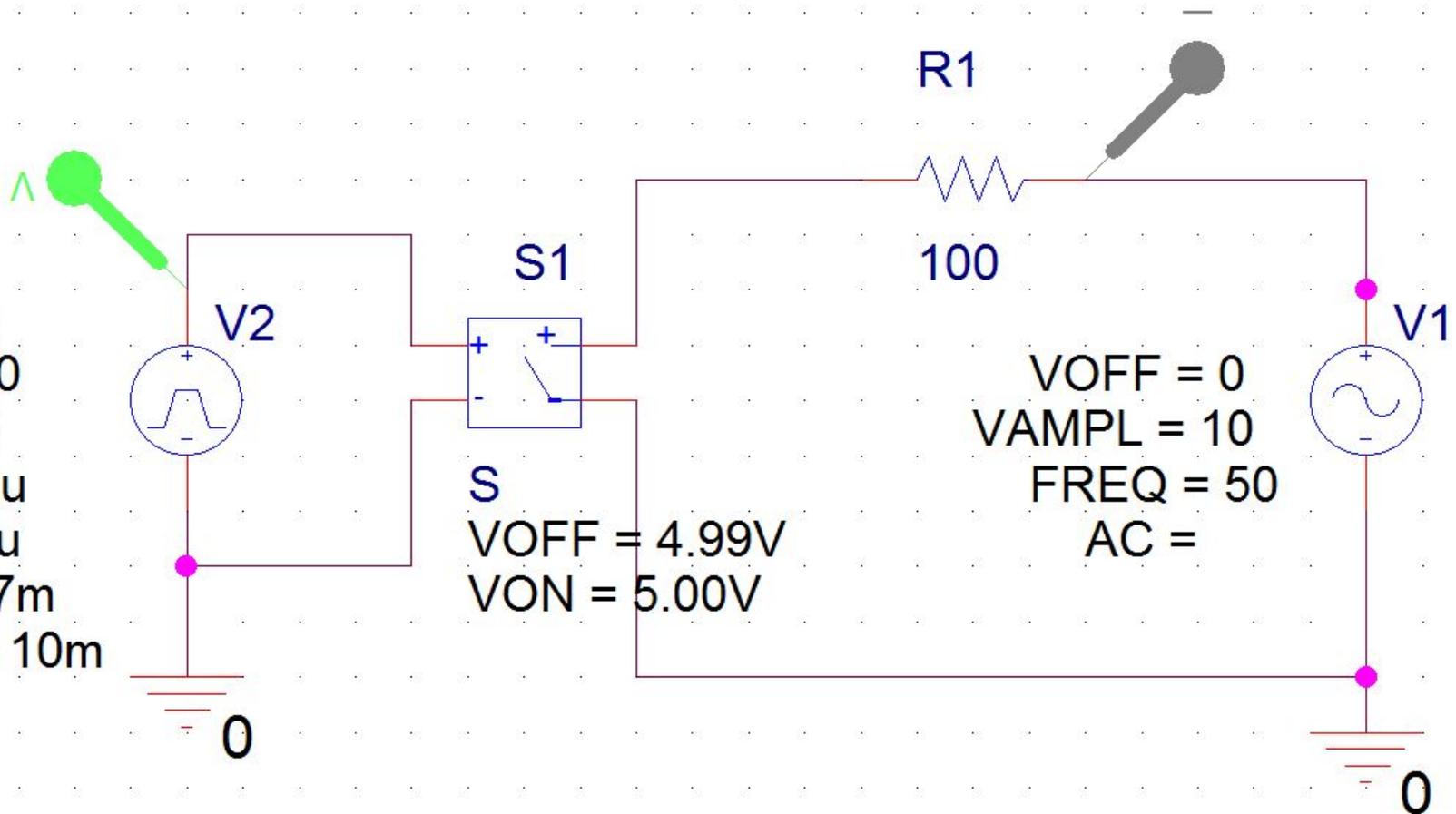
S
 $V_{OFF} = 0.0V$
 $V_{ON} = 1.0V$

A	
SCHEMATIC1 : PAGE1	
Color	Default
Designator	
Graphic	S.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	<none>
Location X-Coordinate	420
Location Y-Coordinate	470
Name	INS1279
Part Reference	S1
PCB Footprint	
Power Pins Visible	<input type="checkbox"/>
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	S^@REFDES %3 %4 %1 %
Reference	S1
ROFF	1e6
RON	1.0
Source Library	C:\CADENCE\SPB_16...
Source Package	S
Source Part	S.Normal
Value	S
VOFF	0.0V
VON	1.0V

Сопротивление
 В
 выключенном
 состоянии
 Сопротивление во
 включенном
 состоянии
 Напряжени
 е
 выключени
 Напряжени
 я
 е
 включения

Идеализированный ключ

V1 = 0
V2 = 10
TD = 0
TR = 1u
TF = 1u
PW = 7m
PER = 10m



Идеализированный ключ

