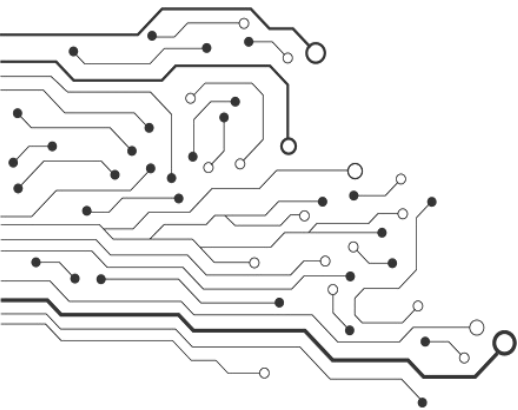
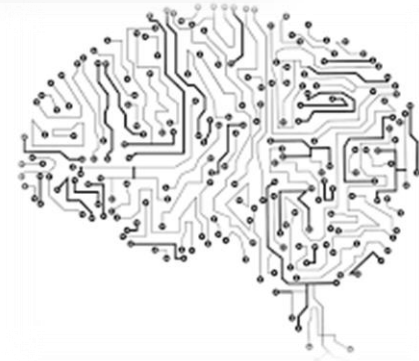


OrCAD Capture-PSPICE



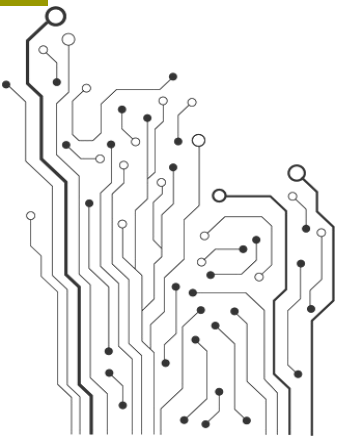
Dr. Sarwan Singh
NIELIT Chandigarh



Agenda

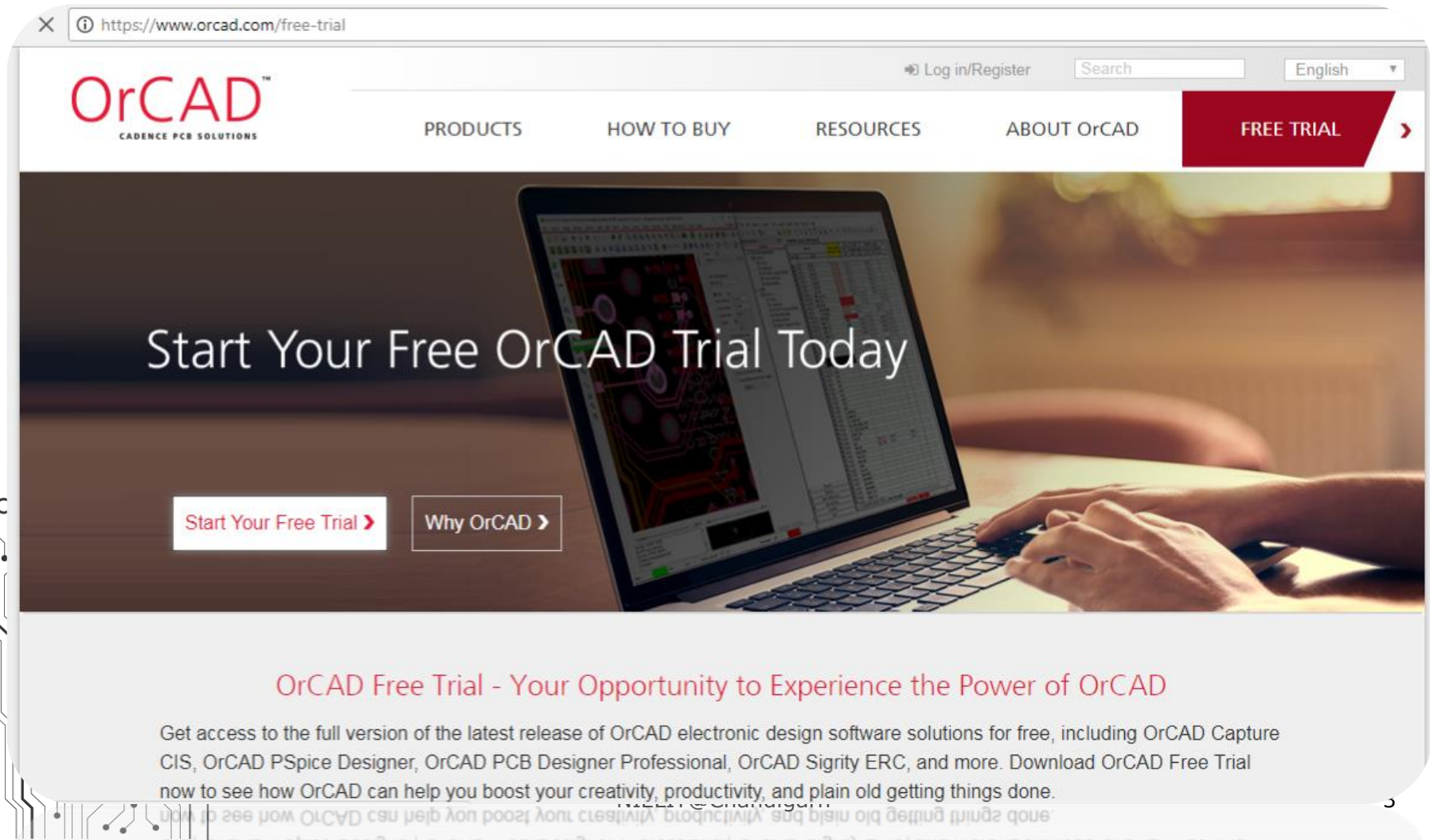


- ❑ Introduction – PSpice
- ❑ Creating Circuit Schematic
- ❑ Place the components, source, ground, probe, etc.
- ❑ Circuit simulation
 - Halfwave rectifier, bridge rectifier



Download free demo

□ www.orcad.com/free-trial



The screenshot shows the OrCAD website's free trial page. The browser address bar displays <https://www.orcad.com/free-trial>. The OrCAD logo is in the top left, with the tagline "CADENCE PCB SOLUTIONS". Navigation links include "PRODUCTS", "HOW TO BUY", "RESOURCES", "ABOUT OrCAD", and a prominent red "FREE TRIAL" button. The main banner features a laptop displaying a circuit design with the text "Start Your Free OrCAD Trial Today". Below the banner are two buttons: "Start Your Free Trial" and "Why OrCAD". The footer section is titled "OrCAD Free Trial - Your Opportunity to Experience the Power of OrCAD" and contains a paragraph about the trial's benefits.

OrCAD
CADENCE PCB SOLUTIONS

Log in/Register Search English

PRODUCTS HOW TO BUY RESOURCES ABOUT OrCAD FREE TRIAL

Start Your Free OrCAD Trial Today

Start Your Free Trial Why OrCAD

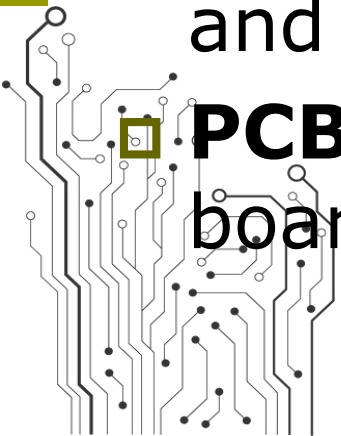
OrCAD Free Trial - Your Opportunity to Experience the Power of OrCAD

Get access to the full version of the latest release of OrCAD electronic design software solutions for free, including OrCAD Capture CIS, OrCAD PSpice Designer, OrCAD PCB Designer Professional, OrCAD Sigrity ERC, and more. Download OrCAD Free Trial now to see how OrCAD can help you boost your creativity, productivity, and plain old getting things done.



PSpice comprises three main applications

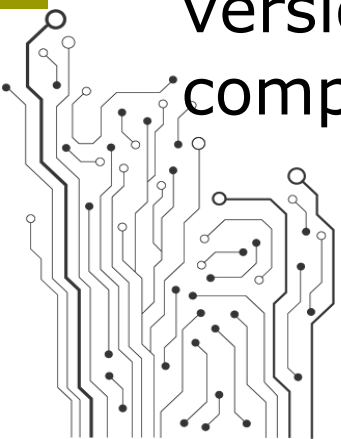
- ❑ **Capture** is used to draw a circuit on the screen, known formally as schematic capture. It offers great flexibility compared with a traditional pencil and paper drawing, as design changes can be incorporated and errors corrected quickly and easily
- ❑ **PSpice** simulates the captured circuit. You can analyse its behaviour in many ways and confirm that it performs as specified.
- ❑ **PCB Editor** is used to design printed circuit boards

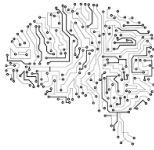




Introduction - SPICE

- ❑ SPICE (**Simulation Program for Integrated Circuits Emphasis**) was developed at the University of California at Berkeley in the 1970s, and for many years has been the most widely used circuit simulator in the electronics industry.
- ❑ SPICE is a general purpose analog circuit simulator that is used to verify circuit designs and to predict the circuit behaviour.
- ❑ PSpice is a PC version of SPICE and HSpice is a version that runs on workstations and larger computers.





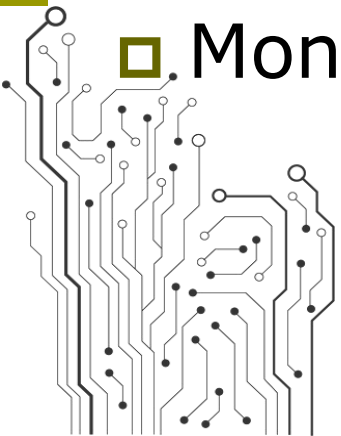
- PSpice has analog and digital libraries of standard components (such as NAND, NOR, flip-flops, and other digital gates, op amps, etc) which makes it a useful tool for a wide range of analog and digital applications.

Steps.

1. Draw an electronic circuit on the computer using Capture.
2. Simulate it with PSpice using specific models for devices.
3. Analyse its behaviour with Probe, which can produce a range of plots. Historically this was a separate application but it is now integrated with PSpice.

Types of Analysis

- ❑ Non-linear DC analysis
- ❑ Non-linear transient analysis
- ❑ Linear AC Analysis
- ❑ Noise analysis
- ❑ Sensitivity analysis
- ❑ Distortion analysis
- ❑ Fourier analysis
- ❑ Monte Carlo Analysis





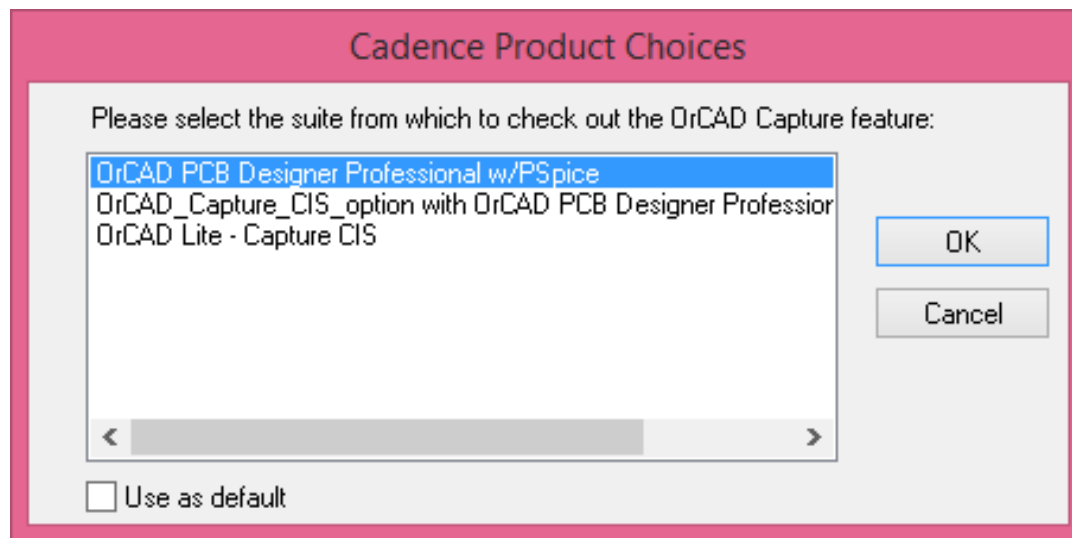
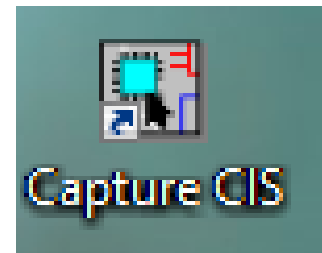
Circuit Components available

- ❑ Independent and dependent voltage and current sources
- ❑ Resistors
- ❑ Capacitors
- ❑ Inductors
- ❑ Mutual inductors
- ❑ Transmission lines
- ❑ Operational amplifiers
- ❑ Switches
- ❑ Diodes
- ❑ Bipolar transistors
- ❑ MOS transistors
- ❑ JFET
- ❑ MOSFET
- ❑ Digital gates

Using Capture CIS



- ❑ Open Capture CIS
- ❑ Select OrCAD PCB Designer w/PSpice



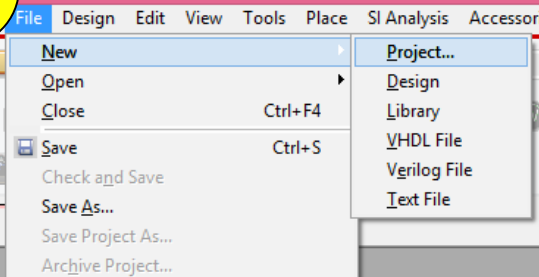
Creating Circuit Schematic

1 **Open new Project**

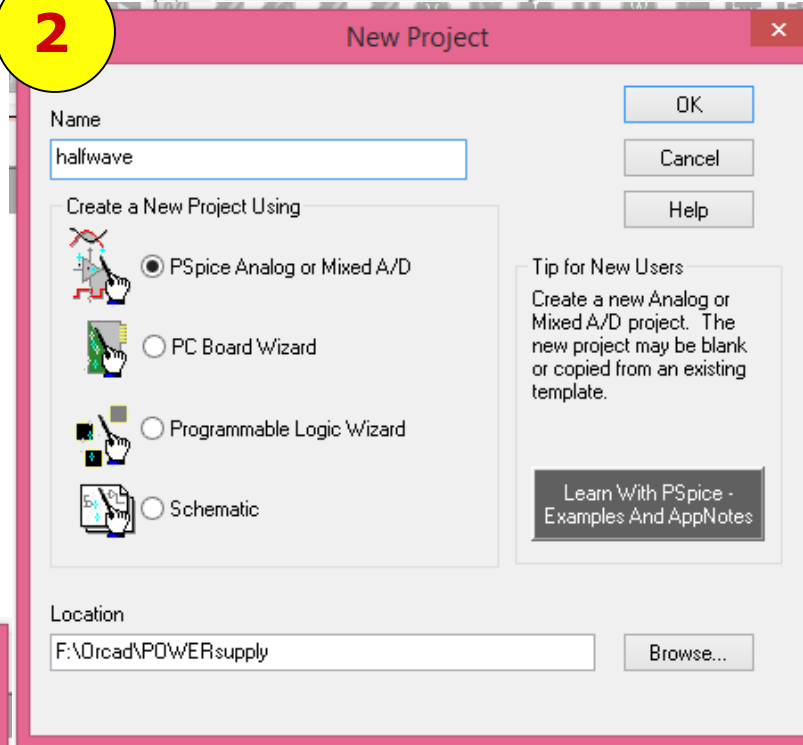
2 **Select PSpice Analog or Mixed A/D**

3 **Create PSpice Project**

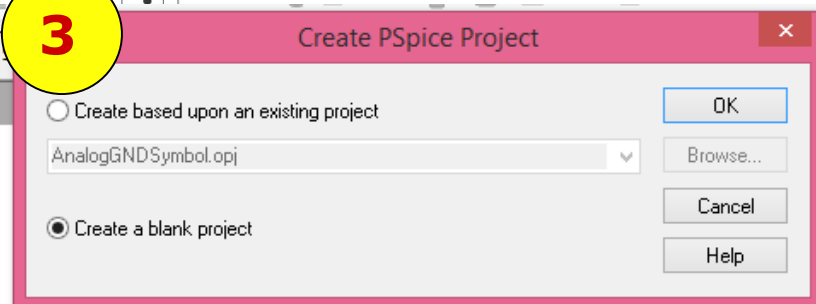
1



2

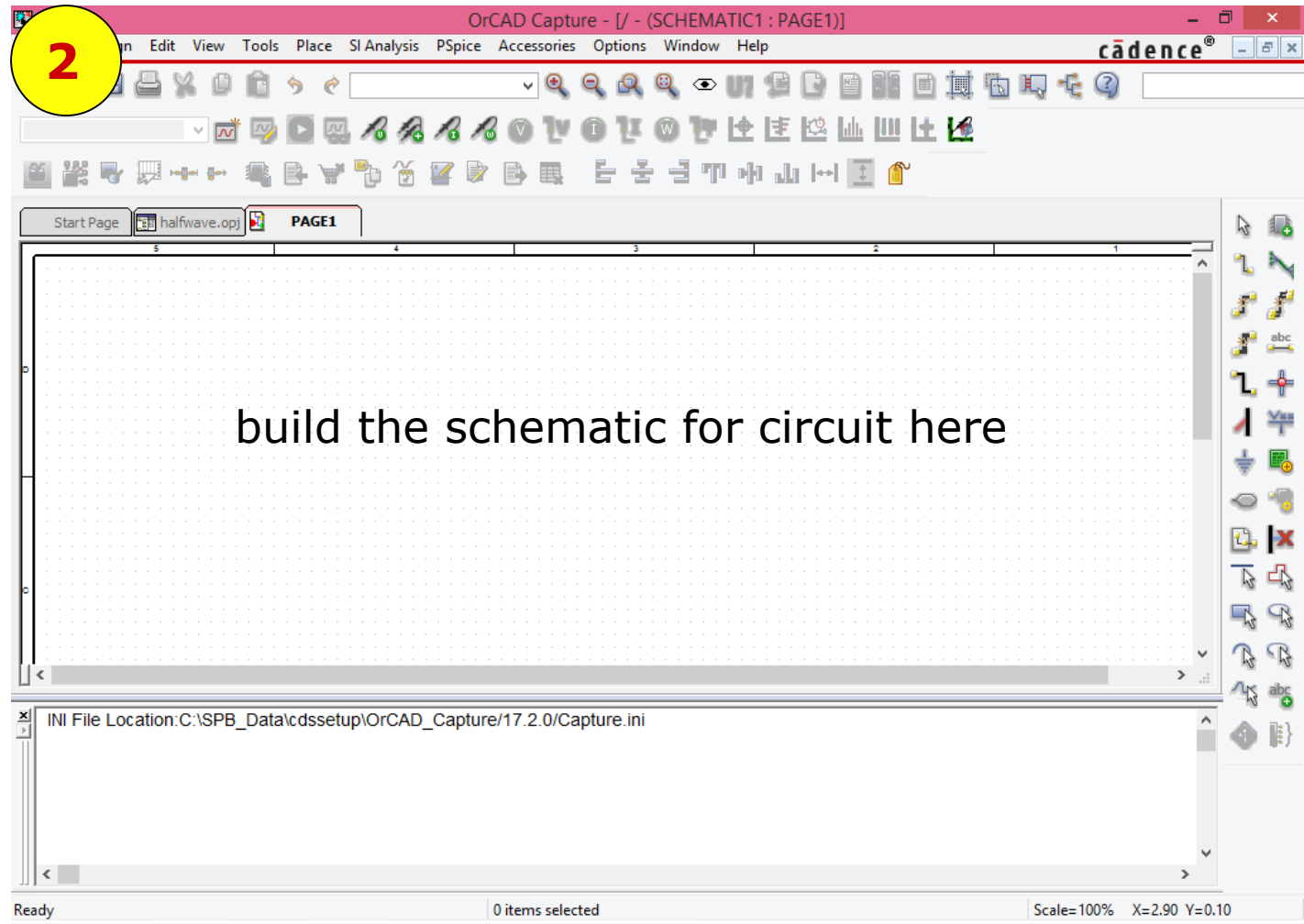
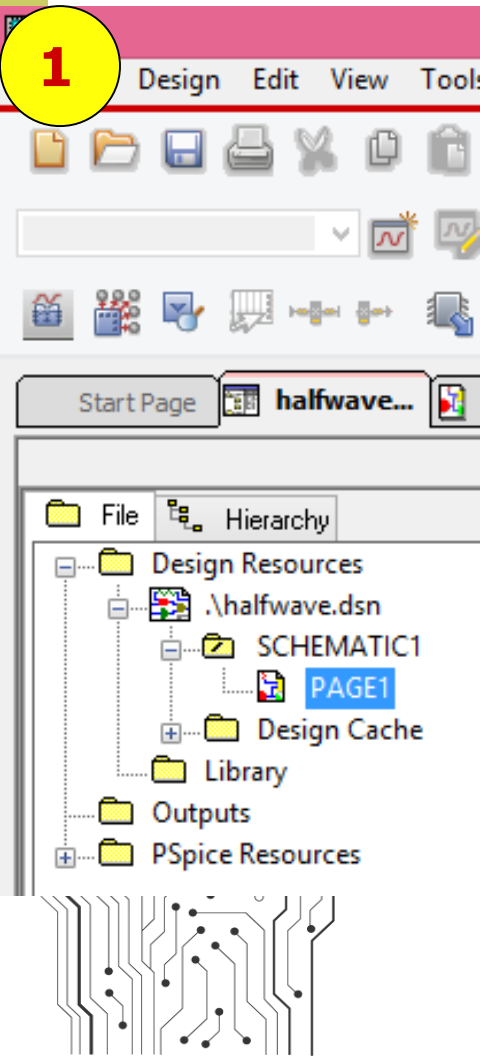


3

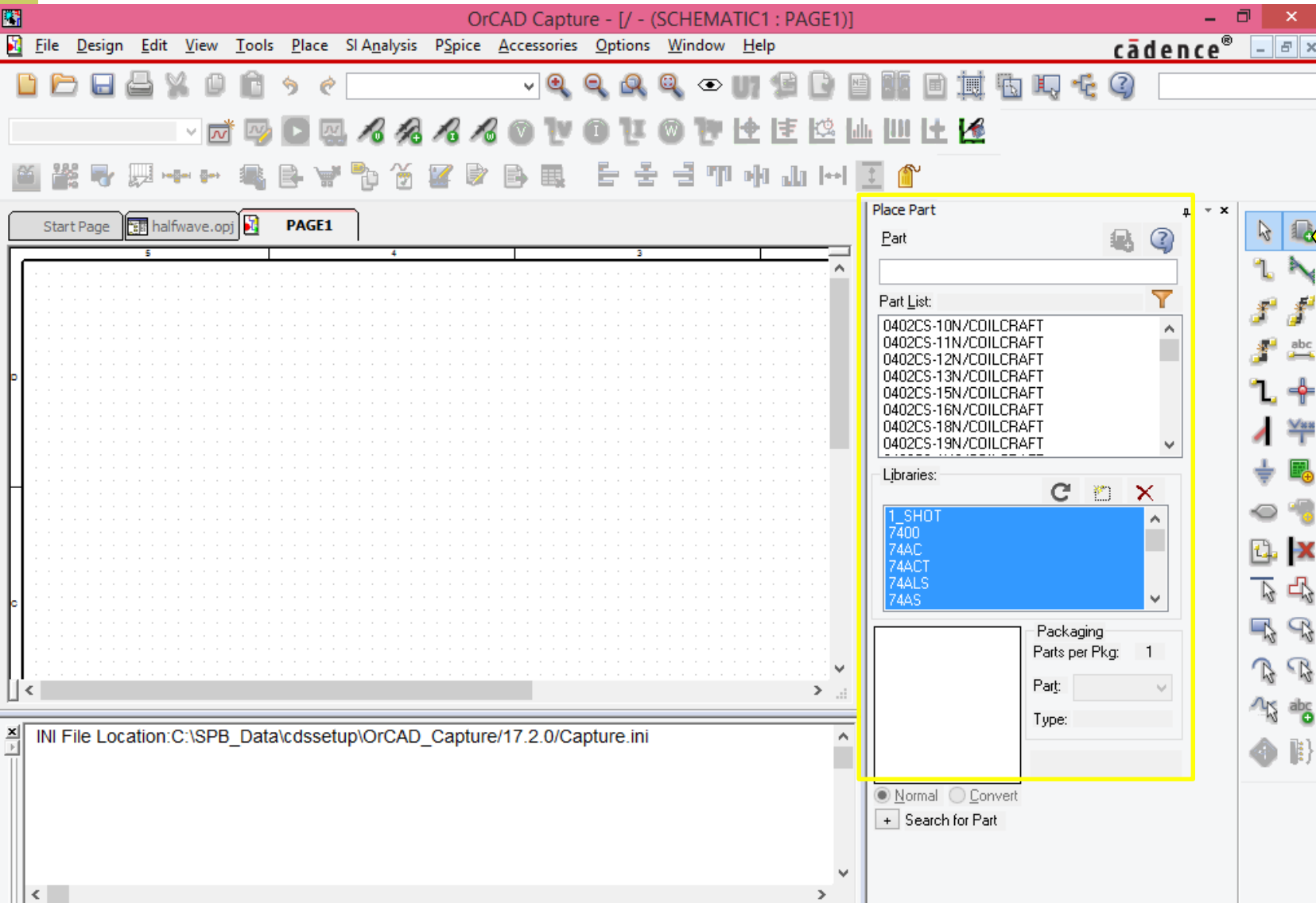
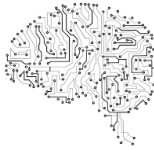


Creating Circuit Schematic...

❑ **blank schematic screen** will open.



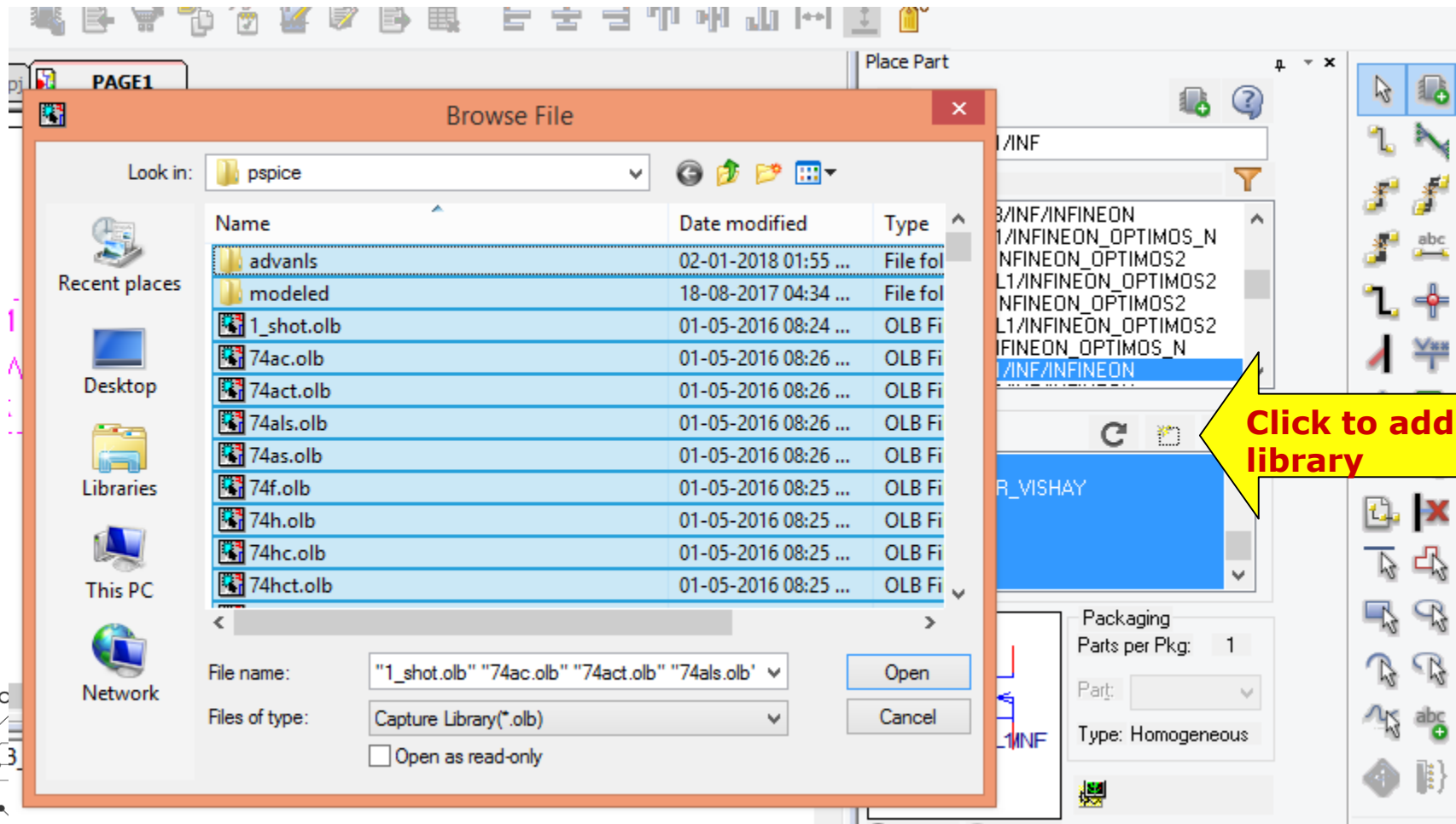
Creating Circuit Schematic...



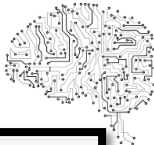
**Click at
Place Part**

Adding Library

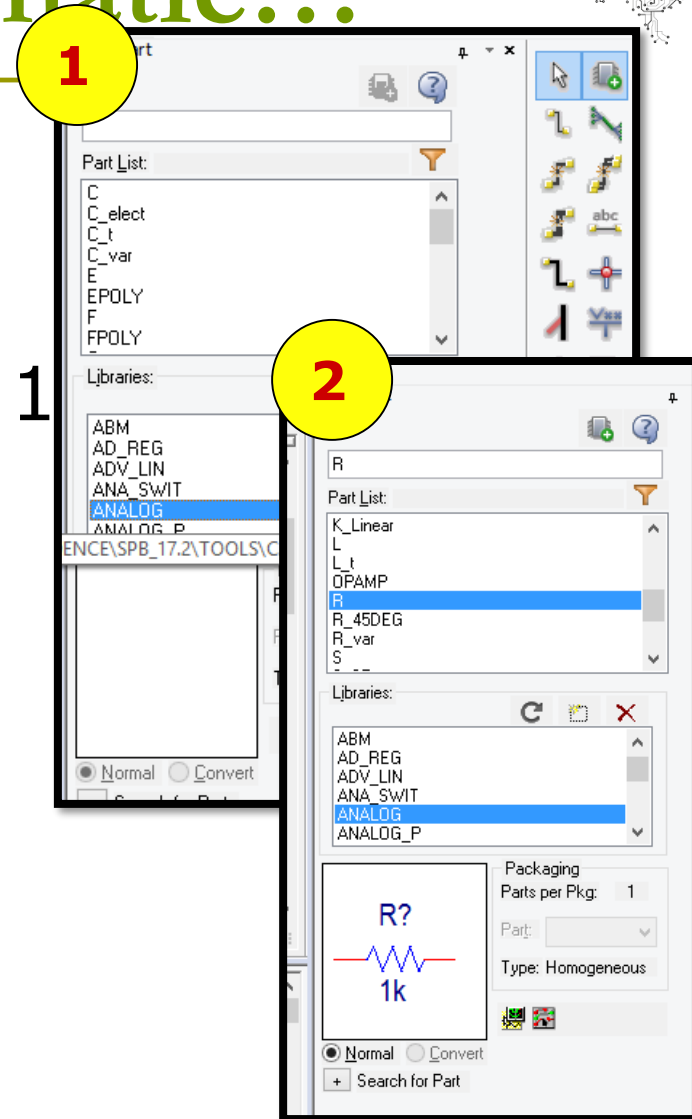
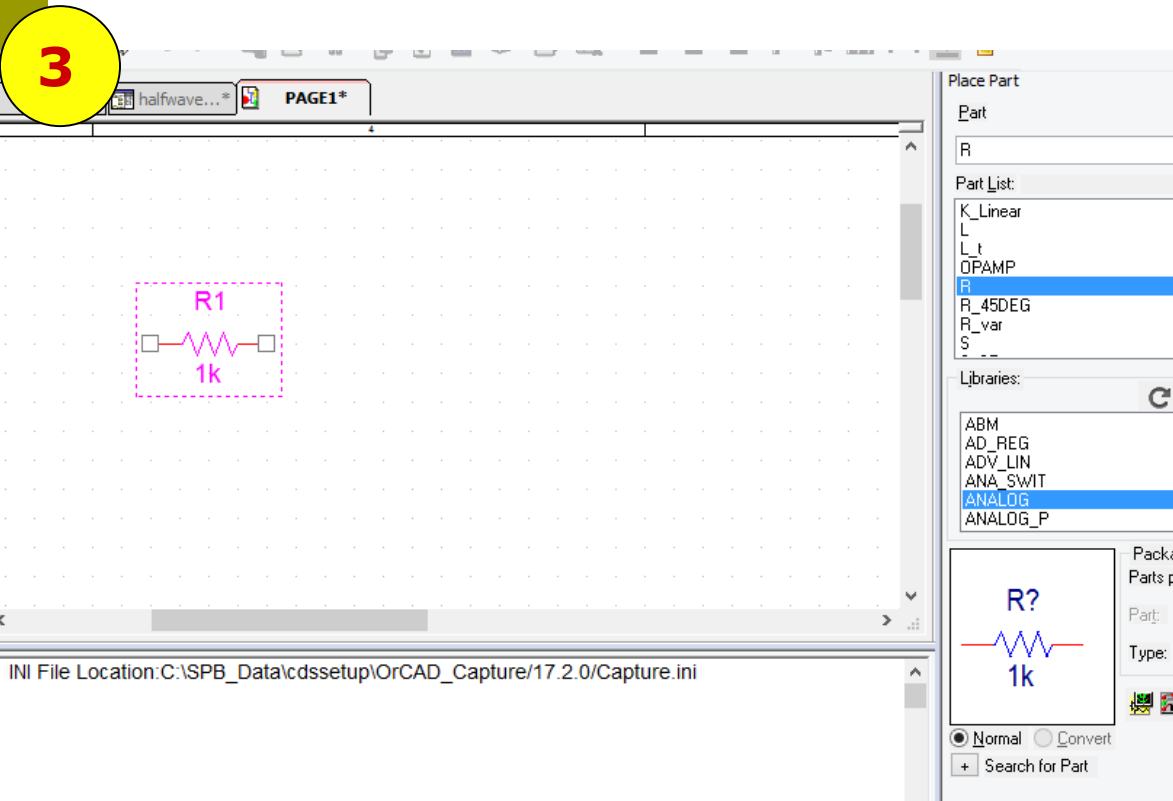
- Ctrl+A to select all libraries -> open



Creating Circuit Schematic...

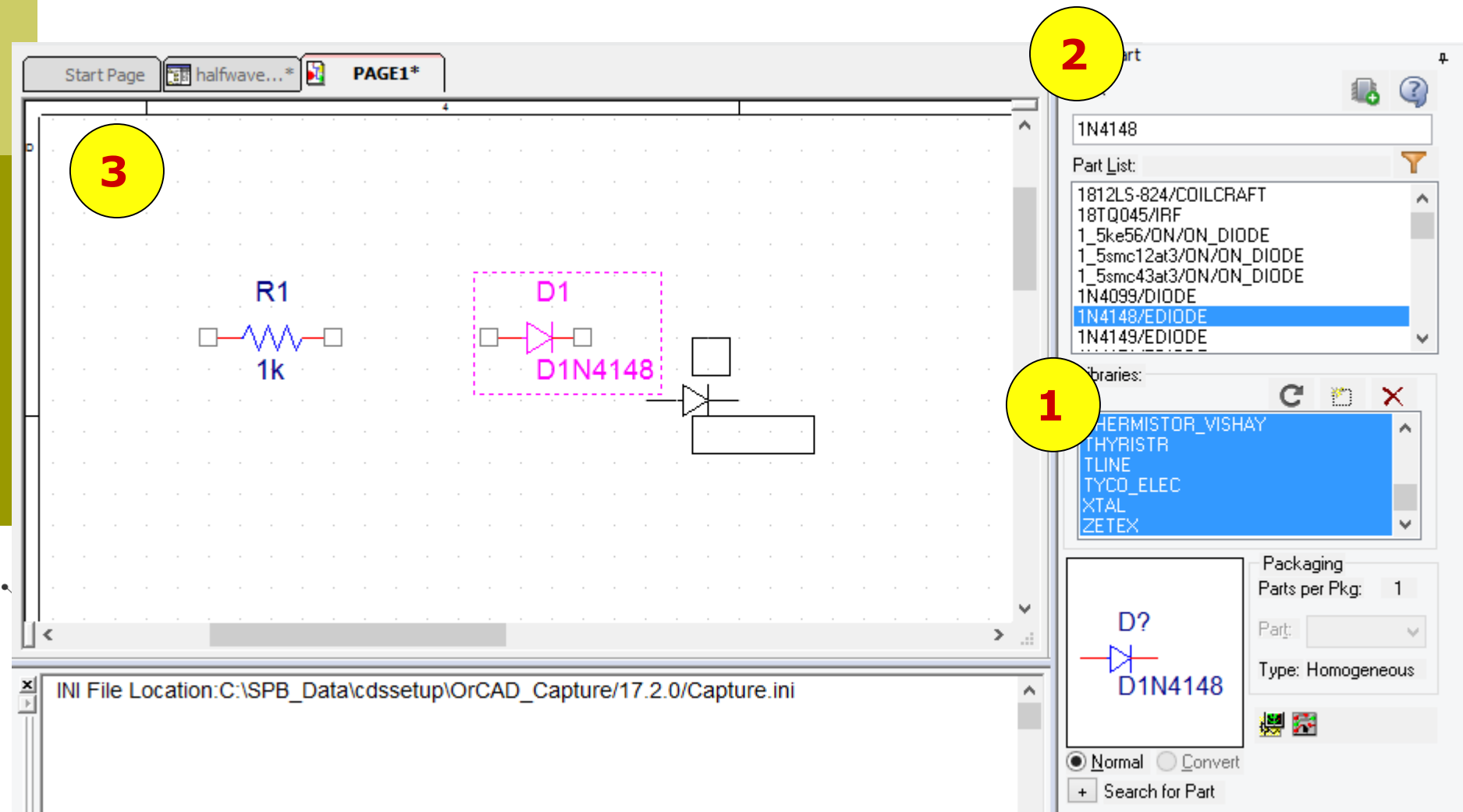


1. Select analog library
2. Double click R in Part List.
3. Resistor is drawn on page 1



Creating Circuit Schematic...

- Adding diode { Tip: press Esc to end }



The screenshot displays a circuit schematic editor interface. The main workspace shows a circuit with a resistor labeled R1 (1k) and a diode labeled D1 (1N4148). The diode is highlighted with a pink dashed box. The right panel shows the part list with 1N4148/EDIODE selected. The bottom status bar shows the INI file location: C:\SPB_Data\cdssetup\OrCAD_Capture/17.2.0/Capture.ini.

3 (Yellow circle) points to the main workspace area.

2 (Yellow circle) points to the part list on the right.

1 (Yellow circle) points to the libraries list on the right.

Part List:

- 1812LS-824/COILCRAFT
- 18TQ045/IRF
- 1_5ke56/ON/ON_DIODE
- 1_5smc12at3/ON/ON_DIODE
- 1_5smc43at3/ON/ON_DIODE
- 1N4099/DIODE
- 1N4148/EDIODE
- 1N4149/EDIODE

Libraries:

- HERMISTOR_VISHAY
- THYRISTR
- TLINE
- TYCO_ELEC
- XTAL
- ZETEX

Part: D1N4148

Package: 1

Type: Homogeneous

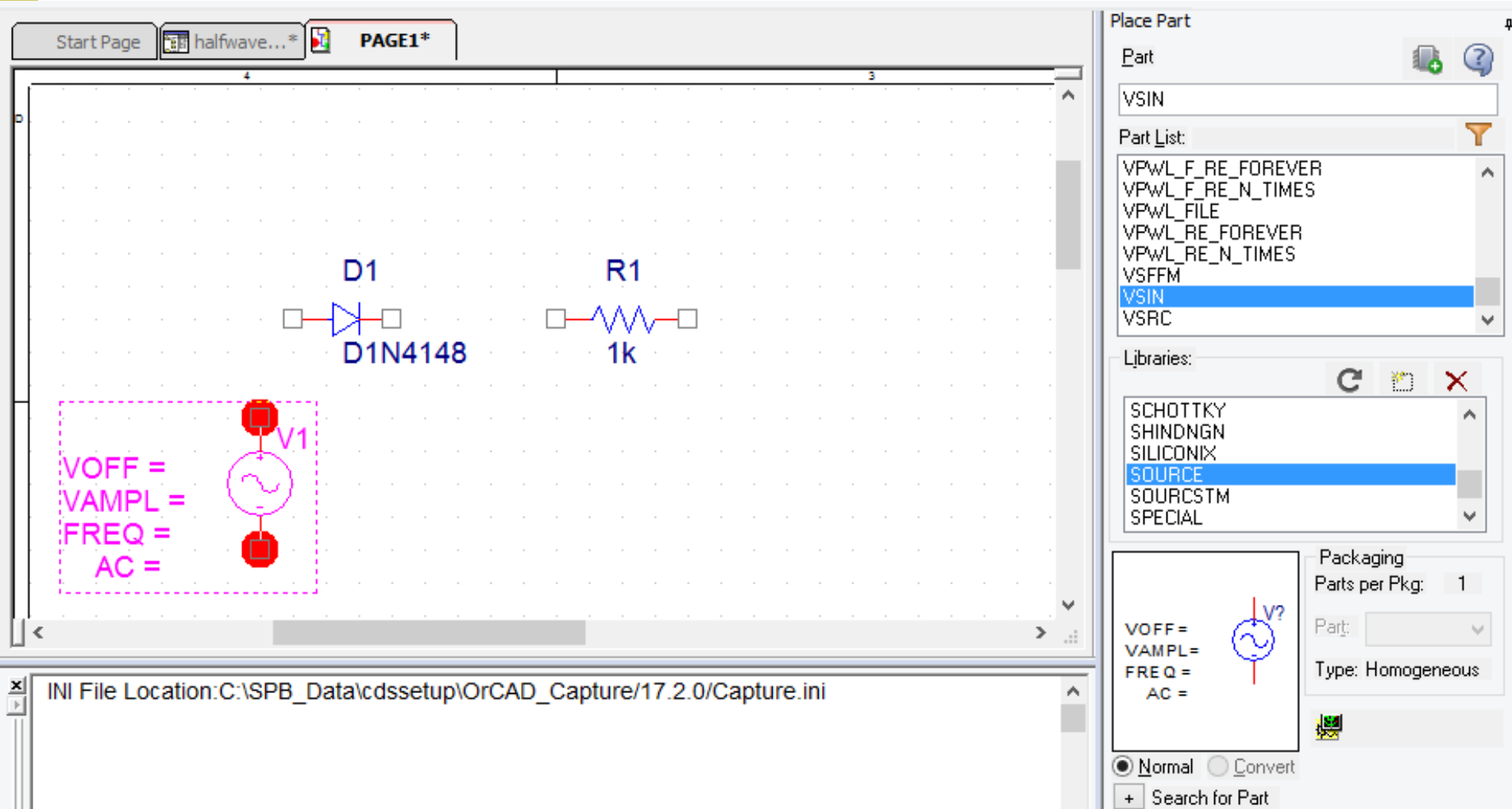
Normal Convert

Search for Part

INI File Location: C:\SPB_Data\cdssetup\OrCAD_Capture/17.2.0/Capture.ini

Creating Circuit Schematic...

□ selecting input voltage source



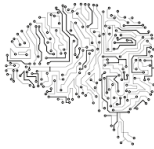
The screenshot displays the OrCAD Capture interface. The main workspace shows a circuit schematic with a diode (D1, D1N4148) and a resistor (R1, 1k). A dashed pink box highlights a voltage source component (V1) with the following parameters:

- VOFF =
- VAMPL =
- FREQ =
- AC =

The 'Place Part' dialog is open on the right side. The 'Part' field contains 'VSIN'. The 'Part List' shows a scrollable list of components, with 'VSIN' selected. The 'Libraries' section shows a list of libraries, with 'SOURCE' selected. The 'Packaging' section shows 'Parts per Pkg: 1'. The 'Part' field is empty, and the 'Type' is 'Homogeneous'. The 'Normal' radio button is selected, and the 'Convert' radio button is unselected. The 'Search for Part' button is visible at the bottom.

INI File Location: C:\SPB_Data\cdssetup\OrCAD_Capture\17.2.0\Capture.ini

Creating Circuit Schematic...

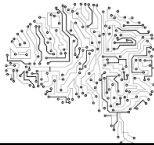


Place wire

The screenshot displays the OrCAD Capture software interface. The main workspace shows a circuit schematic with a voltage source (V1), a diode (D1, D1N4148), and a resistor (R1, 1k). The voltage source is labeled with parameters: VOFF =, VAMPL =, FREQ =, and AC =. A tooltip for the voltage source shows the label [V1/+ Number:1].

The 'Place Part' dialog box is open on the right side of the screen. It shows the 'Part' list with 'VSIN' selected. The 'Libraries' list shows 'SOURCE' selected. The 'Packaging' section shows 'Parts per Pkg: 1' and 'Type: Homogeneous'. The 'Normal' radio button is selected under the 'Normal' and 'Convert' options. A yellow arrow points to the 'Click' button in the 'Place Part' dialog box.

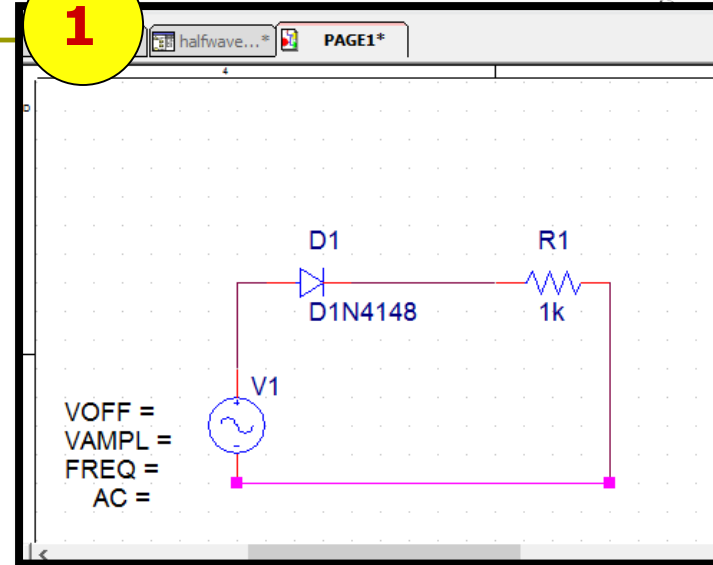
At the bottom of the window, the 'INI File Location' is displayed as: C:\SPB_Data\cdssetup\OrCAD_Capture\17.2.0\Capture.ini



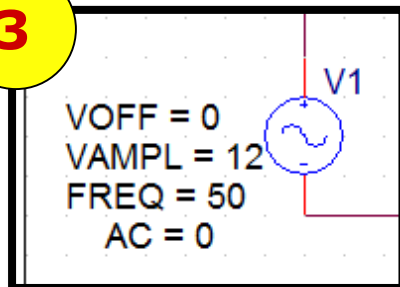
Creating Circuit Schematic...

Editing input source

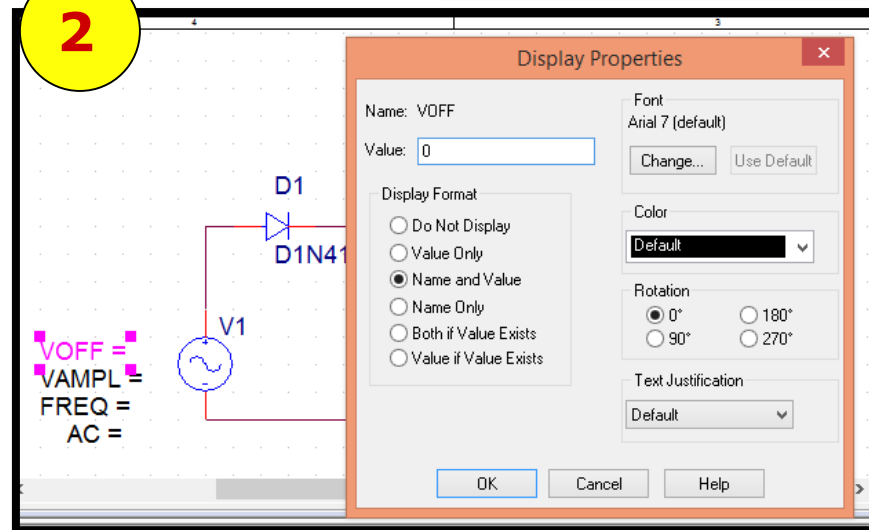
1



3



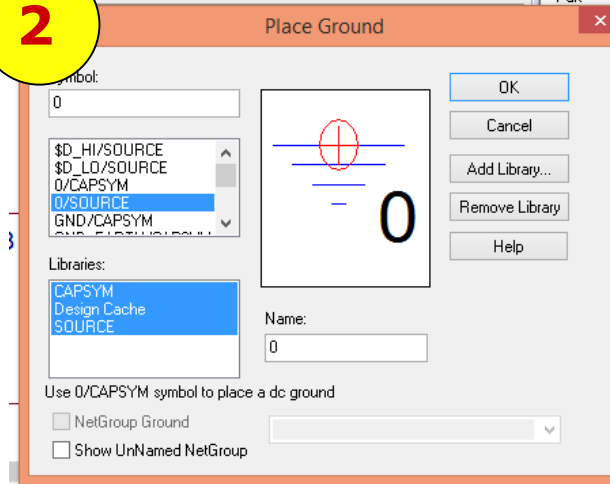
2



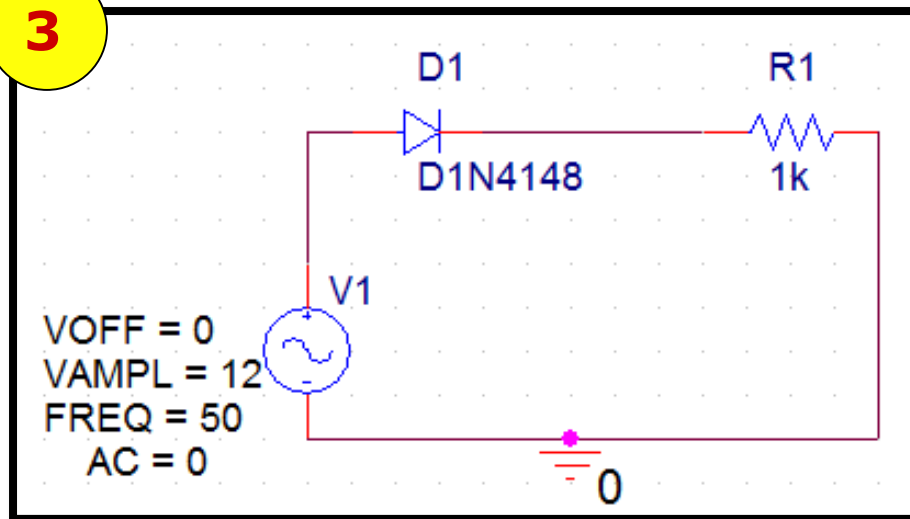
Creating Circuit Schematic...

1. Select ground tool
2. Place Ground
3. Final Half wave circuit

2



3

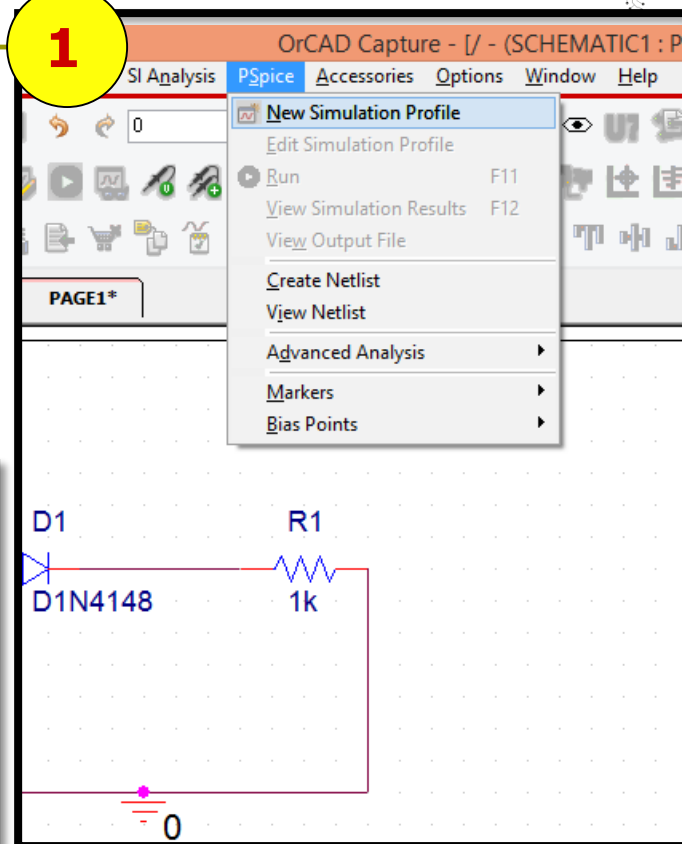


Circuit Simulation

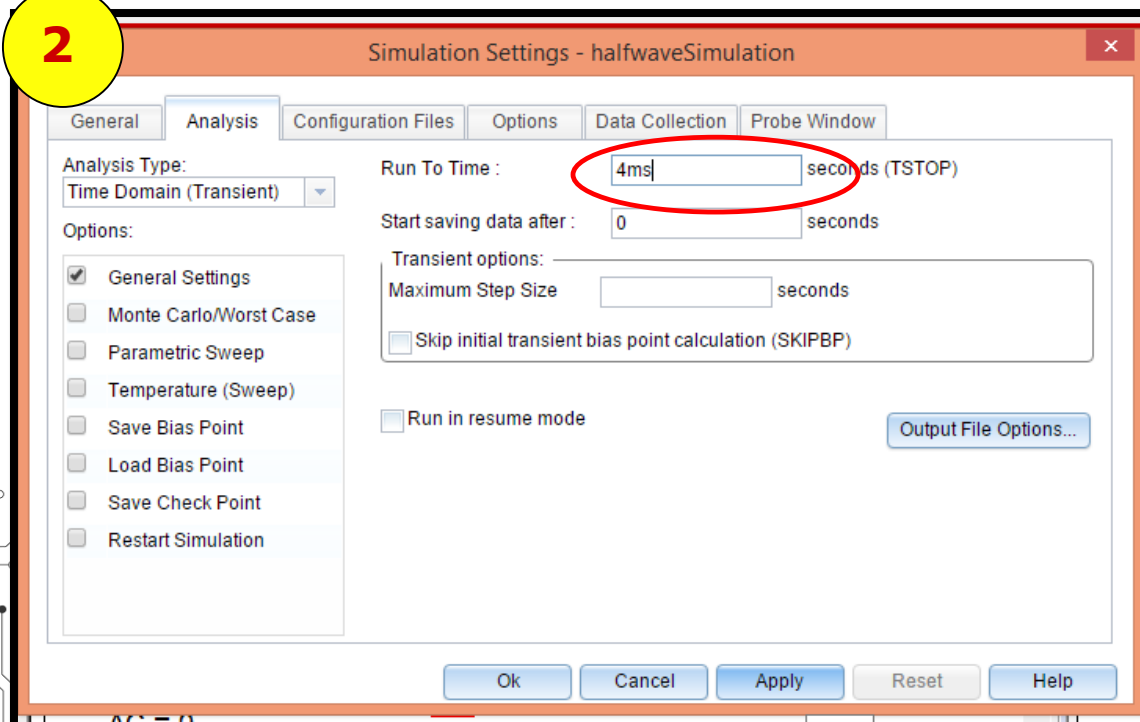


□ Creating new simulation profile

1



2



Circuit Simulation

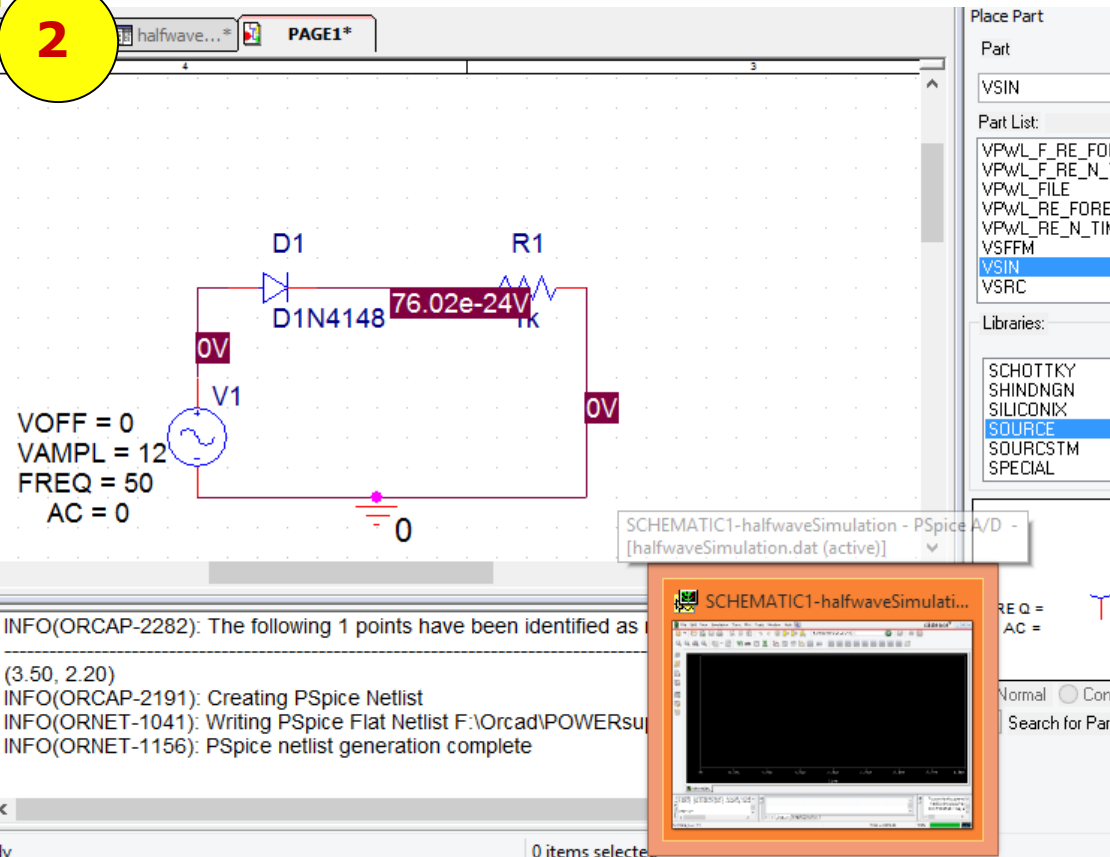
1

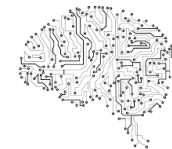
OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]

Design Edit View Tools Place SI Analysis PSpice Accessories Options Window Help



2





□ Add Probe

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

File Design Edit View Tools Place SI Analysis PSpice Accessories Options Window Help

SCHEMATIC1-halfwav

Start Page halfwave...* PAGE1*

VOFF = 0
VAMPL = 12
FREQ = 50
AC = 0

D1
D1N4148
R1
76.02e-24V
V1
0V
0

Place Part

Part

VSIN

Part List:

VPWL_F_RE_FC
VPWL_F_RE_N
VPWL_FILE
VPWL_RE_FOR
VPWL_RE_N_T
VSFFM
VSIN
VSRC

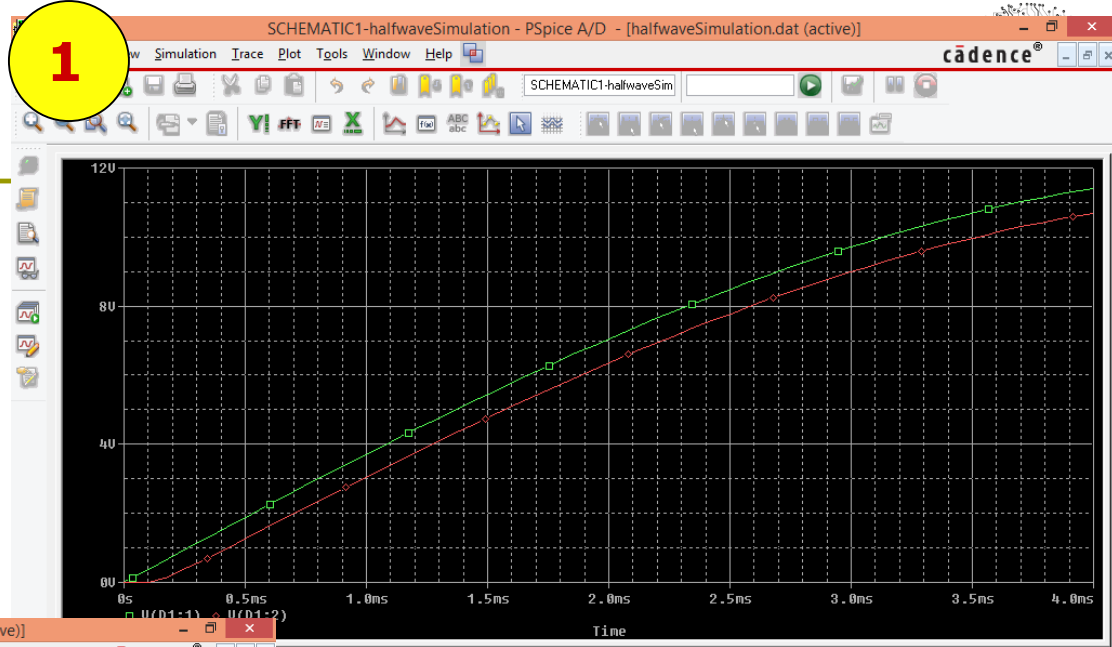
Libraries:

SCHOTTKY
SHINDNGN
SILICONIX
SOURCE
SOURCSTM
SPECIAL

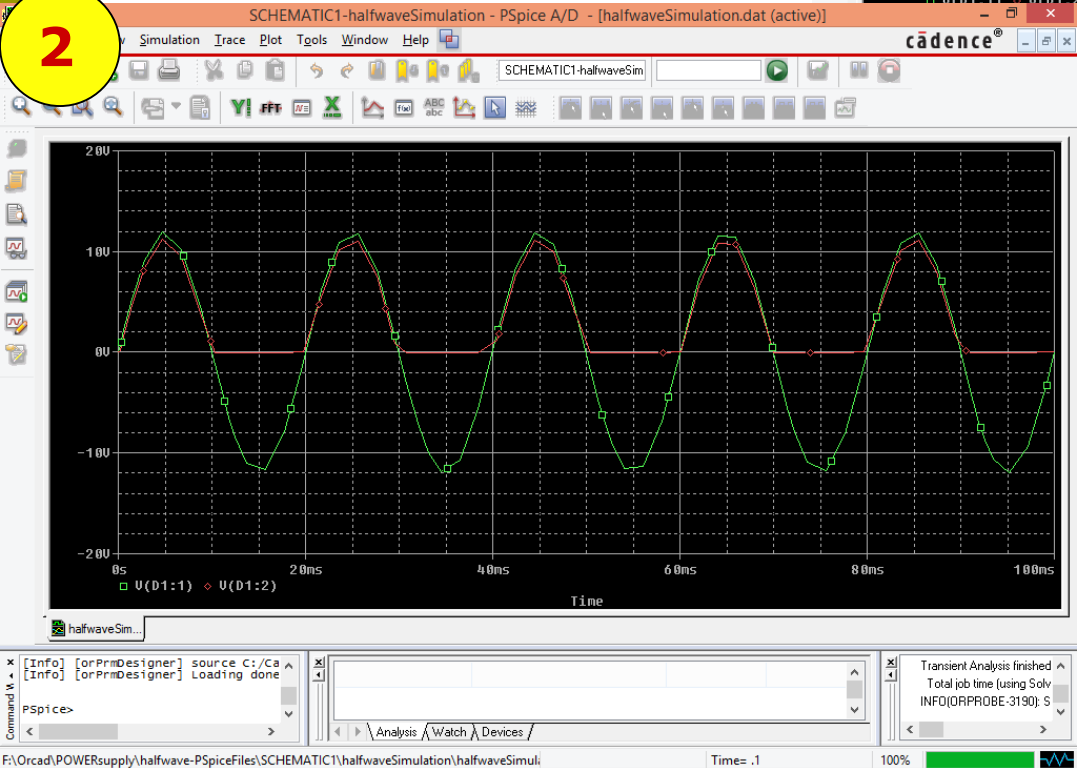
Simulation

- Edit time to 100ms

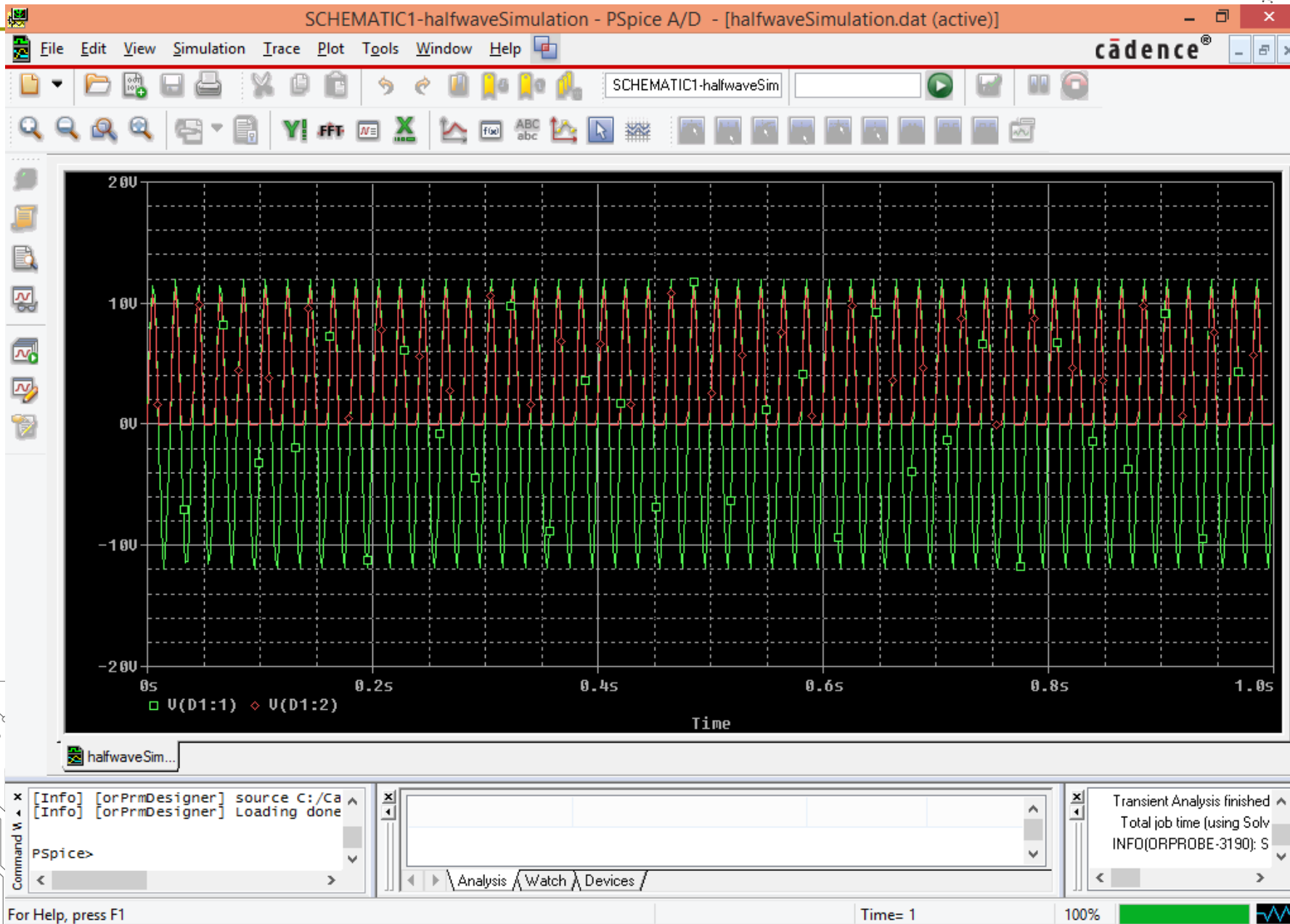
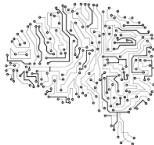
1



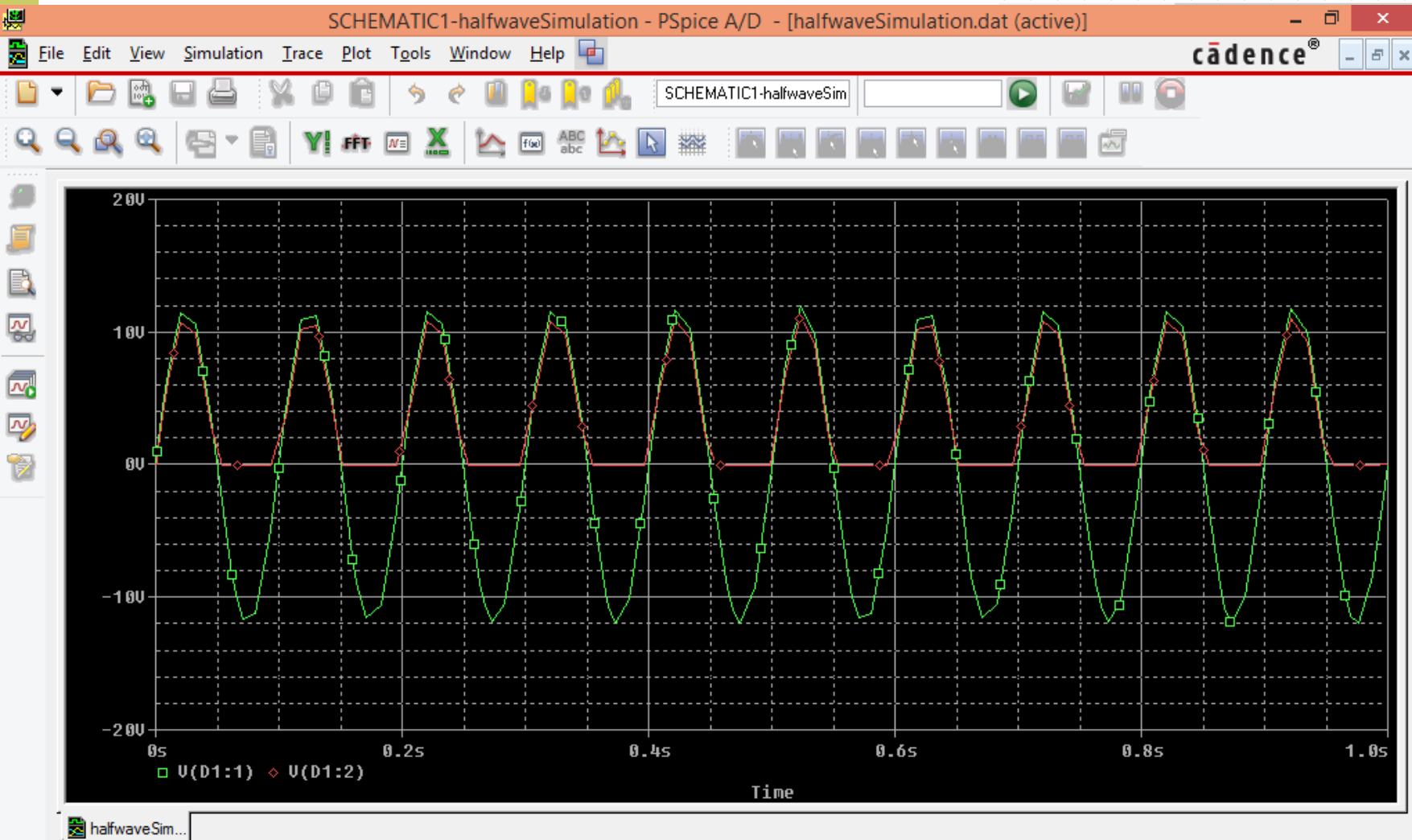
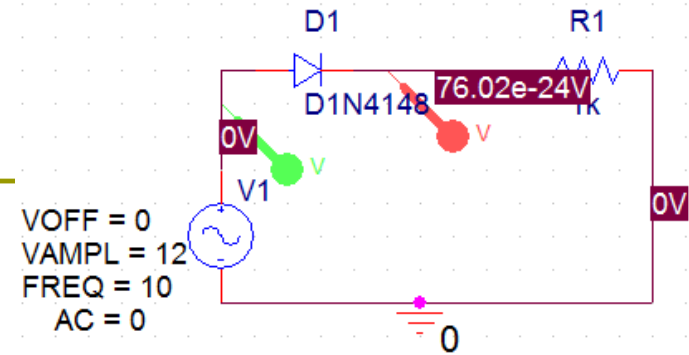
2



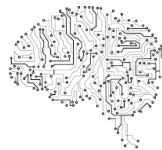
□ Freq=50



■ Freq=10



Run to Time: 2000 ms



1

Simulation Settings - halfwaveSimulation

Analysis Configuration Files Options Data Collection Probe Window

Analysis Type: Time Domain (Transient) Run To Time: 2000ms seconds (TSTOP)

Options: Start saving data after: 0 seconds

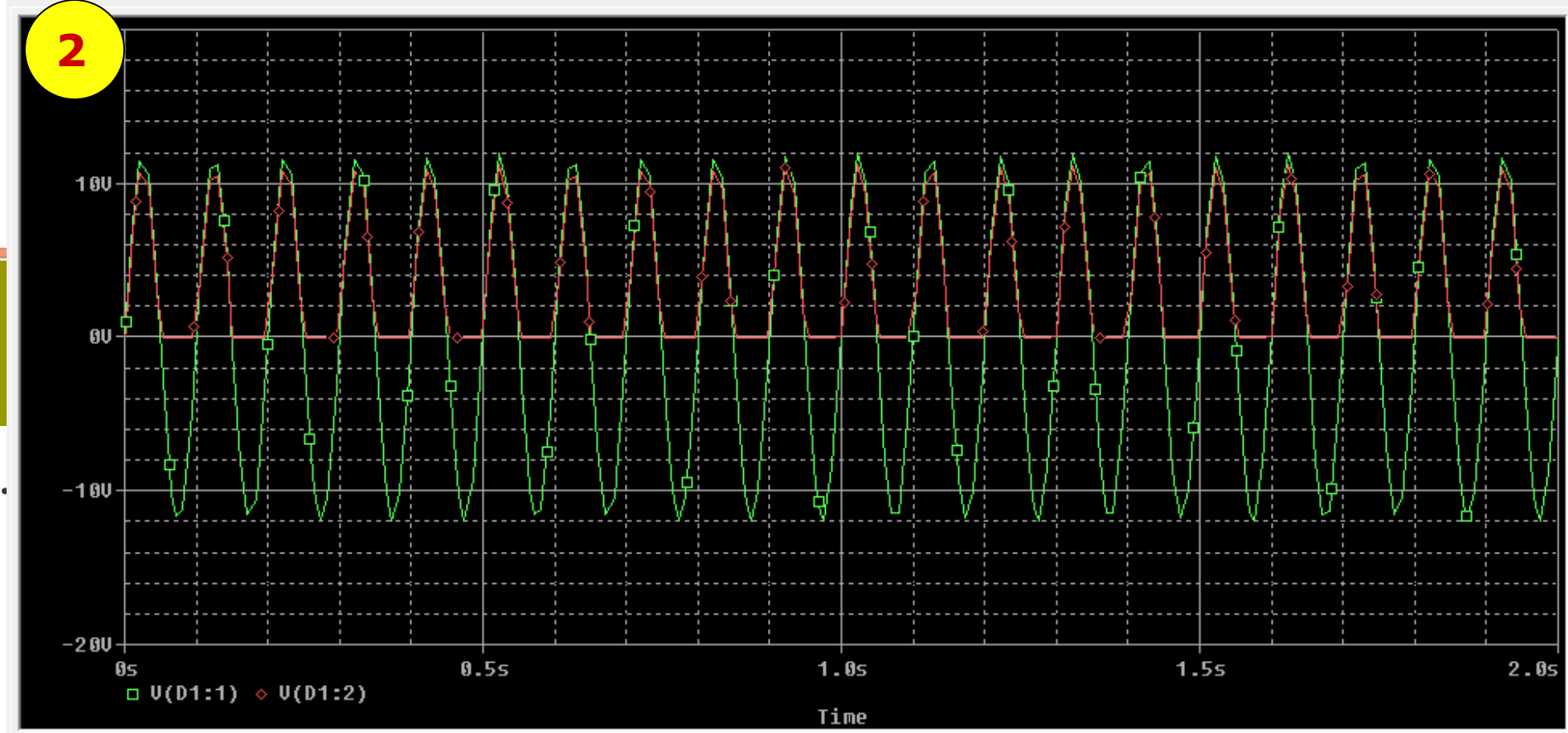
Transient options:

- ☒ General Settings
- ☐ Monte Carlo/Worst Case
- ☐ Parametric Sweep
- ☐ Temperature (Sweep)

Maximum Step Size: seconds

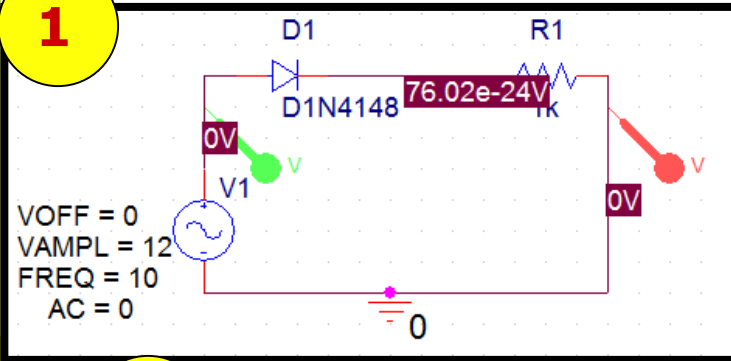
☐ Skip initial transient bias point calculation (SKIPBP)

2

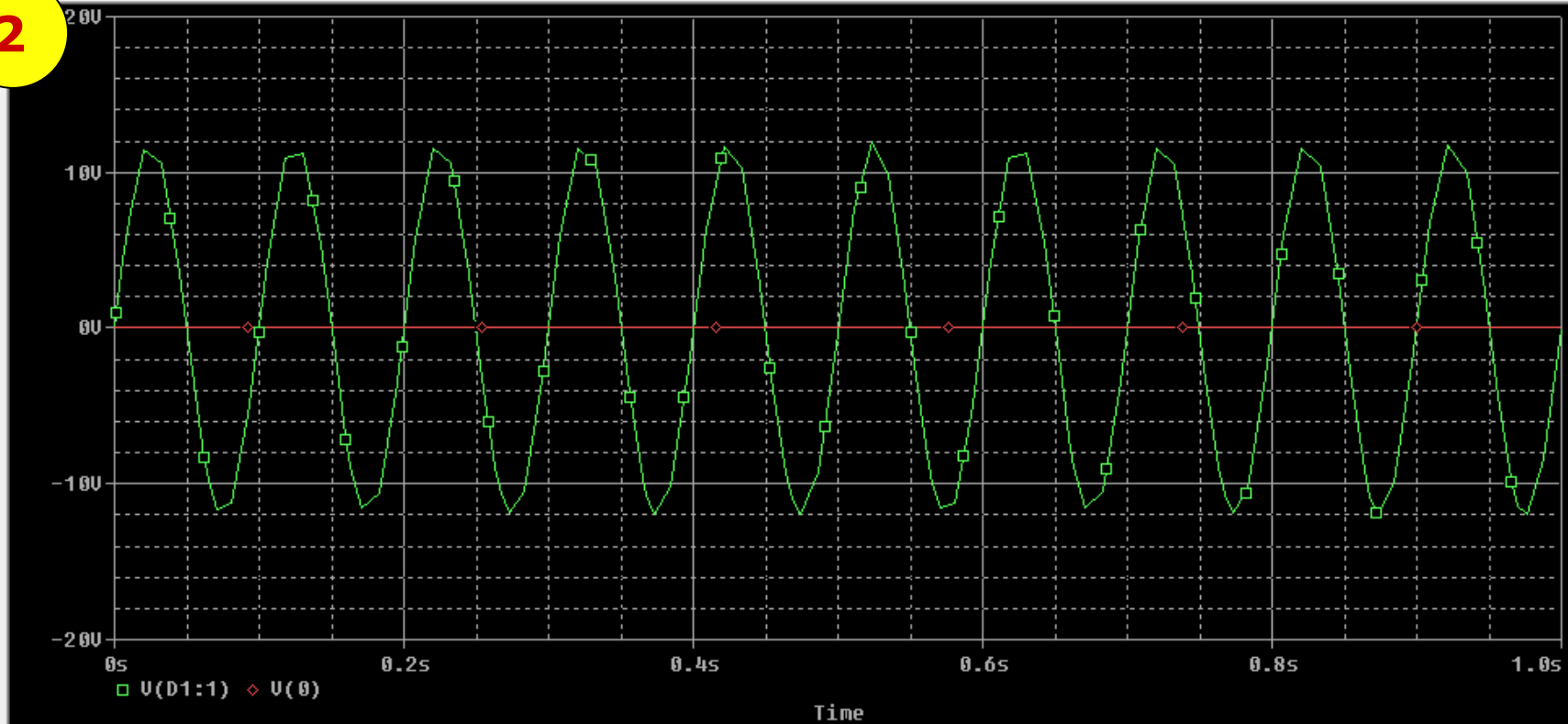


Changing probe position

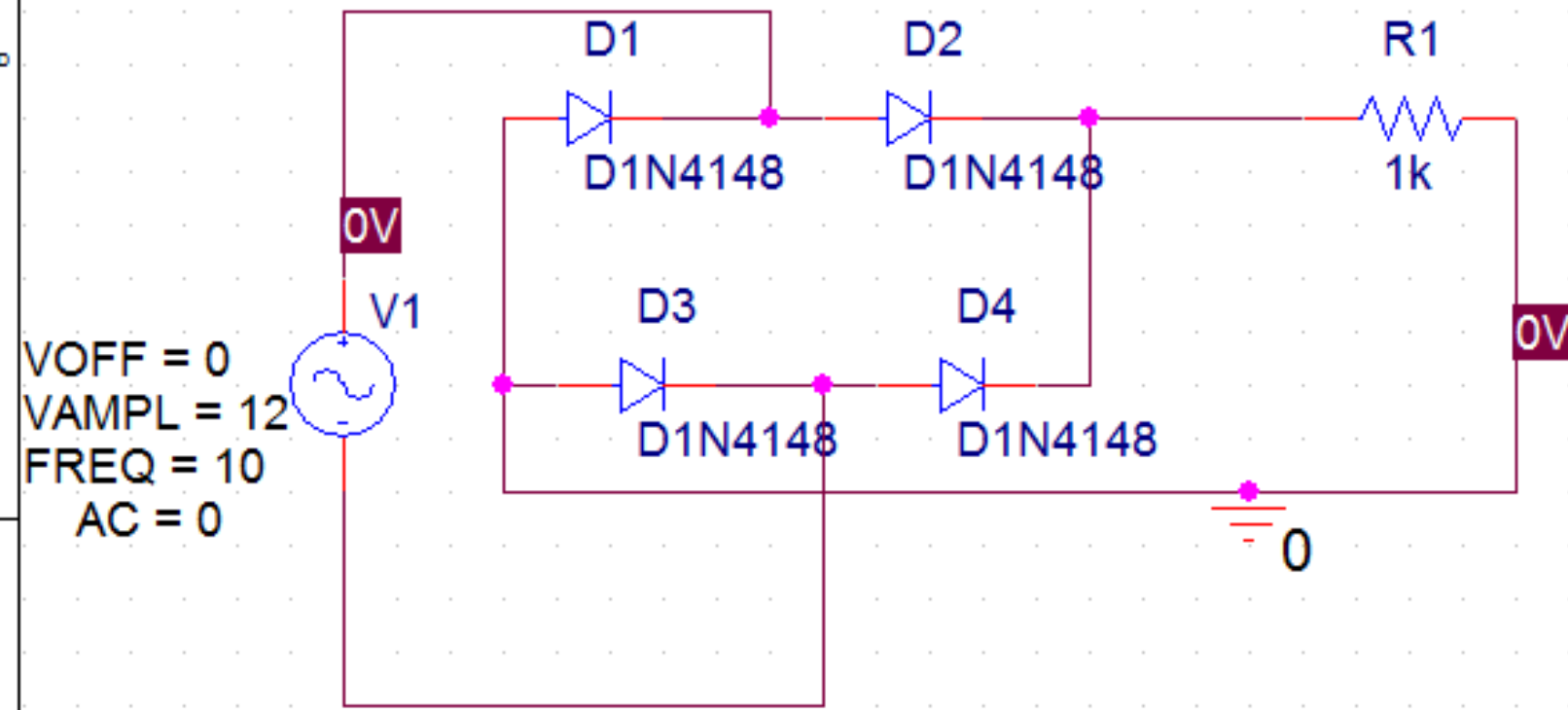
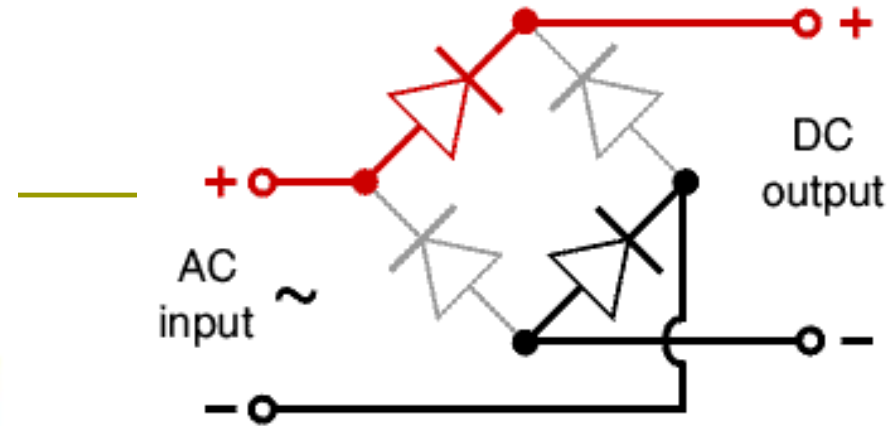
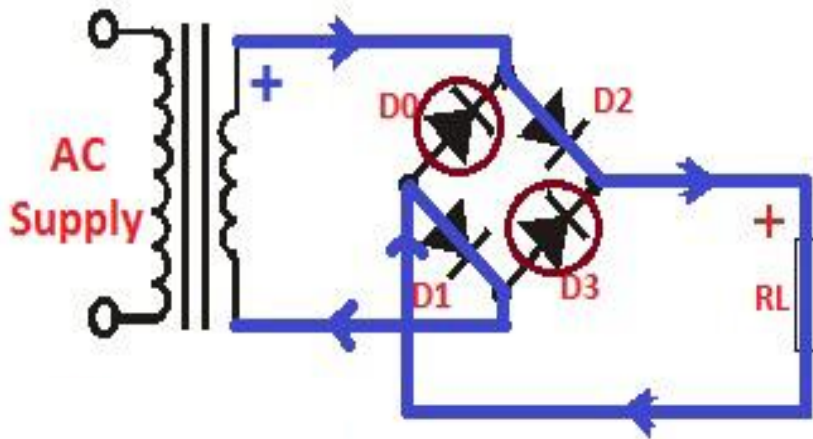
1



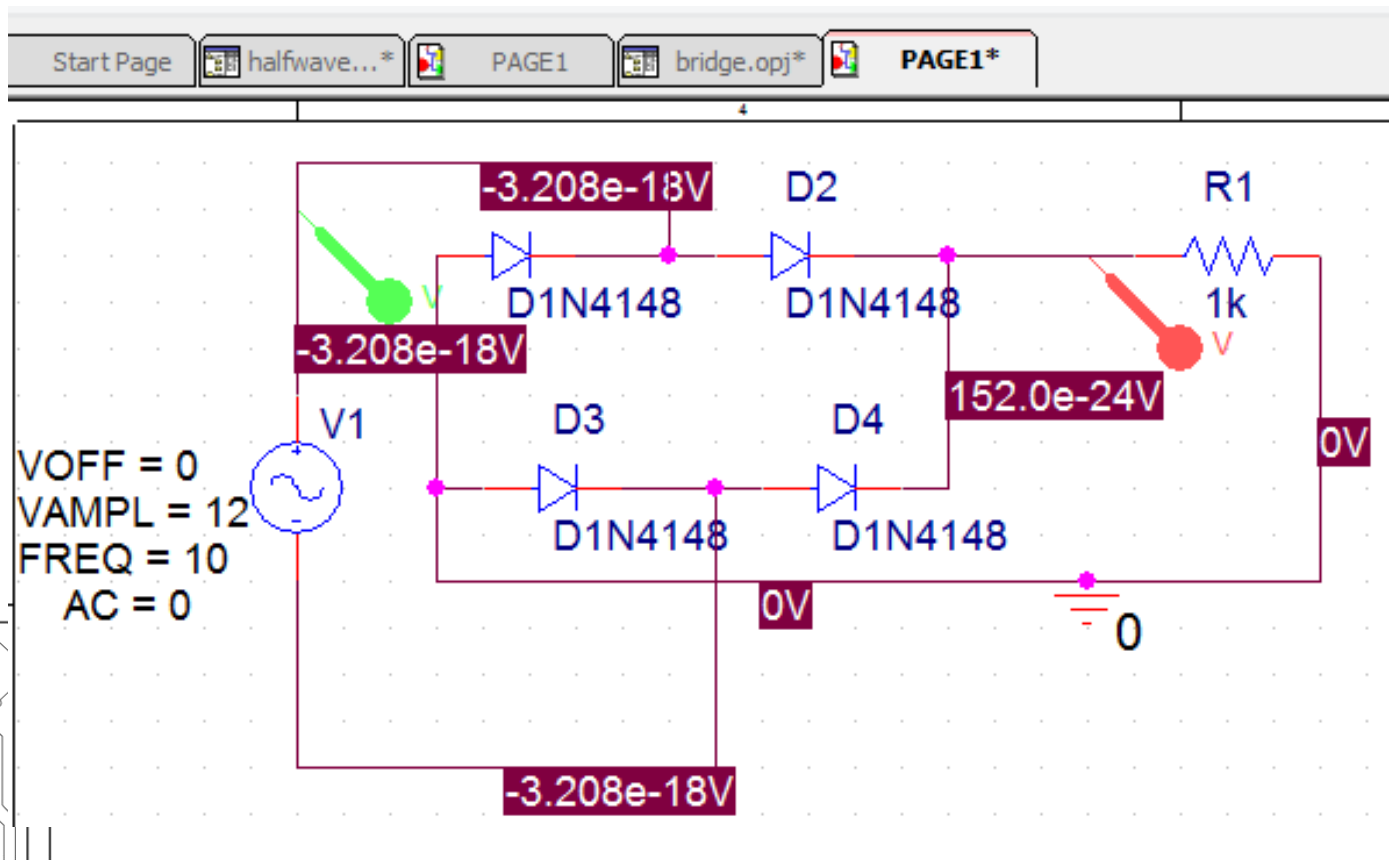
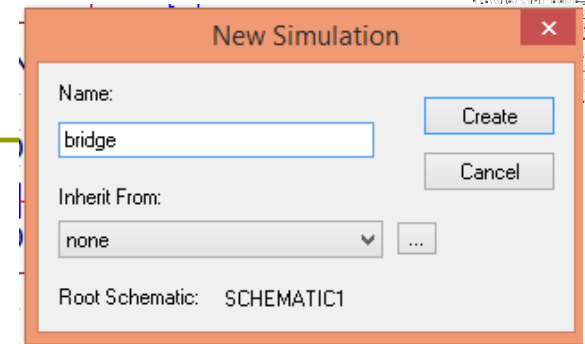
2

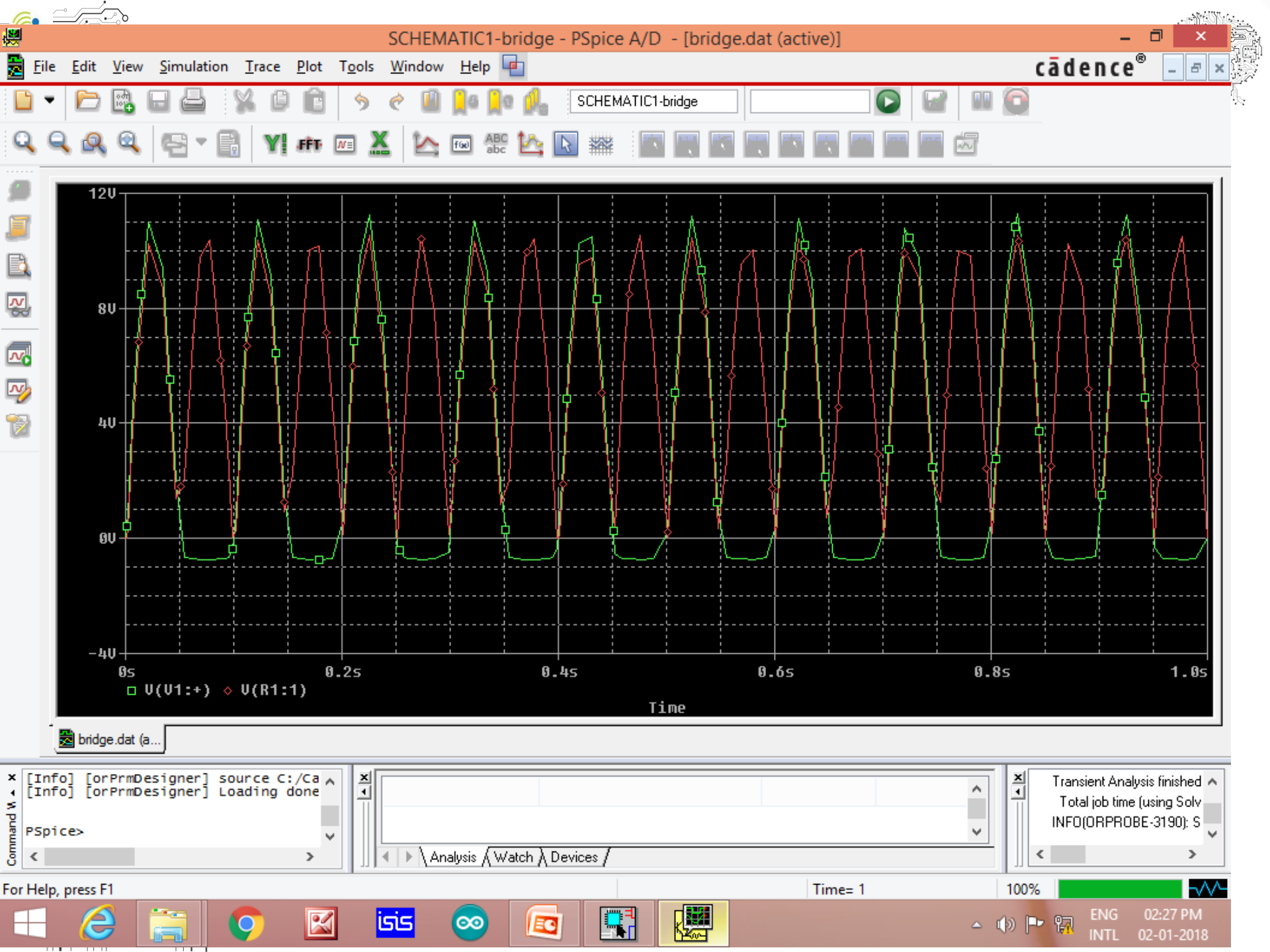


Bridge Rectifier



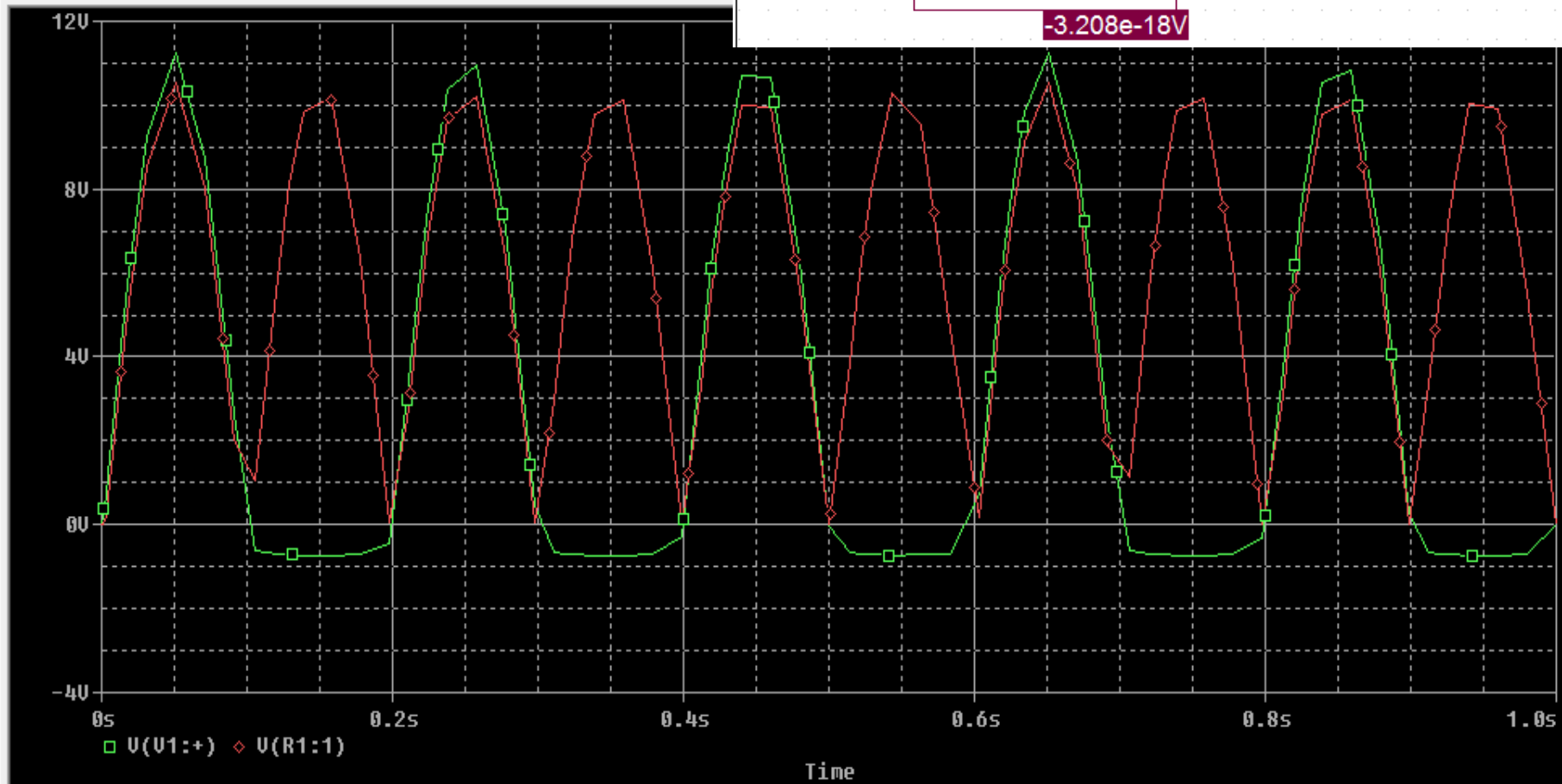
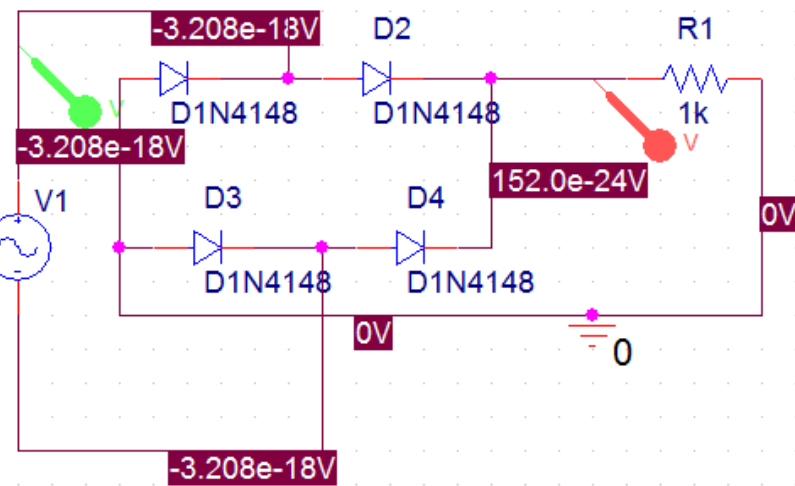
Simulation





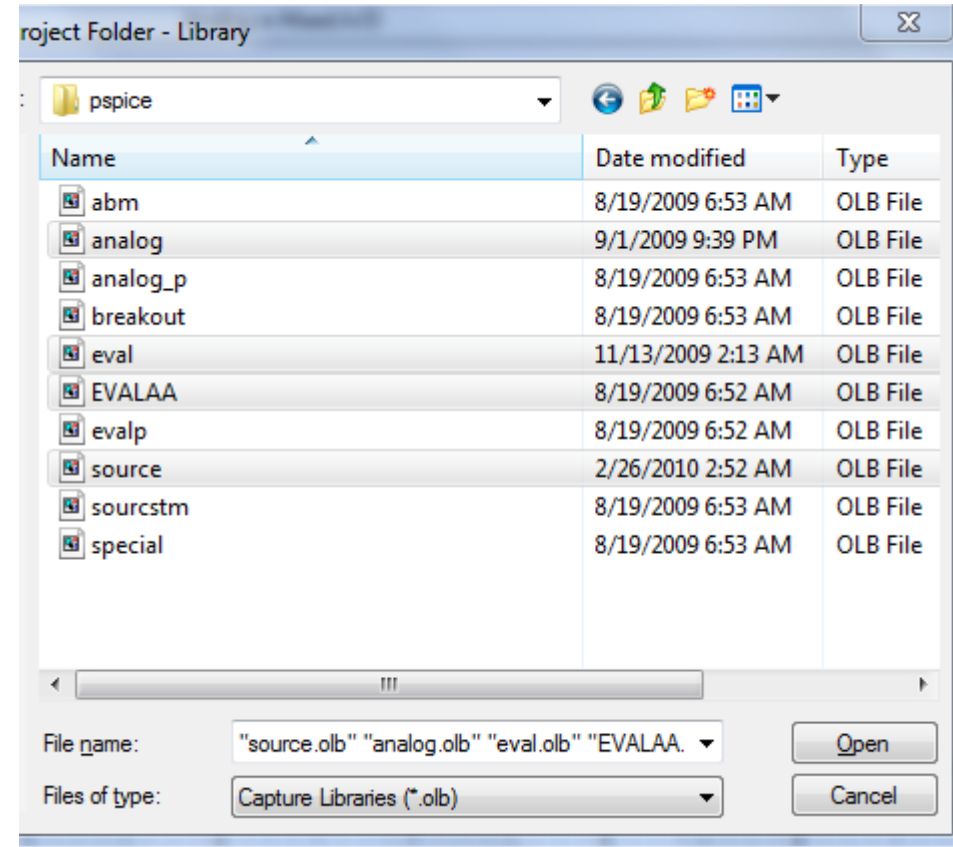
□ Freq = 5

VOFF = 0
VAMPL = 12
FREQ = 10
AC = 0



Adding Library – (1)

- ❑ Opening the Project Page (demo), you'll find a library folder there
- ❑ Right click and choose "Add File"
- ❑ New pop up window opens capture folder
- ❑ Browse: library => PSpice
- ❑ Select: "analog", "eval", "EVALAA" and "source" using Ctrl key and click open



Adding Library – (2)

- Now the added libraries are displayed in the Library Folder
- Use Ctrl + S to save the settings and click on the “Page 1” tab to begin with your circuit schematic

