



ΕΘΝΙΚΟ ΜΕΤΣΟΒΙΟ ΠΟΛΥΤΕΧΝΕΙΟ

ΣΧΟΛΗ ΗΛΕΚΤΡΟΛΟΓΩΝ ΜΗΧΑΝΙΚΩΝ & Μ/Υ

Εργαστήριο Ηλεκτρονικής

Orcad PSpice

Γεώργιος Βιτζηλαίος

Αθήνα 2008




ΕΓΚΑΤΑΣΤΑΣΗ



Ηλεκτρονική Ι - Microsoft Internet Explorer

File Edit View Favorites Tools Help

Back Forward Stop Home Search Favorites Refresh Print Mail News RSS Feeds

Address <http://www.elab.ntua.gr/hlektron2.htm> Go Links

 **Εθνικό Μετσόβιο Πολυτεχνείο**
Τμήμα Ηλεκτρολόγων Μηχανικών & Μηχανικών Υπολογιστών
Ομάδα Σχεδίασης Μικροηλεκτρονικών Κυκλωμάτων

ΕΡΓΑΣΤΗΡΙΑΚΗ ΚΑΙ ΒΙΟΜΗΧΑΝΙΚΗ ΗΛΕΚΤΡΟΝΙΚΗ

5ο εξάμηνο ΗΜΜΥ
Χειμερινό εξάμηνο 2005


Διδάσκων: [Γιάννης Παπανάνος](#), Καθηγητής ΕΜΠ


Ηλεκτρονική διεύθυνση: papan@elab.ntua.gr




Παρουσίαση του Orcad PSpice

Orcad Pspice Student Version

*PSpice, evaluation copy.
Electronic circuit simulator. Used in the Electronics
Laboratory course, 5th semester.*

 [Get Pspice 8.0 or 9.1 from the central FTP server](#)

 [Επικοινωνήστε με την Ομάδα](#)

 [Αρχική Σελίδα](#)
 [Τμήμα ΗΜ & ΜΥ](#)
 [ΕΜΠ](#)

ftp://ftp.ntua.gr/pub/pc/Pspice/ Internet

Start Ηλεκτρονική Ι - Micro... 12:19 PM

ΕΓΚΑΤΑΣΤΑΣΗ



ftp://ftp.ntua.gr/pub/pc/Pspice/ - Microsoft Internet Explorer

File Edit View Favorites Tools Help

Back Search Folders

Address ftp://ftp.ntua.gr/pub/pc/Pspice/ Go Links

Other Places

- pc
- My Documents
- Shared Documents
- My Network Places

Details

Student_versi on_8_0 Student index.html

User: Anonymous Internet

Start ftp://ftp.ntua.gr/pub... pspicee 1.GIF - Paint 12:20 PM



ΕΓΚΑΤΑΣΤΑΣΗ

The screenshot shows a Windows XP file explorer window titled 'Student_version_9_1'. The address bar shows the path 'C:\Documents and Settings\Administrator\Desktop\Student_version_9_1'. The window displays a grid of files:

- 91PSPSTU.EXE (1,120 KB) - A red arrow points to this file.
- DISK1.ZIP (1,406 KB)
- DISK2.ZIP (1,406 KB)
- DISK3.ZIP (1,409 KB)
- DISK4.ZIP (1,408 KB)
- DISK5.ZIP (1,409 KB)
- DISK6.ZIP (1,413 KB)
- DISK7.ZIP (1,413 KB)
- DISK8.ZIP (1,410 KB)
- DISK9.ZIP (1,407 KB)
- DISK10.ZIP (1,414 KB)
- DISK11.ZIP (1,406 KB)
- DISK12.ZIP (1,409 KB)
- DISK13.ZIP (1,408 KB)
- DISK14.ZIP (1,412 KB)
- DISK15.ZIP (1,408 KB)
- DISK16.ZIP (1,407 KB)
- DISK17.ZIP (1,412 KB)
- DISK18.ZIP (1,409 KB)
- DISK19.ZIP (1,410 KB)
- DISK20.ZIP (1,413 KB)
- DISK21.ZIP (39 KB)
- MIXED.PDF (20 KB) - Adobe Acrobat Document
- NETLICGD.PDF (809 KB) - Adobe Acrobat Document
- OPTUG.PDF (5,169 KB) - Adobe Acrobat Document
- PCB2LAY.PDF (815 KB) - Adobe Acrobat Document
- PSPICREF.PDF (3,563 KB) - Adobe Acrobat Document
- PSPICE.PDF (10,482 KB) - Adobe Acrobat Document
- PSPICEAD.PDF (14,101 KB) - Adobe Acrobat Document
- SCH2CAP.PDF (252 KB) - Adobe Acrobat Document
- ANALOG.PDF (663 KB) - Adobe Acrobat Document
- CAPUG.PDF (7,739 KB) - Adobe Acrobat Document
- DIGITAL.PDF (114 KB) - Adobe Acrobat Document

The taskbar at the bottom shows the Start button, several open applications (Education - Microsoft Int..., pspicee, Student_version_9_1, 2.GIF - Paint), and the system tray with the date and time (12:21 PM).

ΕΓΚΑΤΑΣΤΑΣΗ



The screenshot shows a Windows XP file explorer window titled 'orcad'. The address bar indicates the path: C:\Documents and Settings\Administrator\Desktop\orcad. The window displays a grid of files and folders. A red arrow points to the 'Setup.exe' file, which is a 'Setup Launcher' from 'InstallShield Software Corpora...'. Other files include '_ISDel.exe', '_Setup.dll', '_sys1.cab', '_sys1.hdr', '_user1.cab', '_user1.hdr', 'DATA.TAG', 'data1.cab', 'setup.lid', '_INST321.EX_', 'data1.hdr', 'lang.dat', 'layout.bin', 'os.dat', 'setup.bmp', and 'SETUP.INI'. The taskbar at the bottom shows several open applications: 'http://www.elab...', 'pspicee', 'Student_version_...', 'Capture', 'New Folder', 'orcad', and '3.GIF - Paint'. The system clock shows 12:26 PM.

ΕΓΚΑΤΑΣΤΑΣΗ



PSpice Student Setup

PSpice Student

Select Schematic Editors

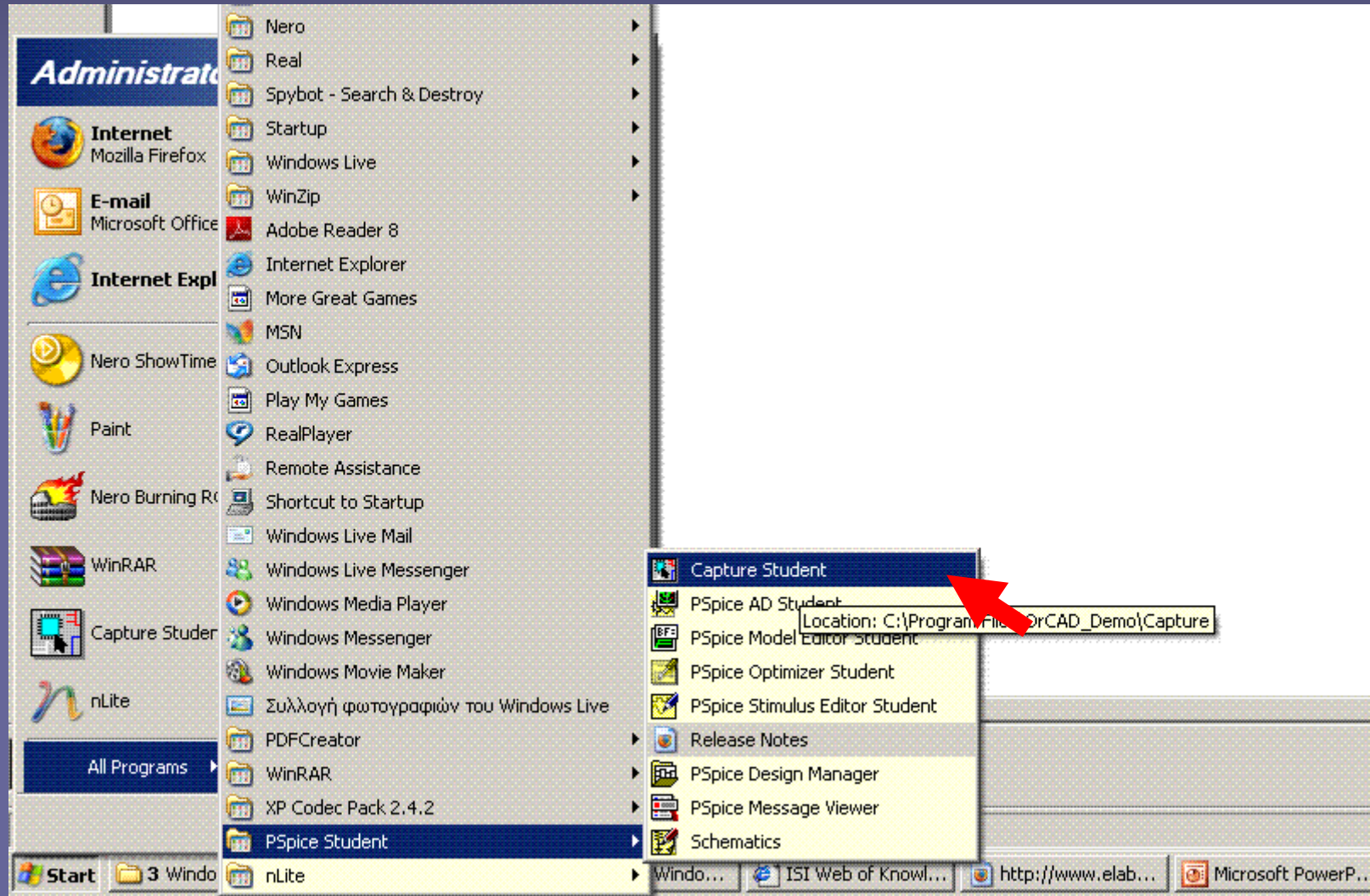
Choose the Schematic Editor(s) you would like to install.

- Capture
- Schematics

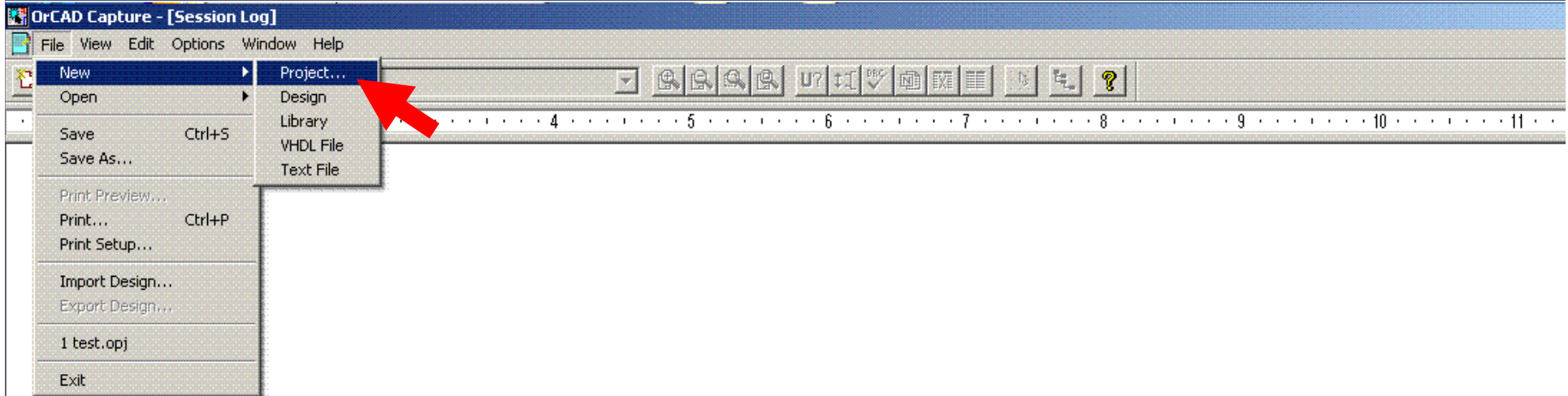
Once you have selected the appropriate installation option(s), click Next to continue. Click Cancel to exit.

< Back Next > Cancel

ΕΝΑΡΧΗ



ΑΡΧΙΚΕΣ ΡΥΘΜΙΣΕΙΣ



ΑΡΧΙΚΕΣ ΡΥΘΜΙΣΕΙΣ



DrCAD Capture - [Session Log]

File View Edit Options Window Help

Name: test

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:\Program Files\DrCAD_Demo

Ready | Session Log

Start | http://www.el... | pspicee | Student_versi... | Capture | New Folder | orcad | 6.GIF - Paint | DrCAD Captu... | 12:32 PM

ΑΡΧΙΚΕΣ ΡΥΘΜΙΣΕΙΣ



The screenshot shows the OrCAD Capture application window with a 'Create PSpice Project' dialog box open. The dialog box has two radio button options: 'Create based upon an existing project' (unselected) and 'Create a blank project' (selected). A red arrow points to the 'Create a blank project' option. The dialog box also includes a text field, a 'Browse...' button, and 'OK', 'Cancel', and 'Help' buttons. The background shows the OrCAD Capture interface with a menu bar, toolbar, and a grid.

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



The screenshot shows the OrCAD Capture software interface. The main workspace is a grid. Two red boxes highlight specific tools in the right-hand toolbar:

- The top box is labeled "Εισαγωγή Στοιχείων" (Place Part) and points to the "Place Part" tool icon.
- The bottom box is labeled "Εισαγωγή GND VCC" (Place Power) and points to the "Place Power" tool icon.

The software title bar reads "OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]". The menu bar includes "File", "Edit", "View", "Place", "Macro", "PSpice", "Accessories", "Options", "Window", and "Help". The status bar at the bottom shows "0 items selected", "Scale=100%", "X=6.60 Y=5.20", and the system tray with the time "12:34 PM".

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



The screenshot shows the OrCAD Capture interface with two dialog boxes open. The 'Place Part' dialog is on the left, and the 'Browse File' dialog is on the right. A red arrow points to the 'Add Library...' button in the 'Place Part' dialog. A green arrow points to the 'Open' button in the 'Browse File' dialog. A red box highlights the text 'Επιλογή Όλων' (Select All) in the 'Browse File' dialog. The 'Browse File' dialog shows a list of files in the 'Pspice' directory, including 'abm.olb', 'analog.olb', 'analog_p.olb', 'breakout.olb', 'eval.olb', 'source.olb', 'sourcstm.olb', and 'special.olb'. The 'File name' field contains the path to the selected file, and the 'Files of type' dropdown is set to 'Capture Library (*.olb)'. The 'Open as read-only' checkbox is unchecked.

Libraries:
Design Cache

Graphic:
 Normal
 Convert

Packaging:
Parts per Pkg: 1
Part:
Type:

File name: "abm.olb""analog.olb""analog_p.olb""breako
Files of type: Capture Library (*.olb)
 Open as read-only

Επιλογή Όλων

Title	Document Number	Rev
<Title>	<Doc>	<Rev Code>

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



The screenshot displays the OrCAD Capture interface with the 'Place Part' dialog box open. The dialog box contains the following elements:

- Part:** A text input field.
- Part List:** A list of parts including 2N1595/EVAL, 2N5444/EVAL, 54152A/EVAL, 555D/EVAL, 7400/EVAL, 7401/EVAL, 7402/EVAL, 7403/EVAL, 7404/EVAL, and 7405/EVAL.
- Libraries:** A list of libraries including ABM, ANALOG, ANALOG_P, BREAKOUT, Design Cache, EVAL, SOURCE, SOURCSTM, and SPECIAL.
- Graphic:** Radio buttons for 'Normal' and 'Convert'.
- Packaging:** A section with 'Parts per Pkg: 1', a 'Part:' dropdown menu, and a 'Type:' field.
- Buttons:** OK, Cancel, Add Library..., Remove Library, Part Search..., and Help.

At the bottom right of the dialog, there is a table with the following structure:

Title		
Size	Document Number	Rev
A	<Doc>	<Rev Code>

The Windows taskbar at the bottom shows the Start button and several open applications: http://www.el..., pspicee, Student_versi..., Capture, New Folder, orcad, 10.GIF - Paint, OrCAD Captu..., and the system clock showing 12:37 PM.

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



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

File Edit View Place Macro PSpice Accessories Options Window Help

Part:

Part List:

- F
- FPOLY
- G
- GPOLY
- H
- HPOLY
- K_Linear
- L
- R**
- R_war

Libraries:

- ABM
- ANALOG**
- ANALOG_P
- BREAKOUT
- Design Cache
- EVAL
- SOURCE
- SOURCSTM
- SPECIAL

Graphic:

- Normal
- Convert

Packaging:

Parts per Pkg: 1

Part:

Type: Homogeneous

OK

Cancel

Add Library...

Remove Library

Part Search...

Help

Ready

0 items selected Scale=100% X=10.40 Y=1.00

Start | http://www.elab... | Student_version... | Capture | New Folder | orcad | 10.GIF - Paint | OrCAD Capture ... | 12:40 PM

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



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

File Edit View Place Macro PSpice Accessories Options Window Help

R

Place Part

Part: VDC

Part List:

- ISFFM
- ISIN
- ISRC
- STIM1
- STIM16
- STIM4
- STIM8
- VDC
- VFXP

Libraries:

- ABM
- ANALOG
- ANALOG_P
- BREAKOUT
- Design Cache
- EVAL
- SOURCE
- SOURCSTM
- SPECIAL

Graphic:

Normal

Convert

Packaging:

Parts per Pkg: 1

Part: [v]

Type: Homogeneous

0Vdc

V?

OK

Cancel

Add Library...

Remove Library

Part Search...

Help

Title <Title>

Size	Document Number	Rev
A	<Doc>	<Rev Code>

Ready

0 items selected Scale=100% X=10.80 Y=0.70

Start http://www.elab... Student_version... Capture New Folder orcad 12.GIF - Paint OrCAD Capture ... 12:43 PM

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



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

File Edit View Place Macro PSpice Accessories Options Window Help

VDC

Place Ground

Symbol:

GND

GND
GND_EARTH
GND_FIELD SIGNAL
GND_POWER
GND_SIGNAL

Libraries:

CAPSYM
Design Cache

Name:

GND

OK
Cancel
Add Library...
Remove Library
Help

Να μην χρησιμοποιηθεί αυτό το στοιχείο!

Title <Title>

Size	Document Number	Rev
A	<Doc>	<Rev Code>

Ready 0 items selected Scale=100% X=10.90 Y=2.00

Start http://www.elab... Student_version... Capture New Folder orcad 13.GIF - Paint OrCAD Capture ... 12:44 PM

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



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

File Edit View Place Macro PSpice Accessories Options Window Help

Place Ground

Symbol: GND

OK Cancel Add Lib Remove Lib Help

Libraries: CAPSYM Design Cache

Name: GND

Browse File

Look in: Pspice

- abm.olb
- analog.olb
- analog_p.olb
- breakout.olb
- eval.olb
- source.olb
- sourcstm.olb
- special.olb

File name: source.olb

Files of type: Capture Library (*.olb)

Open Cancel

Open as read-only

Ready

0 items selected Scale=100% X=10.90 Y=2.80

Start http://www.elab... Student_version... Capture New Folder orcad 15.GIF - Paint OrCAD Capture ... 12:48 PM

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



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

File Edit View Place Macro PSpice Accessories Options Window Help

0

Place Ground

Symbol:
0

\$D_HI
\$D_LO
0

Libraries:
CAPSYM
Design Cache
source

Name:
0

OK
Cancel
Add Library...
Remove Library
Help

R1

V1

0Vdc

Title	<Title>	
Size	Document Number	Rev
A	<Doc>	<Rev Code>

Start | http://www.elab... | Student_version... | Capture | New Folder | orcad | 15.GIF - Paint | OrCAD Capture ... | 12:49 PM

ΕΙΣΑΓΩΓΗ ΣΤΟΙΧΕΙΩΝ



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

File Edit View Place Macro PSpice Accessories Options Window Help

GND

0Vdc V1

R1 1k

R2 1k

0 items selected Scale=200% X=5,90 Y=1.60

Start http://www.el... Student_versi... Capture New Folder orcad pspicee 17b.GIF - Paint OrCAD Captu... 1:19 PM

ΕΙΣΑΓΩΓΗ ΚΑΛΩΔΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface for a schematic diagram. The circuit includes a 0Vdc voltage source (V1), two resistors (R1 and R2), and two ground symbols (0). Resistor R1 is labeled with a value of 1k. A red arrow points to the 'Wire' tool in the right-hand toolbar. The status bar at the bottom indicates '0 items selected', 'Scale=200%', and coordinates 'X=5.20 Y=1.40'. The Windows taskbar at the very bottom shows the Start button and several open applications, including a browser, file explorer, and OrCAD Capture.

ΕΠΕΞΕΡΓΑΣΙΑ ΤΙΜΩΝ



The screenshot shows the OrCAD Capture interface with a circuit diagram and a 'Display Properties' dialog box. The circuit consists of a DC voltage source V1 (10Vdc) connected in series with a resistor R1 (1k), which is then connected to another resistor R2 (1k) that is grounded. The 'Display Properties' dialog box is open, showing the 'Value' field set to '10Vdc'. A red arrow points to the '10Vdc' label in the circuit, and a green arrow points to the 'Value' field in the dialog box. The dialog box also shows options for 'Display Format' (Value Only is selected), 'Color' (Default), and 'Rotation' (0° is selected).

OrCAD Capture - [/ - {SCHEMATIC1 : PAGE1}]

File Edit View Place Macro PSpice Accessories Options Window Help

0

Display Properties

Name: DC

Value: 10Vdc

Display Format

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

Font: Arial 7 (default)

Color: Default

Rotation: 0° 90° 180° 270°

OK Cancel Help

1 item selected Scale=200% X=4.30 Y=1.70

Start http://www.elab... Student_version_... Capture New Folder orcad 18.GIF - Paint OrCAD Capture ... 12:55 PM

ΕΠΕΞΕΡΓΑΣΙΑ ΤΙΜΩΝ



The screenshot displays the OrCAD Capture interface. The main workspace shows a circuit diagram with a 10Vdc voltage source (V1) connected to two resistors, R1 and R2. Resistor R1 is highlighted with a red dashed box and a red arrow pointing to it. Resistor R2 is labeled with a value of 1k. The Property Editor window is open at the bottom, showing the properties for the selected component R1. The Value field is highlighted with a green arrow and contains the text "0.5k".

Reference	Source Library	Source Package	TOLERANCE	Value
SCHEMATIC1 : PAGE1 : R1	C:\PROGRAM FILES\ORCAD_DEMO\CAPTURE\LIBRARY\ANALOG	R		0.5k

ΠΙΝΑΚΑΣ ΕΛΕΓΧΟΥ



OrCAD Capture

File Design Edit View Tools PSpice Accessories Options Window Help

test1.opj / - (SCHEMATIC1 : PAGE1)

10Vdc V1 R1 0.5k R2 1k

0 0

Display the project manager for this window.

Start <http://www.elab...> Student_version_... Capture New Folder orcad 20.GIF - Paint OrCAD Capture 12:59 PM

ΟΝΟΜΑΣΙΑ ΚΟΜΒΩΝ



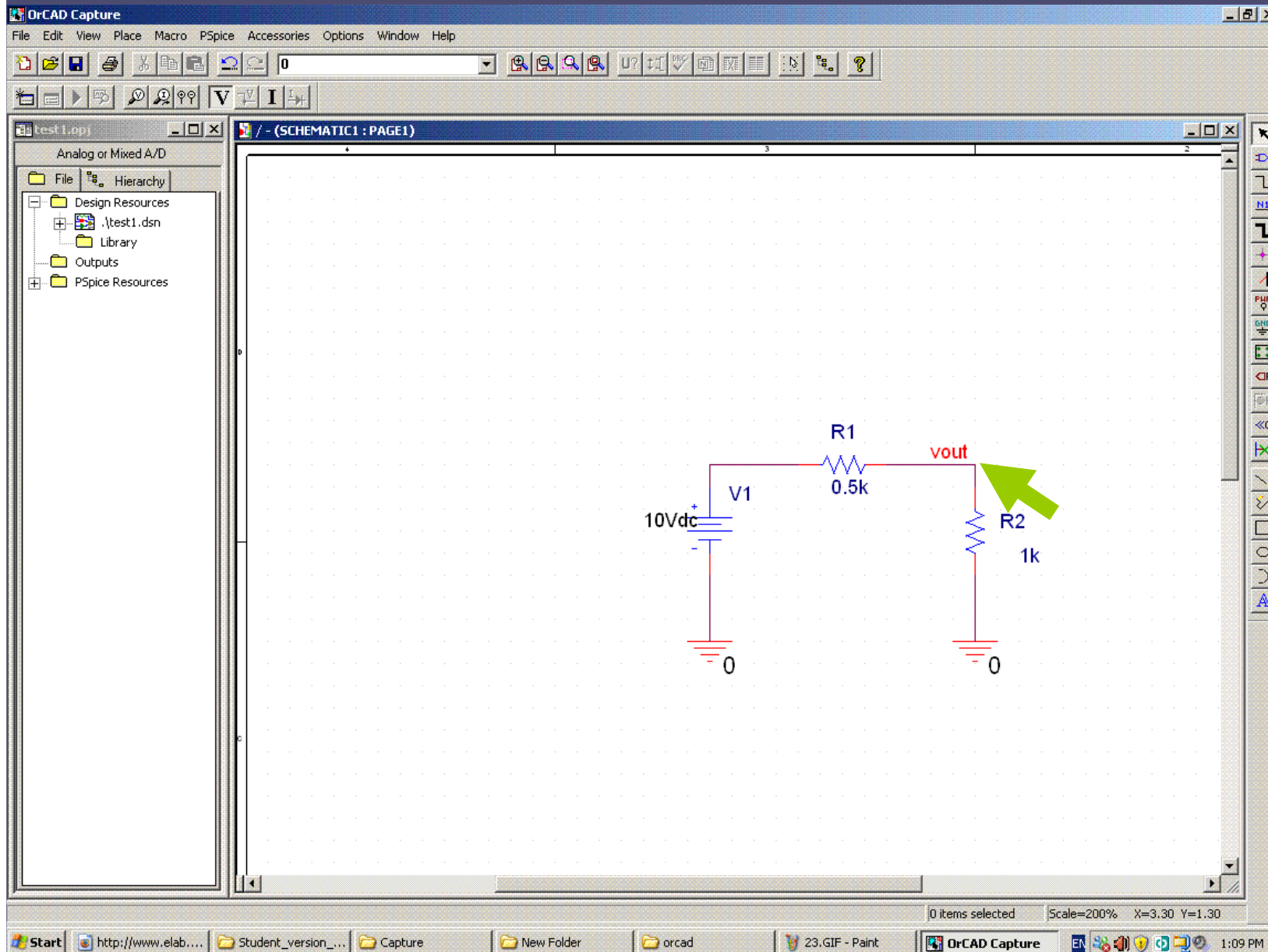
The screenshot displays the OrCAD Capture interface. The 'Place' menu is open, with a green arrow pointing to the 'Part...' option. The circuit diagram in the center shows a 10Vdc voltage source (V1) connected in series with a 0.5k resistor (R1), which is then connected to a 1k resistor (R2) that is grounded. The status bar at the bottom indicates 'Place a net alias.' and '0 items selected'. The Windows taskbar at the very bottom shows the Start button and several open applications, including OrCAD Capture.

ΟΝΟΜΑΣΙΑ ΚΟΜΒΩΝ



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with a 10Vdc voltage source (V1) connected to a 0.5k resistor (R1), which is in series with a 1k resistor (R2). Both resistors are connected to ground (0). A 'Place Net Alias' dialog box is open, with the text 'vout' entered in the 'Alias:' field. A green arrow points to this field. The dialog box also shows options for 'Color' (Default), 'Rotation' (0°, 90°, 180°, 270°), and 'Font' (Arial 7 (default)). The status bar at the bottom indicates 'Ready', '0 items selected', 'Scale=200%', 'X=6.80 Y=0.30', and the system tray shows the Start button, a URL, and the time '1:03 PM'.

ΟΝΟΜΑΣΙΑ ΚΟΜΒΩΝ



ΔΗΜΙΟΥΡΓΙΑ ΑΡΧΕΙΟΥ ΠΡΟΣΟΜΙΩΣΗΣ



OrCAD Capture

File Edit View Place Macro PSPICE Accessories Options Window Help

test1.opj

Analog or Mixed A/D

- File
- Hierarchy
- Design Resources
 - .test1.dsn
- Library
- Outputs
- PSpice Resources

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

10Vdc V1
R1 0.5k
vout
R2 1k
0 0

0 items selected Scale=200% X=2.90 Y=0.20

Start http://www.elab... Student_version... Capture New Folder orcad 24.GIF - Paint OrCAD Capture 1:09 PM

ΕΛΕΓΧΟΣ ΑΡΧΕΙΟΥ ΠΡΟΣΟΜΟΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface. The main workspace shows a schematic diagram of a circuit. The circuit consists of a 10Vdc voltage source (V1) connected in series with a 0.5k resistor (R1). The output voltage across R1 is labeled 'vout'. This output is connected to a 1k resistor (R2) which is connected to ground. The schematic is on a grid background.

On the left side, there is a file explorer window titled 'test1.opj'. It shows a hierarchy of folders: 'Design Resources', 'Outputs', and 'PSpice Resources'. A green arrow points to the file '.test1-schematic1.net' under the 'Outputs' folder.

At the bottom, a command window titled 'test1-schematic1.net' displays the netlist for the simulation:

```
1: source TEST1
2: R_R1      N00060 VOUT 0.5k
3: V_V1      N00060 0 10Vdc
4: R_R2      VOUT 0 1k
5:
```

The status bar at the bottom indicates 'Ready', 'Line 1, Col 1', and 'INS'. The Windows taskbar at the very bottom shows the Start button, a URL, Windows Explorer, Paint, and OrCAD Capture, along with the system clock showing 1:23 PM.

ΣΤΗΣΙΜΟ ΠΡΟΣΟΜΟΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface. The main workspace shows a circuit diagram with a 10Vdc source (V1), a 0.5k resistor (R1), and a 1k resistor (R2). The output voltage is labeled vout. The 'New Simulation Profile' menu is open, and a green arrow points to it. The status bar at the bottom indicates 'Create a new simulation profile.' and 'Scale=200% X=3.70 Y=0.20'.

ΣΤΗΣΙΜΟ ΠΡΟΣΟΜΟΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with two resistors, R1 (0.5k) and R2 (1k), connected in series between a ground symbol and a node labeled 'vout'. The 'vout' node is connected to the output of R1. A 'New Simulation' dialog box is open, with 'SIM_TEST1' entered in the Name field. The dialog box has a green arrow pointing to the 'Name' field. The dialog box also shows 'Inherit From: none' and 'Root Schematic: SCHEMATIC1'. The taskbar at the bottom shows the Start button, a web browser, Windows Explorer, Paint, and OrCAD Capture. The system clock shows 1:25 PM.

ΣΤΗΣΙΜΟ ΠΡΟΣΟΜΟΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with a 1k resistor labeled R2 connected to ground. A red wire labeled 'vout' is connected to the top terminal of the resistor. A 'Simulation Settings - SIM_TEST1' dialog box is open, showing the following configuration:

- Simulation Profile: SIM_TEST1
- Input:
 - Project Name (.OPJ): C:\Program Files\OrCAD_Demo\test1.opj
 - Schematic filename (.DSN): C:\Program Files\OrCAD_Demo\test1.dsn
 - Schematic name: SCHEMATIC1
- Output:
 - Output filename: C:\Program Files\OrCAD_Demo\test1-SCHEMATIC1-SI
 - Waveform data filename: C:\Program Files\OrCAD_Demo\test1-SCHEMATIC1-SI
- Notes: (Empty text area)

The dialog box has buttons for OK, Cancel, Apply, and Help. The background schematic shows a grid with a ground symbol at the bottom center and another ground symbol at the bottom right, connected to the resistor R2.

ΣΤΗΣΙΜΟ ΠΡΟΣΟΜΟΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with a resistor labeled R2 (1k) connected to ground. A red wire labeled vout is connected to the top terminal of the resistor. A simulation settings dialog box titled "Simulation Settings - SIM_TEST1" is open, showing the following configuration:

- Analysis type: Time Domain (Transient)
- Run to time: 1000ns seconds (TSTOP)
- Start saving data after: 0 seconds
- Options: General Settings (checked), Monte Carlo/Worst Case (unchecked), Parametric Sweep (unchecked), Temperature (Sweep) (unchecked), Save Bias Point (unchecked), Load Bias Point (unchecked)
- Transient options: Maximum step size (empty), Skip the initial transient bias point calculation (SKIPBP) (unchecked)
- Buttons: OK, Cancel, Apply, Help

The Windows taskbar at the bottom shows the Start button, a URL, Windows Explorer, Paint, OrCAD Capture, and Simulation Settings. The system clock indicates 1:29 PM.

ΣΤΗΣΙΜΟ ΠΡΟΣΟΜΟΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with a resistor labeled R2 (1k) connected to a ground symbol. A red wire labeled vout is connected to the top terminal of the resistor. A 'Simulation Settings - SIM_TEST1' dialog box is open, showing the 'Options' tab. The dialog box contains the following settings:

Parameter	Value	Units	Option
Relative accuracy of V's and I's:	0.001		(RELTOL)
Best accuracy of voltages:	1.0u	volts	(VNTOL)
Best accuracy of currents:	1.0p	amps	(ABSTOL)
Best accuracy of charges:	0.01p	coulombs	(CHGTOL)
Minimum conductance for any branch:	1.0E-12	1/ohm	(GMIN)
DC and bias "blind" iteration limit:	150		(ITL1)
DC and bias "best guess" iteration limit:	20		(ITL2)
Transient time point iteration limit:	10		(ITL4)
Default nominal temperature:	27.0	°C	(TNOM)

Additional options in the dialog box include:

- Use GMIN stepping to improve convergence. (STEPGMIN)
- Use reordering to reduce matrix fill-in. (PREORDER)

Buttons at the bottom of the dialog box include: MOSFET Options..., Advanced Options..., Reset, OK, Cancel, Apply, and Help.

ΣΤΗΣΙΜΟ ΠΡΟΣΟΜΟΙΩΣΗΣ



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with a resistor labeled R2 (1k) connected to ground. A red wire labeled vout is connected to the top terminal of the resistor. A dialog box titled "Simulation Settings - SIM_TEST1" is open, showing the "Schematic/Circuit Data" tab. The "All voltages, currents, and digital states" option is selected. The "Save data in the CSDF format (.CSD)" checkbox is unchecked. The taskbar at the bottom shows the Start button, a web browser, Windows Explorer, Paint, OrCAD Capture, and the Simulation Settings dialog box. The system clock indicates 1:30 PM.



DC Ανάλυση

Simulation Settings - SIM_TEST1

General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window

Analysis type:

- Bias Point
- Time Domain (Transient)
- DC Sweep
- AC Sweep/Noise
- Bias Point
- Temperature [Sweep]
- Save Bias Point
- Load Bias Point

Output File Options:

- Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)
- Perform Sensitivity analysis (.SENS)
- Calculate small-signal DC gain (.TF)

Output variable(s):

From Input source name:

To Output variable:

OK Cancel Apply Help

vout

R2
1k



DC Ανάλυση

The screenshot displays the OrCAD Capture interface. The main workspace shows a schematic diagram of a circuit with a 10Vdc source (V1), a 0.5k resistor (R1), and a 1k resistor (R2). The output voltage is labeled 'vout'. A simulation window titled 'SCHEMATIC1-SIM_TEST1 - OrCAD PSpice A/D Demo' is open, showing a status bar with the following text: 'No recognized product configur... Profile: "SCHEMATIC1-SIM... Reading and checking circuit... Circuit read in and checked, no... Calculating bias point... Bias point calculated... Simulation complete'. The status bar also indicates 'Position cursor: at slope' and '100%'. The Windows taskbar at the bottom shows the Start button, a URL 'http://www.elab.ntua.gr...', '5 Windows Explorer', '33.GIF - Paint', 'OrCAD Capture', and 'SCHEMATIC1-SIM_TE...'. The system clock shows '1:41 PM'.

DC Ανάλυση



**** 11/01/08 13:40:09 ***** Evaluation PSpice (Nov 1999) *****

** Profile: "SCHEMATIC1-SIM_TEST1" [C:\Program Files\OrCAD_Demo\test1-SCHEMATIC1-SIM_TEST1.sim]

**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C

NODE	VOLTAGE	NODE	VOLTAGE	NODE	VOLTAGE	NODE	VOLTAGE
(VOUT)	6.6667	(N00060)	10.0000				

VOLTAGE SOURCE CURRENTS

NAME	CURRENT
V_V1	-6.667E-03

TOTAL POWER DISSIPATION 6.67E-02 WATTS

JOB CONCLUDED

TOTAL JOB TIME .08

No recognized product configuration selected.
** Profile: "SCHEMATIC1-SIM_TEST1" [C:\Program Files\OrCAD_Demo\test1-SCH
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point
Bias point calculated
Simulation complete

Analysis / Watch / Devices /

For Help, press F1

100%

Start | http://www.elab.ntua.gr... | 5 Windows Explorer | 34.GIF - Paint | OrCAD Capture | SCHEMATIC1-SIM_TE... | 1:43 PM

DC Ανάλυση



The screenshot displays the OrCAD Capture interface for a DC analysis simulation. The circuit diagram shows a 10V DC source (V1) connected in series with a 0.5k resistor (R1). This series combination is connected to a 1k resistor (R2) which is grounded. The output voltage (vout) across R2 is shown as 6.667V. The software interface includes a menu bar (File, Design, Edit, View, Tools, PSpice, Accessories, Options, Window, Help), a toolbar, a project browser on the left, and a status bar at the bottom.

DC Ανάλυση



OrCAD Capture

File Design Edit View Tools PSpice Accessories Options Window Help

test1.opj

Design Resources

- .test1.dsn
 - SCHMATIC1
 - PAGE1

Library

Outputs

PSpice Resources

- Include Files
- Model Libraries
- Simulation Profiles
- SCHMATIC1-SIM
- Stimulus Files

10Vdc V1 6.667mA 0.5k R1 vout 6.667mA R2 1k

Ready 0 items selected

Start http://www.elab.ntua.gr... 5 Windows Explorer 36.GIF - Paint OrCAD Capture SCHEMATIC1-SIM_TEST... 1:46 PM



ΠΑΡΑΜΕΤΡΙΚΗ Ανάλυση

The image shows the OrCAD Capture software interface. The main window displays a schematic diagram of a circuit with a 10Vdc source (V1), a 0.5k resistor (R1), and a 1k resistor (R2). The output voltage is labeled as vout. The current through R1 is 6.667mA, and the current through R2 is also 6.667mA. A simulation settings dialog box is open, showing the analysis type set to DC Sweep. The sweep variable is V1, and the sweep type is Linear. The sweep range is from 0 to 10Vdc with an increment of 1Vdc. The dialog box also shows the sweep type options: Linear, Logarithmic, and Value list. The Linear option is selected, and the sweep range is set to 0 to 10Vdc with an increment of 1Vdc. The dialog box also shows the sweep type options: Linear, Logarithmic, and Value list. The Linear option is selected, and the sweep range is set to 0 to 10Vdc with an increment of 1Vdc.

Simulation Settings - SIM_TEST1

Analysis type: DC Sweep

Sweep variable: Voltage source Name: V1

Sweep type: Linear

Start value: 0 End value: 10Vdc Increment: 1Vdc



ΠΑΡΑΜΕΤΡΙΚΗ Ανάλυση

Simulation Settings - SIM_TEST1

General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window

Analysis type: DC Sweep

Sweep variable: Voltage source Name: V1

Options: Primary Sweep

Sweep type: Linear Start value: 1 End value: 10 Increment: 1

6.667mA R2 1k

Ready 0 items selected

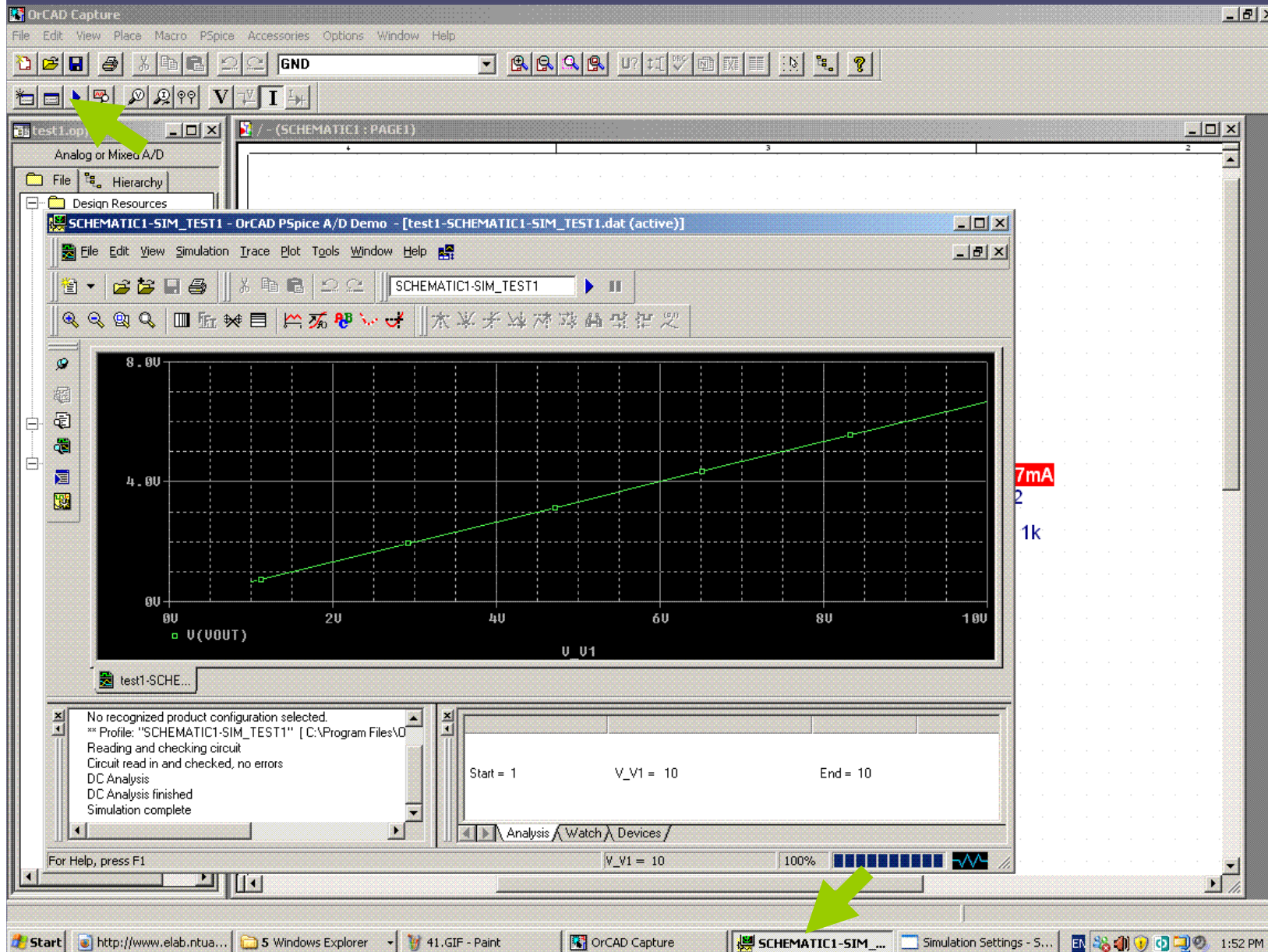
Start http://www.elab.ntua... 5 Windows Explorer 38.GIF - Paint OrCAD Capture SCHEMATIC1-SIM_TE... Simulation Settings... 1:49 PM

ΠΑΡΑΜΕΤΡΙΚΗ Ανάλυση



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with a 10Vdc voltage source (V1) connected to a 0.5k resistor (R1). The current through R1 is 6.667mA. R1 is connected to a node labeled 'vout', which is also connected to a 1k resistor (R2). The current through R2 is 6.667mA. A voltmeter (V) is connected across R2, showing a reading of 6.667mA. The status bar at the bottom indicates '0 items selected', 'Scale=200%', 'X=5.40', and 'Y=1.80'. The taskbar at the bottom shows the Start button and several open applications, including 'http://www.elab.ntua...', '5 Windows Explorer', '40.GIF - Paint', 'OrCAD Capture', 'SCHEMATIC1-SIM_TE...', and 'Simulation Settings - S...'. The system clock shows '1:51 PM'.

ΠΑΡΑΜΕΤΡΙΚΗ Ανάλυση



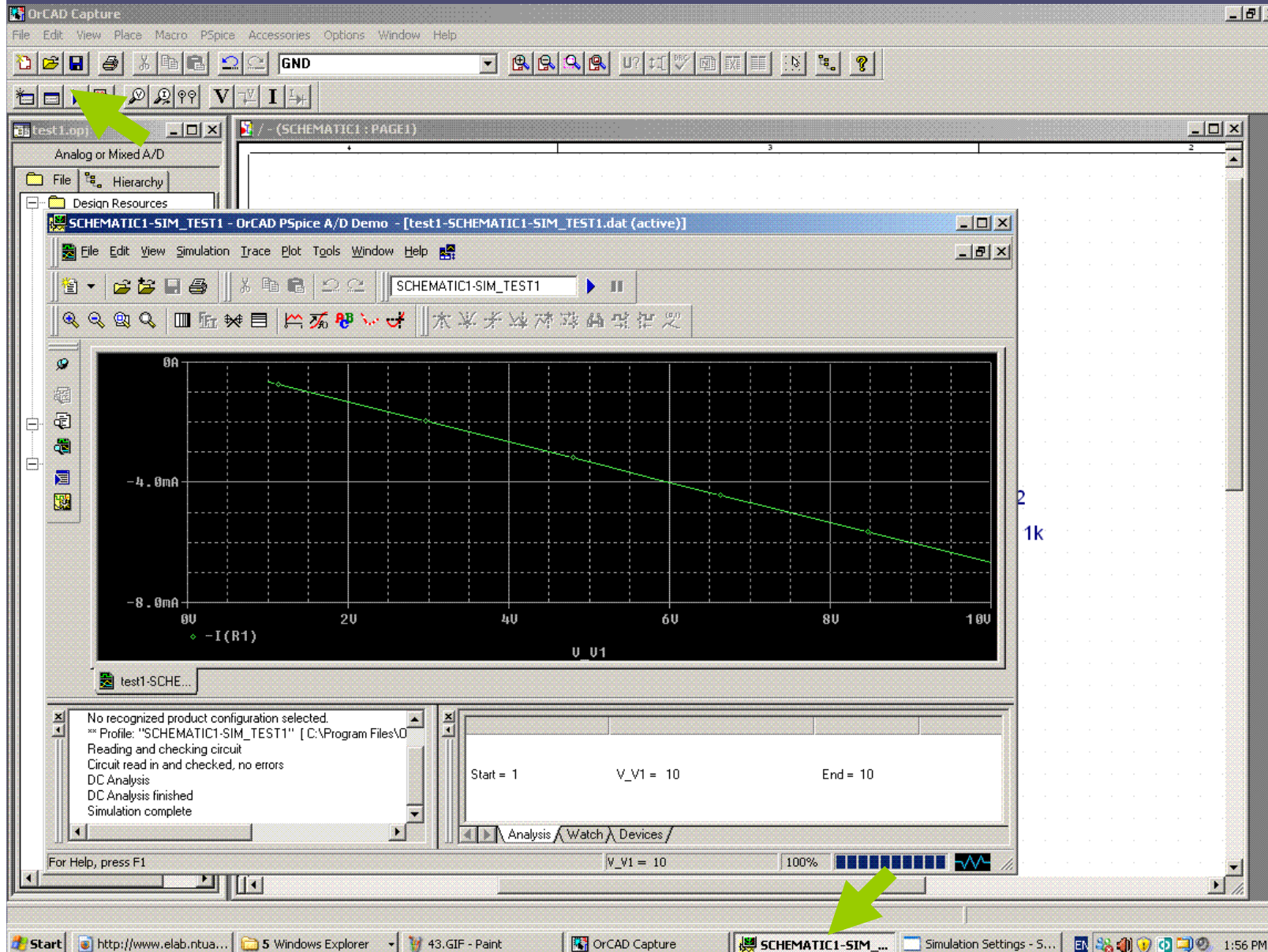


ΠΑΡΑΜΕΤΡΙΚΗ Ανάλυση

The screenshot displays the OrCAD Capture interface. The main workspace shows a circuit schematic with a 10Vdc voltage source (V1) connected in series with a resistor R1 (0.5k). This series combination is connected to a load resistor R2 (1k). The output voltage across R2 is labeled as vout. A green arrow points from the 'P' icon in the top toolbar to the schematic, indicating the parametric analysis tool. The left sidebar shows the project hierarchy for 'test1.opj', including Design Resources, Library, Outputs, and PSpice Resources. The bottom status bar indicates '0 items selected', 'Scale=200%', and coordinates 'X=3.30 Y=1.70'. The Windows taskbar at the bottom shows the Start button, a web browser, Windows Explorer, Paint, OrCAD Capture, and other background applications.



ΠΑΡΑΜΕΤΡΙΚΗ Ανάλυση



TRANSIENT Ανάλυση



The screenshot displays the OrCAD Capture interface. The main window shows a schematic diagram with a voltage source labeled 'vout' and a resistor labeled 'R2 1k'. A green arrow points to the 'Place Part' dialog box, which is open and shows a list of parts including 'VSIN'. The dialog box also displays parameters for the selected part: 'VOFF =', 'VAMPL =', and 'FREQ ='. The 'Place Part' dialog box has the following fields and options:

- Part: VSIN
- Part List: VPWL, VPWL_ENH, VPWL_F_RE_FOREVER, VPWL_F_RE_N_TIMES, VPWL_FILE, VPWL_RE_FOREVER, VPWL_RE_N_TIMES, VSFFM, VSIN, VSRC
- Libraries: ABM, ANALOG, ANALOG_P, BREAKOUT, Design Cache, EVAL, SOURCE, SOURCSTM, SPECIAL
- Graphic: Normal, Convert
- Packaging: Parts per Pkg: 1
- Type: Homogeneous

The schematic diagram shows a circuit with a voltage source 'vout' and a resistor 'R2 1k'. The voltage source is connected to ground, and the resistor is connected to ground. The voltage source is represented by a sine wave symbol with a 'V?' label. The resistor is represented by a zigzag line with 'R2' and '1k' labels. The circuit is connected to ground symbols.



TRANSIENT Ανάλυση

The screenshot shows the DrCAD Capture software interface. The main window displays a circuit schematic with a voltage source labeled V2 and a resistor labeled R2 (1k). The output voltage is labeled vout. A 'Display Properties' dialog box is open, showing the name 'VOFF' and a value of '0'. The dialog box has several options for display format and rotation. A green arrow points to the 'VAMP' parameter in the dialog box.

Display Properties dialog box fields:

- Name: VOFF
- Value: 0
- Font: Arial 7 (default)
- Color: Default
- Rotation: 0° (selected), 90°, 180°, 270°
- Display Format options:
 - Do Not Display
 - Value Only
 - Name and Value
 - Name Only
 - Both if Value Exists

Labels in the schematic:

- V2 (Voltage source)
- R2 1k (Resistor)
- vout (Output voltage)

Labels in the dialog box:

- VAMP =
- FREQ =

TRANSIENT Ανάλυση



The screenshot displays the OrCAD Capture interface. The main workspace shows a circuit schematic on a grid. On the left, a hierarchical tree view shows the project structure: Design Resources, .test1.dsn, SCHEMATIC1, PAGE1, Design Cache, Library, Outputs, and PSpice Resources. The circuit schematic includes an AC voltage source labeled V2 with parameters: $V_{OFF} = 0$, $V_{AMPL} = 1$, and $FREQ = 1k$. The source is connected to a resistor labeled R2 with a value of 1k. The output voltage across the resistor is labeled vout. The circuit is connected to ground. The status bar at the bottom indicates 'Ready', '0 items selected', 'Scale=200%', 'X=2.80 Y=0.80', and the system tray shows the time as 1:59 PM.



TRANSIENT Ανάλυση

DrCAD Capture

File Edit View Place Macro PSpice Accessories Options Window Help

VSIN

test1_top] / - (SCHEMATIC1 : PAGE1)

Analog or Mixed A/D

File Hierarchy

Design Resources

Place Part

Part: C

Part List:

- C
- C_elect
- C_var
- E
- EPOLY
- F
- FPOLY
- G
- GPOLY
- H

Libraries:

- ABM
- ANALOG
- ANALOG_P
- BREAKOUT
- Design Cache
- EVAL
- SOURCE
- SOURCSTM
- SPECIAL

Graphic

Normal

Convert

Packaging

Parts per Pkg: 1

Part: []

Type: Homogeneous

C?

1n

V2

V2

VOFF = 0

/AMPL = 1

FREQ = 1k

vout

R2

1k

Ready

0 items selected Scale=200% X=6.80 Y=0.40

Start http://www.elab.ntua... 5 Windows Explorer 47.GIF - Paint OrCAD Capture SCHEMATIC1-SIM_TE... Simulation Settings - S... 2:00 PM

TRANSIENT Ανάλυση



The screenshot shows the OrCAD Capture interface. On the left, a file hierarchy tree is visible, including folders for Design Resources, Design Cache, Library, and Outputs. The main workspace displays a circuit schematic with an AC voltage source (V2), a capacitor (C1), and a resistor (R2, 1k). The output voltage is labeled 'vout'. A 'Display Properties' dialog box is open, showing settings for the selected component. The dialog includes fields for Name (Value), Value (10), Font (Arial 7), Color (Default), and Display Format (Value Only selected). Simulation parameters are listed on the left: VOFF = 0, VAMPL = 1, and FREQ = 1k. The status bar at the bottom indicates '1 item selected', 'Scale=200%', 'X=5.10 Y=1.50', and the system tray shows the time as 2:01 PM.

TRANSIENT Ανάλυση



TRANSIENT Ανάλυση



Simulation Settings - SIM_TEST1

General Analysis Include Files Libraries Stimulus Options Data Collection Probe Window

Analysis type: Time Domain (Transient)

Run to time: 5m 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:

- Maximum step size: 10u seconds
- Skip the initial transient bias point calculation (SKIPBP)

Output File Options...

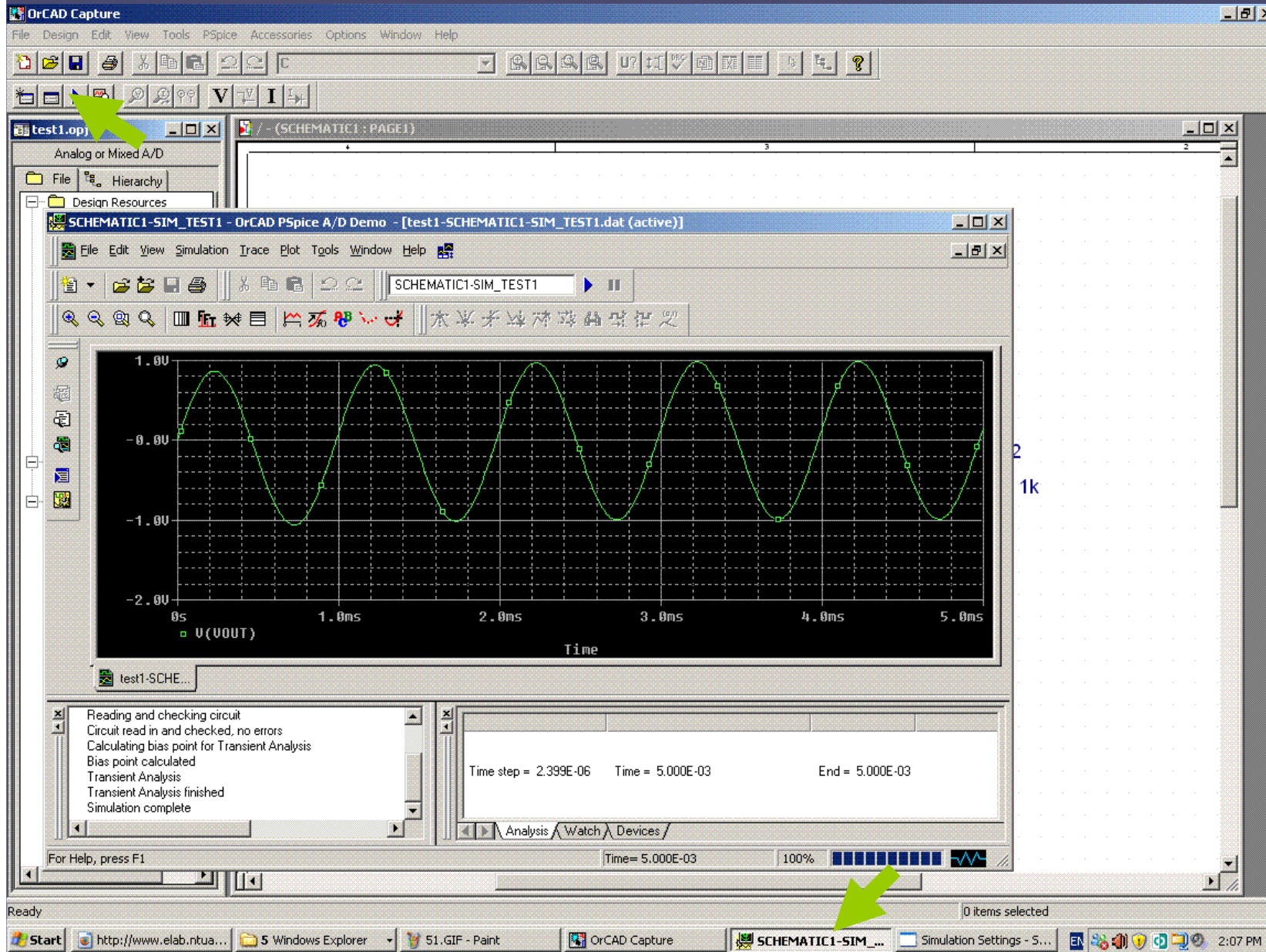
OK Cancel Apply Help

vout

R2

1k

TRANSIENT Ανάλυση



AC Ανάλυση



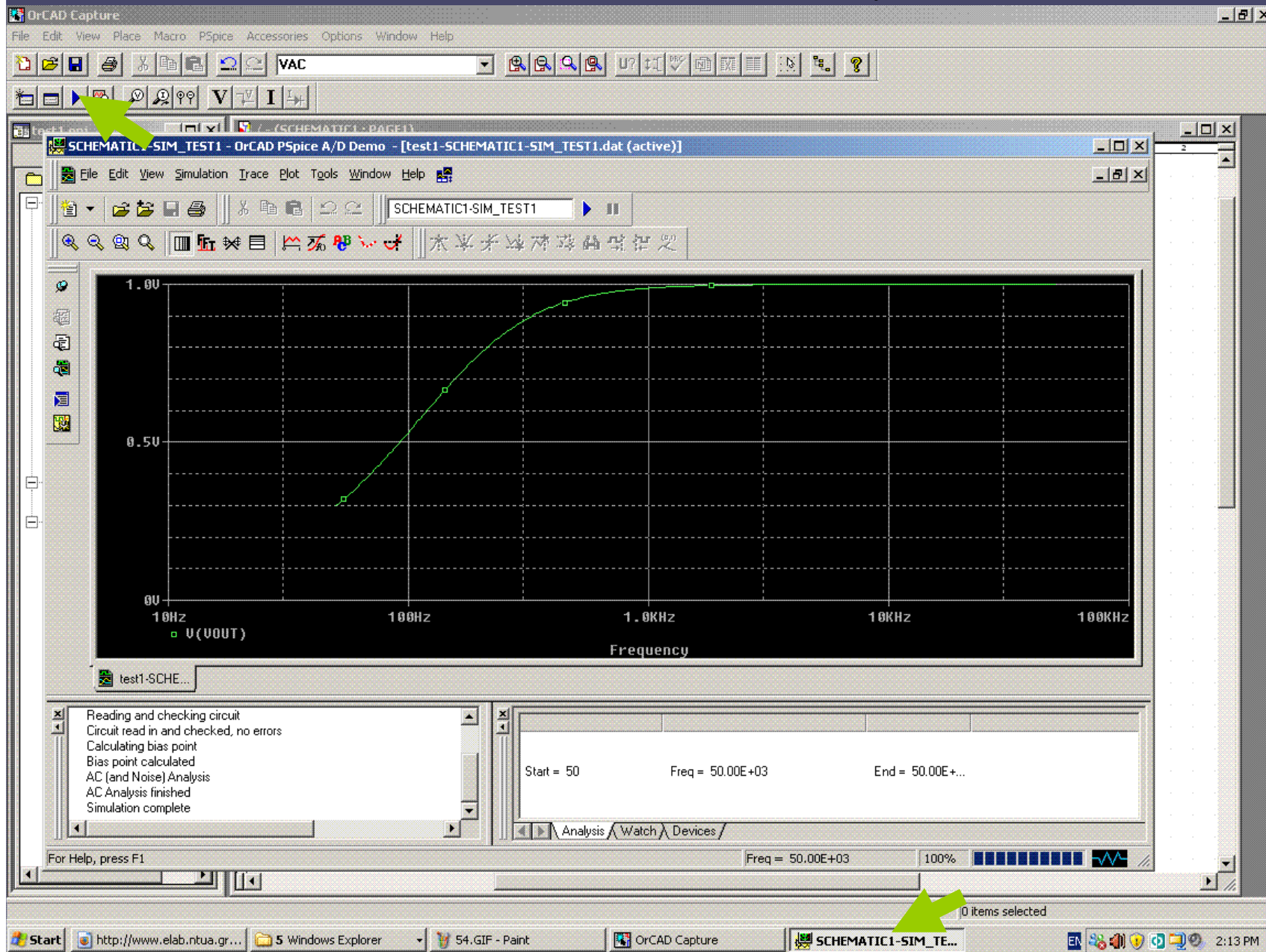
The screenshot displays the OrCAD Capture interface for an AC analysis. The main workspace shows a circuit diagram with a 1V AC voltage source (labeled '1Vac' and '0Vdc'), a capacitor C1 (1uF), and a resistor R2 (1k). The output voltage is labeled 'vout'. A green arrow points to the 'Place Part' dialog box, which is open and shows the 'VAC' part selected. The dialog box includes fields for 'Part:' (VAC), 'Part List:', and 'Libraries:'. The 'Part List' includes components like IPwL_RE_N_TIMES, ISFFM, ISIN, ISRC, STIM1, STIM16, STIM4, STIM8, VAC, and VDC. The 'Libraries' section lists various component libraries, with 'SOURCE' selected. The 'Graphic' section has 'Normal' selected, and the 'Packaging' section has 'Parts per Pkg: 1'. The 'Type' is 'Homogeneous'. The 'Place Part' dialog box also includes buttons for 'OK', 'Cancel', 'Add Library...', 'Remove Library', 'Part Search...', and 'Help'. The circuit diagram shows a 1V AC source connected to a 1uF capacitor (C1), which is connected to a 1k resistor (R2). The output voltage is labeled 'vout'. A green arrow points to the 'Place Part' dialog box, and a blue arrow points to the 'VAC' part in the 'Part List'.

AC Ανάλυση

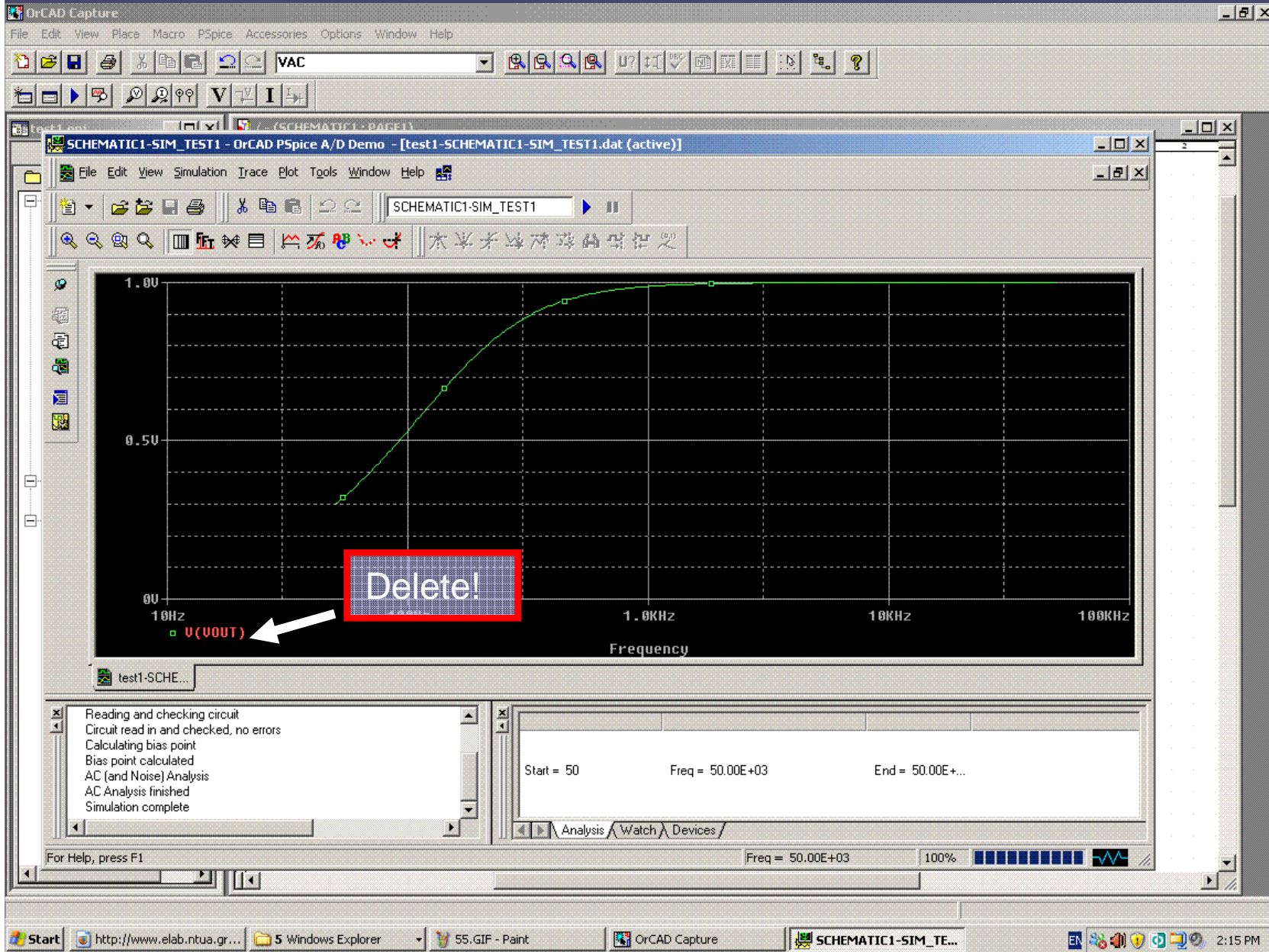


The screenshot displays the OrCAD Capture interface. On the left, a project tree shows the design hierarchy for 'test1.opj', including 'SCHEMATIC1' and 'PAGE1'. The main workspace shows a schematic diagram of a circuit with a 1k resistor labeled 'R2' and an output node 'vout'. A 'Simulation Settings - SIM_TEST1' dialog box is open, showing the 'Analysis' tab. The 'Analysis type' dropdown is set to 'AC Sweep/Noise', with a green arrow pointing to it. The 'AC Sweep Type' is set to 'Logarithmic' with a 'Decade' sweep rate. The 'Start Frequency' is 50, 'End Frequency' is 50K, and 'Points/Decade' is 100. The 'Noise Analysis' section is currently disabled. The 'Output File Options' section includes a checkbox for 'Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)'. The Windows taskbar at the bottom shows the Start button and several open applications, including 'http://www.elab.ntua...', '5 Windows Explorer', '53.GIF - Paint', 'OrCAD Capture', 'SCHEMATIC1-SIM_TE...', and 'Simulation Settings...'. The system clock shows 2:11 PM.

AC Ανάλυση



AC Ανάλυση



AC Ανάλυση



OrCAD Capture

File Edit View Place Macro PSpice Accessories Options Window Help

VAC

SCHEMATIC1-SIM_TEST1 - OrCAD PSpice A/D Demo - [test1-SCHEMATIC1-SIM_TEST1.dat (active)]

File Edit View Simulation Trace Plot Tools Window Help

SCHEMATIC1-SIM_TEST1

10Hz 30Hz 100Hz 300Hz 1.0kHz 3.0kHz 10kHz 30kHz 100kHz

Frequency

test1-SCHE...

Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point
Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

Start = 50 Freq = 50.00E+03 End = 50.00E+...

Analysis / Watch / Devices /

For Help, press F1 Freq = 50.00E+03 100%

0 items selected

Start http://www.elab.ntua.gr... 5 Windows Explorer 56.GIF - Paint OrCAD Capture SCHEMATIC1-SIM_TE... 2:15 PM

AC Ανάλυση



OrCAD Capture

File Edit View Place Macro PSpice Accessories Options Window Help

VAC

SCHEMATIC1-SIM_TEST1 - OrCAD Capture Demo - [test1-SCHEMATIC1-SIM_TEST1.dat (active)]

Trace Plot Tools Window Help

- Add Trace... Insert
- Delete All Traces Ctrl+Delete
- Undelete Traces Ctrl+U
- Fourier
- Performance Analysis...
- Cursor
- Macros...
- Goal Functions...
- Eval Goal Function...

10Hz 30Hz 100Hz 300Hz 1.0kHz 3.0kHz 10kHz 30kHz 100kHz

Frequency

test1-SCHE...

Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point
Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

Start = 50 Freq = 50.00E+03 End = 50.00E+...

Add trace[s] to the selected plot Freq = 50.00E+03 100%

Start http://www.elab.ntua.gr... 5 Windows Explorer 57.GIF - Paint OrCAD Capture SCHEMATIC1-SIM_TE... 2:16 PM

AC Ανάλυση



The screenshot displays the OrCAD Capture interface during an AC analysis simulation. The 'Add Traces' dialog box is open, allowing the user to select variables and functions for the simulation output. The 'Simulation Output Variables' list includes various voltage and current variables, with 'V(vout)' selected. The 'Functions or Macros' list includes mathematical functions, with 'LOG10()' selected. The 'Trace Expression' field shows the combination 'LOG10(V(vout))'. The background plot window shows a logarithmic scale for the AC analysis, with a frequency of 50.00E+03 Hz.

AC Ανάλυση



OrCAD Capture

File Edit View Place Macro PSpice Accessories Options Window Help

VAC

SCHEMATIC1-SIM_TEST1 - OrCAD PSpice A/D Demo - [test1-SCHEMATIC1-SIM_TEST1.dat (active)]

File Edit View Simulation Trace Plot Tools Window Help

SCHEMATIC1-SIM_TEST1

Add Traces

Simulation Output Variables

- Frequency
- I(C1)
- I(R2)
- I(V3)
- V(0)
- V(C1:1)
- V(C1:2)
- V(N00060)
- V(R2:1)
- V(R2:2)
- V(V3:~)
- V(V3:-)
- V(vout)
- V1(C1)
- V1(R2)
- V1(V3)
- V2(C1)

Analog
 Digital
 Voltages
 Currents
 Noise [V/Hz]
 Alias Names

Functions or Macros

Analog Operators and Functions

- ()
- *
- +
-
- /
- @
- ABS()
- ARCTAN()
- ATAN()
- AVG()
- AVGX(.)
- COS()
- D()
- DB()
- ENVMAX(.)
- ENVMIN(.)
- EXP()
- G()
- IMG()
- LOG()
- LOG10()
- M()
- MAX()
- MIN()

Trace Expression: $20 * \text{LOG}(V(\text{vout}))$

Start = 50 Freq = 50.00E+03 End = 50.00E+...

For Help, press F1 Freq = 50.00E+03 100%

Reading and checking circuit
Circuit read in and checked, no
Calculating bias point
Bias point calculated
AC (and Noise) Analysis
AC Analysis finished
Simulation complete

Start http://www.elab.ntua.gr... 5 Windows Explorer 59.GIF - Paint OrCAD Capture SCHEMATIC1-SIM_TE... 2:19 PM



AC Ανάλυση

